T·FLEX CAD 16

Fundamentals. Two-Dimensional Design

User Manual

TOP SYSTEMS

T-FLEX Parametric CAD

Fundamentals. Two-Dimensional Design

User Manual

«Top Systems» Moscow, 2018

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Introduction

Features and Area of Application

T-FLEX CAD is a parametric design and drawing system. T-FLEX provides high levels of drawing flexibility and supports modifications of the drawings while maintaining constraints imposed by the designer on the drawing elements. The unique parametric engine and a complete set of professional tools for computer-aided design simplify the design workflow and speed up preparation of drawing materials. T-FLEX CAD gives a designer a familiar feel of working with traditional paper and ruler equipment.

Associative design driven by assigning and modifying variable parameters is the way to follow by all design and drawing automation systems. The particular success of T-FLEX CAD is based, in the first place, on the new paradigm of geometric modeling. This paradigm is about a new, deeper, level of parameterization, compared to other systems. The idea of parameterization itself has nowadays become a standard in CAD. By "parameterization" we usually mean a provision for a drawing extensive reuse by means of modifying its parameters. Virtually all CAD vendors claim parametric capabilities of their systems. However, these systems, originally introduced long before parameterization was adopted, often use their legacy data structures that are inherently non-parametric. This causes their solutions to suffer from ineffectiveness or limited range of applicability. The T-FLEX CAD's revolutionary approach to the idea of parameterization and the fact that the drawings are based on inherently parametric models provide a new dimension for parametric design.

T-FLEX CAD uses concepts and practices that are familiar to designers. At the same time, the user does not need to care about making a precise drawing at once. The modification capabilities via both dimensioning and free dragging are unmatched across other CAD systems.

The assembly drawing environment is unique in its wide range of capabilities. T-FLEX CAD permits creating complex drawings where certain fragments can be bound by relations. A relation can be established either via geometrical properties or by parameters. The system correctly handles lines visibility throughout modifications, if some portions of the drawing overlap the others, with no limitation on the number of overlaps. It takes seconds to create drawings of a new product in a family by varying assembly drawing parameters. The modifications instantly reflect not only on the assembly but also on the member fragments (parts) and all the rest of the related documentation.

One typical attribute of the parametric CAD systems is a language for programming parametrical relations. T-FLEX CAD has another advantage in this area. The engineer is not required to have any training in programming. The drawing parameters can be represented by variables. These variables can be related in simple mathematical expressions. This is done without using any programming language. The variables can be assigned either at creation of an element or while editing an existing one. The values of the variables can be obtained from other drawings or automatically input from a database. This provides for unlimited modification capabilities in drawing.

Along with parametric design, T-FLEX CAD supports wide usage of quick drawing producing non-parametric sketches. This approach allows creating drawings in a way similar to major CAD systems by using a standard set of tools for drawing, i.e. various primitives, as arcs, circles, line segments, etc. The snapping mechanism is provided for easy sketching of new entities, such as horizontal and vertical alignment of the cursor with the existing entities or their ends, center-of-arc and center-of-circle snapping, etc. When creating arcs, snapping occurs around the 90, 180 and 270 degrees. The cursor also snaps to horizontal and vertical alignments with the arc center. The system automatically identifies multiple pairs of same-object snappings. Snapping to any object can be locked with the Function key, and the cursor will follow to the locked snapping condition. Thus, the sketcher provides a way of quick drawing, however, such drawings do not take full advantage of parametric dimension modifications. Therefore, this method is only recommended when no substantial modifications are expected on the drawing.

Creation of parametric construction-based drawings can be accelerated with the special parametric sketch mode. This mode combines efficiency of non-parametric drafting with flexibility of parametric construction. This goal is achieved by simultaneous actions of a user and the application: user creates his drawings using ordinary sketch features, and application "puts" geometrically related construction elements under the sketch lines thus producing a parametric drawing.

The highly effective functionalities of T-FLEX CAD make the system usable in a wide range of situations. The system can well be used in mechanical design, such as design of industrial equipment and tooling, development of molds and stamps, design of consumer goods, etc. It also supports development of manufacturing process flow charts and BOM, numerically controlled machining and other technological procedures. Other possible application fields include construction and architectural design, charting various types of graphs, dynamical visualization of processes and mechanisms, industrial and graphic design. The most effective uses of T-FLEX CAD occur when the parametric design paradigm dominates the design process, and when all stages of design are involved, from sketch to scratch drawing to production drawing. T-FLEX CAD facilitates considerable speed-up of graphic design and documentation cycle.

T-FLEX CAD offers a complete range of drawing tools, such as creating various-type line drawings, hatches, dimensions, text, roughnesses, special symbols, etc. Important that all these design attributes can be associated with the parameters of the drawing. This means, modifying a drawing parameter would cause adjustment of the design attributes. The drawings follow the user-specified international standard. T-FLEX CAD also supports instant switching from one drawing standard to another.

The three-dimensional version CAD 3D is intended for making parametric 3D models. The 3D solid bodies authored by the system can easily be modified. Parametric modifications of the 2D drawings propagate on the model's 3D representation, and vise versa.

T-FLEX CAD can be used as a base for developing specialized CAD systems. The system supports exporting parametric drawing data to custom processing modules. Vise versa, externally generated parameter values can be imported into the system and assigned to the drawing parameter variables. The model is then automatically regenerated, and the new design drawing is ready.

The system software utilizes the latest GUI standards. Even a novice user can easily start working with the system. The menu and icon layout is easy to use. The command dialog boxes are intuitive. The various drawing elements and the libraries of drawings allow effortless manipulation. The built-in context-dependent Help facilitates quick learning. Every command is realized in a way that provides users – engineers and designers - with confidence in operating the system.

The theory and algorithms used in the system are unique yet unambiguous to end-users.

CONVENTIONS ADOPTED IN THE T-FLEX CAD GUIDELINES

The following standard conventions are adopted in this document:

<Enter>, <L>, <Esc>, etc. – notations for the keys on the computer keyboard.

[OK], **[View]**, etc. – notations for graphic buttons in the dialog boxes.

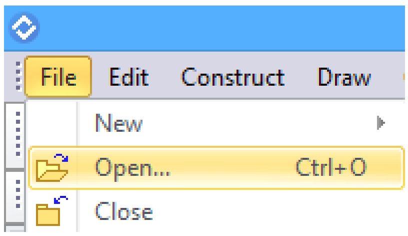
- Left mouse button click.

- Right mouse button click.

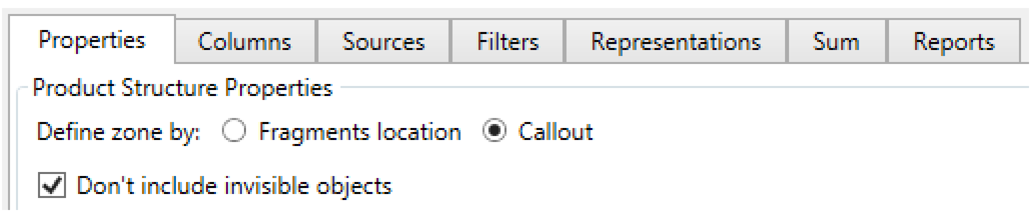
- Left mouse button double-click.

, , etc. – icons on a toolbar, ribbon or automenu.

File > Open... etc. – selection of a textual menu bar item File, followed by a pull-down menu item Open....



Properties > **Product structure properties** > **Don't include invisible objects**, etc means that you select **Don't include invisible objects** item in the **Product structure properties** group of the **Properties** tab in the dialogue window.



O: Open Model, **EL: Construct Ellipse**, etc. – names of T-FLEX CAD commands. Note that the character combinations before the colon define the keystroke accelerator sequences for invoking commands by typing in the status bar.

A command can be invoked in T-FLEX CAD by the following three ways:

By typing,

By selecting the toolbar item, and

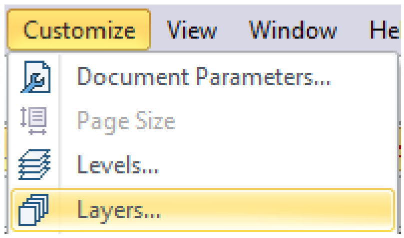
By selecting the textual menu/ribbon item.

The system manuals describe the commands in a table. For instance, the command **QL: Layers** would appear in a table as follows:

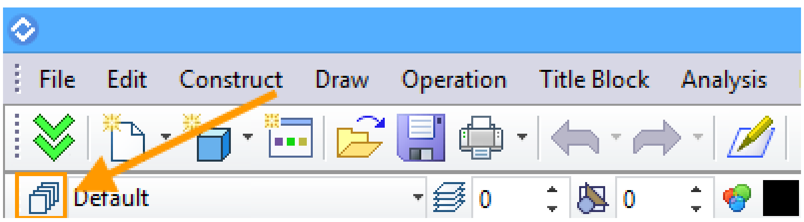
Icon	Ribbon
	Edit → Document → Layers…
Keyboard	Textual Menu
<QL>	Customize > Layers

This means, the command can be invoked in the following ways:

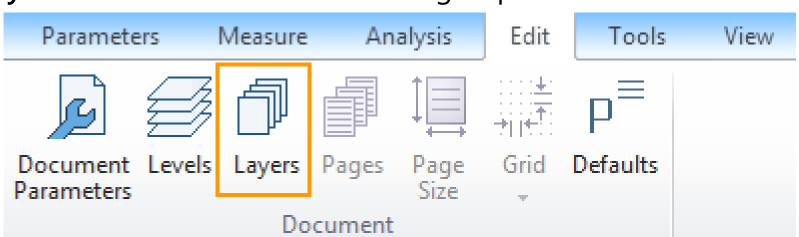
1. Press the key <Q> then <L> on the keyboard,
2. Select **Customize** item in the text menu and activate **Layers…** command from the drop-down list.



3. Select the icon in the appropriate toolbar.

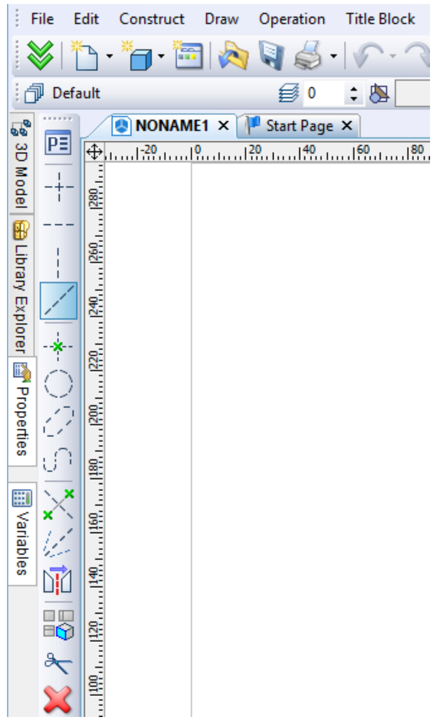


4. Select **Layers** command in the **Document** group of the **Edit** tab.



Certain most common commands can also be invoked with the function keys. For instance, pressing <F7> causes Redraw operation.

Select an element instruction in the manuals means placing the cursor over the element and pressing left mouse button or <Enter>.



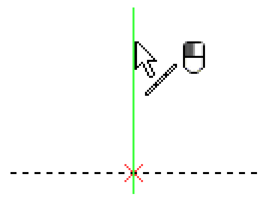
Select an icon, press an icon, select an input box, press a button instruction means placing the cursor over the item (icon, input box, dialog box button) and pressing left mouse button.

Point at an element, point at an icon, point at a button means just placing the cursor over the item.

Each command usually brings a list of options available under this command. An option is one specific action performed within the command, as delete an element, select an element of a particular type, switch to another mode, etc.

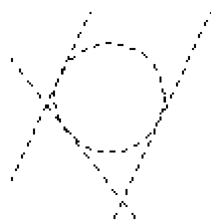
Each option is represented by a button and an icon in the automenu.

Invoking an option via the keystroke mechanism might be different than by selecting the icon. Typing the keystroke sequence instantly invokes the action, while selecting an icon may work in two ways. First possibility is – an instant action occurs, as, for instance, when specifying parameters of an element via .



Second – after selecting the icon, the system waits for a specific user action, with the cursor being modified with a glyph corresponding to the action. The action completes when the cursor is pointed at an appropriate element and left mouse button pressed. For instance, this can be a selection of a construction line – .

The command description contains various ways of creating elements. For instance, the following sequence describes creation of a construction circle:



The command description contains various ways of creating elements. For instance, the following sequence describes creation of a construction circle:

<L>, <L>, <L> - a circle tangent to three lines.

The above sequence uses a typical notation which implies that the respective automenu icon picks can be used instead of the keystrokes, for instance,

<L>, , <L> is a way of creating a three-line-tangent circle using both the keystrokes and the icon.

, , is a way of creating a three-line-tangent circle via the automenu icon picks.

, <L>, <L>, etc. – other possible combinations.

In the system description, "Press" usually means that either left mouse button or <Enter> key can be pressed. The <Enter> *key works as while working within the command dialog box.*

In the system description, "Press" means that either right mouse button or <Esc> key can be pressed. This convention about also holds when working in the drawing area of the application. Use of in other areas of the screen follows the standard conventions of Windows (usually, this invokes the context-sensitive menu).

System Fundamentals. Customization

Getting Started

This chapter contains sections helpful in getting started with the system setup and basic operation: "System Requirements", "T-FLEX CAD System Setup", "Basic Terms and Drawing Techniques", "Quick Reference on User Interface".

System Requirements

Hardware Requirements

Software Requirements	Windows XP (Service Pack 3), Windows Vista, Windows 7, Windows 8/ Windows 10
Minimum Processor: Recommended Processor:	32-bit or 64-bit Intel or AMD processor with SSE2 support Core i7 processor or equivalent
Videocard: Recommended videocard:	Videocard with OpenGL 3.3 support is recommended for efficient work with assemblies. OpenGL 4.2 – for improved visualization of models in 3D scene. OpenGL 1.0 allows to work with 3D models, but with restrictions (threads, welds, graphic sections and textures will not display). Videocard supporting CUDA 2.3 or higher is required for creation of photorealistic images with NVIDIA Optix technology. High-performance NVIDIA videocard with 1GB of memory or higher and with Open GL 4.2 support or higher
Minimum RAM: Recommended RAM:	1G 4G or more* (for very large assemblies)

* 32-bit operating systems Microsoft Windows have a limitation of 4GB of memory address space. This 4GB space is evenly divided into two parts, with 2GB dedicated for kernel usage, and 2GB left for application usage. Each application (including T-FLEX CAD) gets its own 2GB, but all applications have to share the same 2GB kernel space. For Windows XP and Windows Vista it is possible to increase the default allocation capabilities up to 3GB (3GB for user mode, 1GB reserved for kernel). Such capability requires additional tunings in order to be effective (see http://support.microsoft.com for more information).

64-bit operating system Windows does not have limitations in terms of size of random access memory and does not require any additional settings to control it. Up to 4GB of memory is automatically allocated for 32-bit applications (such as T-FLEX CAD).

To fully exploit the capabilities of 64-bit operating system, there is a special 64-bit version T-FLEX CAD x64. Combination of T-FLEX CAD x64 with Windows x64 allows using unlimited amount of random access memory in working with T-FLEX CAD.

T-FLEX CAD System Setup

Protection keys

Security keys are created by the Sentinel HASP technology and serve to protect T-FLEX CAD from unauthorized usage. There are two types of protection keys: hardware and software.

The hardware protection key is recorded on a physical device that is inserted into the computer USB port. The key has its own memory, which contains information about the available user licenses.

Software protection key is a program and does not require the physical device. It is associated with a specific computer.

The hardware and the software key may be network or local.

Local software protection key is installed on a single computer and can be re-hosted to another one, if it is necessary.

Local hardware protection key allows you to work on any computer that is connected to the physical device with the recorded key.

If you are using a network security key of any type, the network administrator is granted access to manage available licenses and information about these licenses. Access to the network key license management is allowed through the program **Sentinel Admin Control Cente**r.

Software for the protection key is installed automatically as part of the T-FLEX CAD installation process.

Installing Software and Hardware Protection Key

> If you use the hardware protection key do not insert it into USB-port until T-FLEX CAD installation complete.

T-FLEX CAD installation with examples, libraries and auxiliary utilities is available on DVD.

Perform the following steps to install T-FLEX CAD on your PC:

1. Insert DVD-disk with T-FLEX CAD installation into DVD-drive.

2. Run *Setup.exe* file from *T-FLEX Prerequisites* folder.

This installation should be performed one time even after T-FLEX CAD reinstalling or uninstalling.

3. Run *T-FLEX CAD.msi* from "*T-FLEX CA*" folder and follow installation wizard instructions.

You can install either T-FLEX CAD x32 or T-FLEX CAD x64 running *msi* file from appropriate folder.

Installation wizard will install examples and standard parts automatically.

4. Run appropriate *msi* files to install tutorials and necessary add-ons.

Press **[Cancel]** button to interrupt installation process.

After installation complete new folder will be created in Program Files directory on your PC.

What is Going on in Setup?

The T-FLEX CAD application files on the CD ROM are in a compressed format. The installer extracts and copies these files into the specified folder on your PC's hard disk. The memory and disk space are monitored during the installation, and an error message is displayed if these are insufficient.

T-FLEX CAD is distributed with a set of sample drawings, and a library of standard elements. The installer program creates appropriate subfolders under the installation home folder. The data structure of these subfolders is as follows:

\T-FLEX Parametric CAD\

PROGRAM	The T-FLEX CAD system files
Libraries	The library element files
Documents	The system reference files
API	Examples on Open API and Application Wizard usage for developing T-FLEX CAD add-on applications.

The T-FLEX CAD Main Window Layout

After launching of T-FLEX CAD the dialog box **"Start Page"** opens up. It includes several sections. In the section **Recent Documents** a list of recently used documents is shown. To open any of these documents, it is sufficient to point the cursor at any of them and press . The button **[Open...]** can also be used.

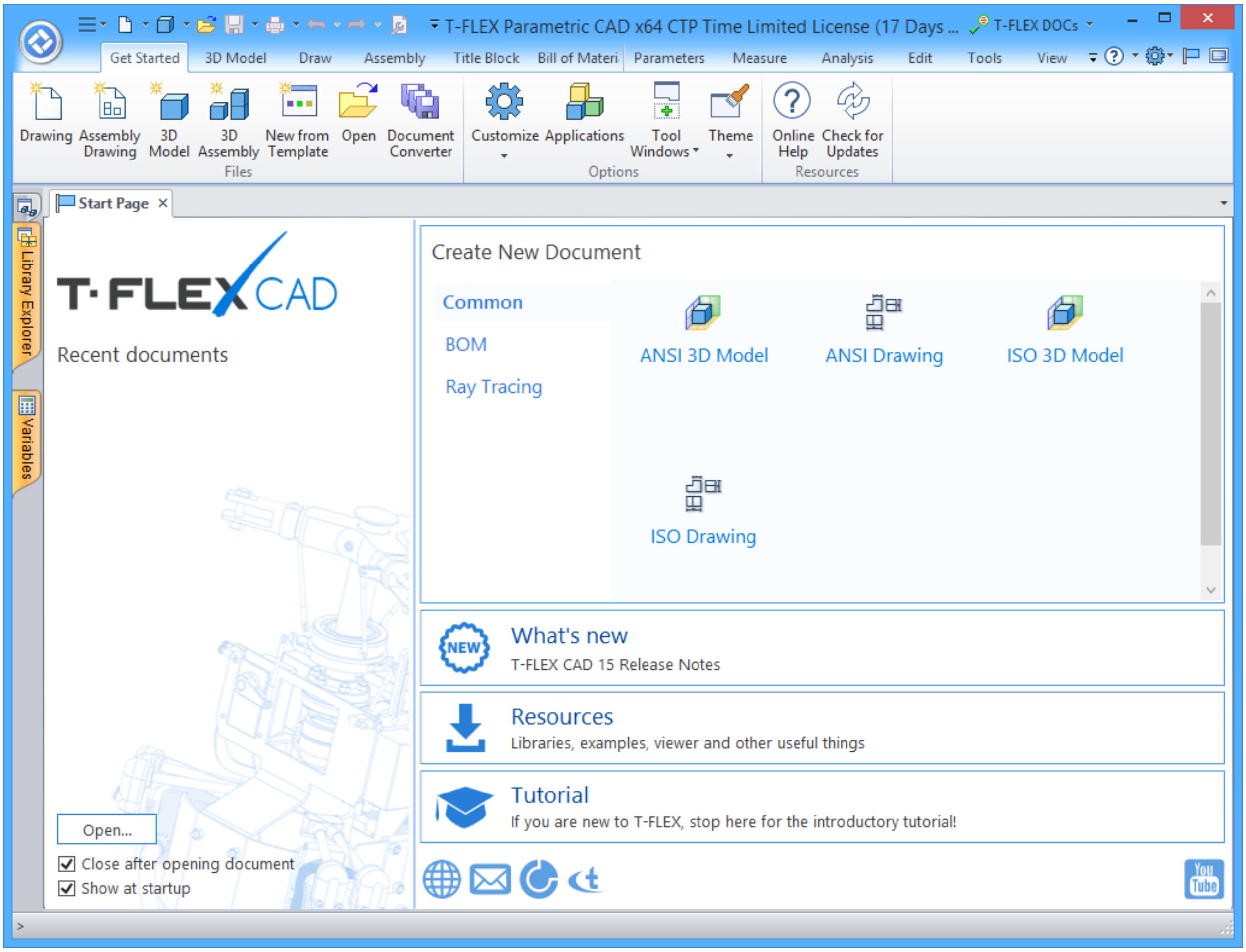
The section **Create New Document** allows creating a new document on the basis of any of the existing templates. For convenience all templates are divided into groups ("Common", "BOM", "Ray Tracing").

> The content of these sections duplicates the functionality of the menu **File** > **Recent Files** and the command **FP: Create New Document Based on Prototype**. More details on how to use these capabilities will be given in the chapter "Main Concepts of System Operation".

The chapter **What's new** contains information about the installed version.

When you select **Resources** tab, tflexcad.ru page opens in the browser. Here you can download the sample files, libraries, etc.

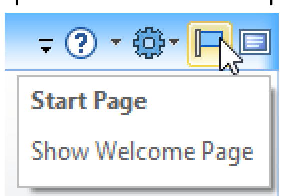
The chapter **Tutorial** contains basic information about T-FLEX CAD. The information is useful for novice users.



The dialog box **Start Page** is always visible on the screen when the standard settings of the system are used. Its tab will be aligned with the tabs of the open documents of the system (see below).

The window can be controlled by the **Customize** > **Tool Windows** > **Start Page item.**

In ribbon interface, you need to choose a special icon in the top right part of the window.

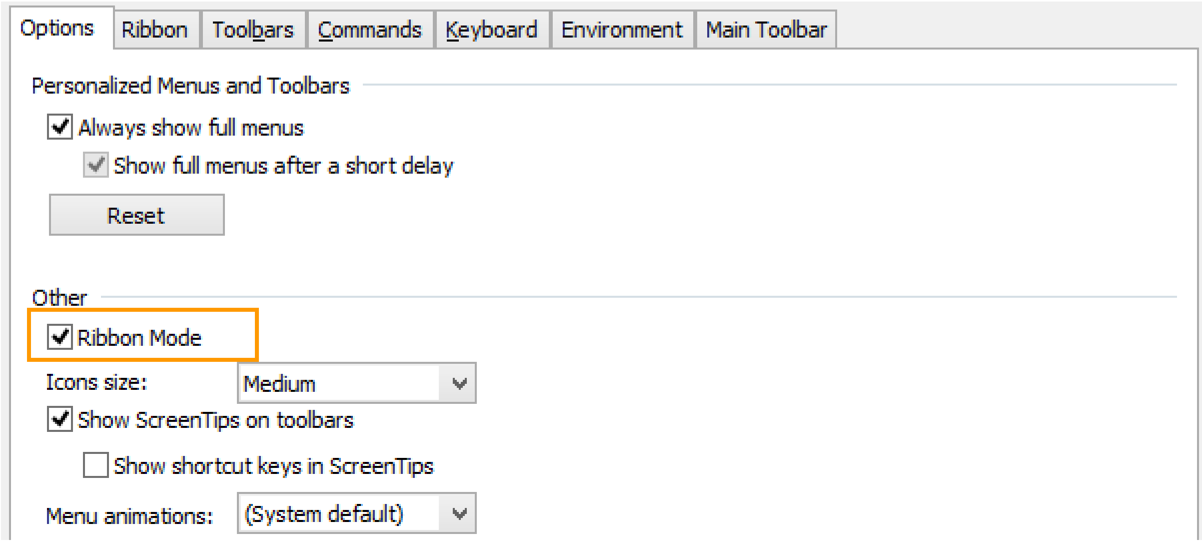


Control of the displaying of the dialog box **Start Page** during all sessions can be carried out through the dialog box of the command **SO: Set Systems Options** parameter **Show Welcome Page at Start** on the tab **Startup**.

INTERFACE

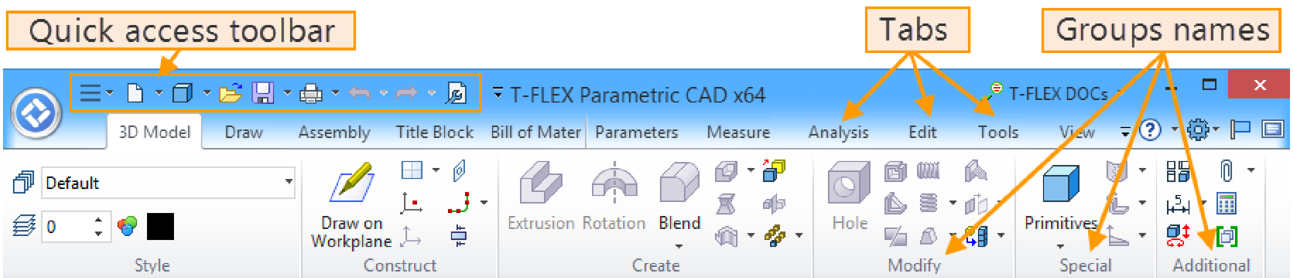
There are two interfaces in T-FLEX CAD: ribbon and textual. Ribbon interface offers a convenience of commands usage and their easy searching. Textual interface was used in previous versions of the system.

You can use flag **Ribbon mode** in the **Customize system** command on the **Options** tab.

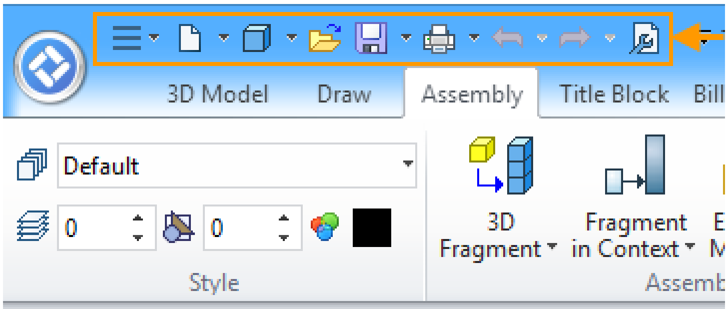


Ribbon

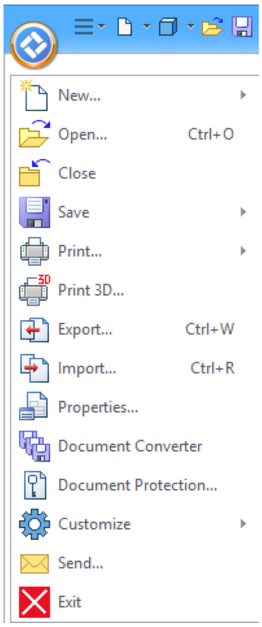
All commands on ribbon are distributed on tabs. Name of each tab reflects the content. Commands icons are grouped.

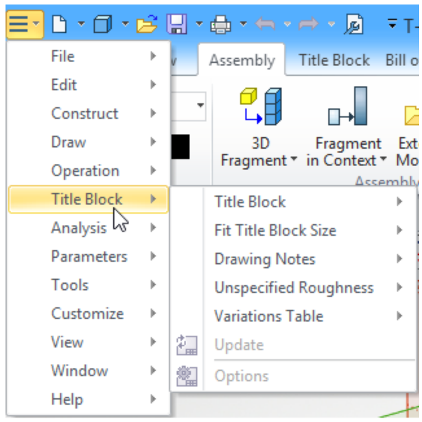


The Quick Access Toolbar containing main commands for document management is located in the header of the window. The commands are: **2D Drawing**, **Assembly drawing**, **3D Model**, **3D Assembly**, **Open**, **Save**, **Undo**, **Redo**. These commands are always available and do not depend on the active tab. The **Set Document Parameters** command  is also located here.



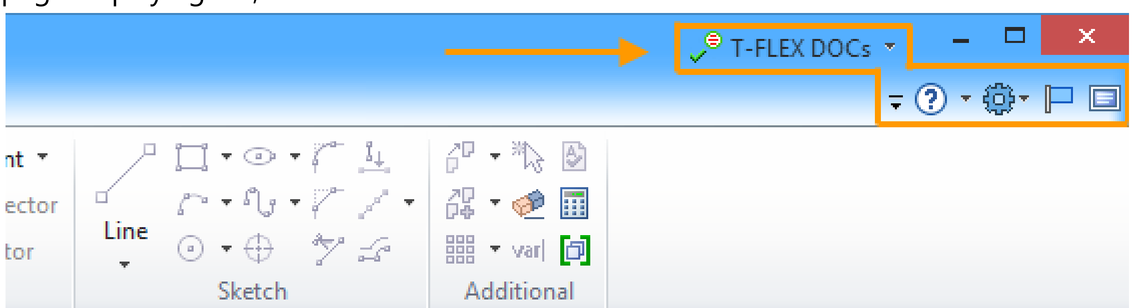
The **File** button contains commands for operating with the document.





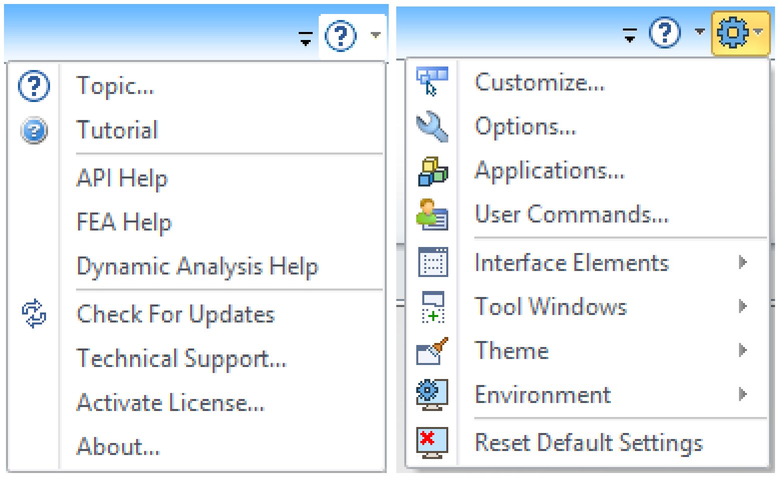
Text menu, which contains the full set of commands, can be accessed by clicking on the button.

In the upper right corner are: a field for displaying of integration with T-FLEX DOCs T-FLEX DOCs, a drop-down menu of help information, a drop-down menu of system settings, a command for the start page displaying, a command for the full screen mode activation.



In the drop-down menu you can specify displaying of the tabs.

A help drop-down menu contains commands for invoking help information about the system.

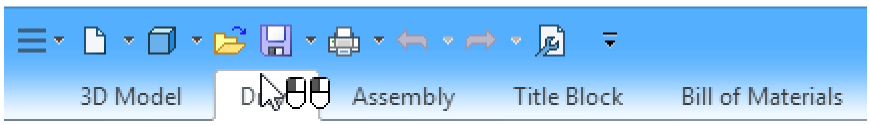


The customize drop-down menu contains commands for changing various system settings.

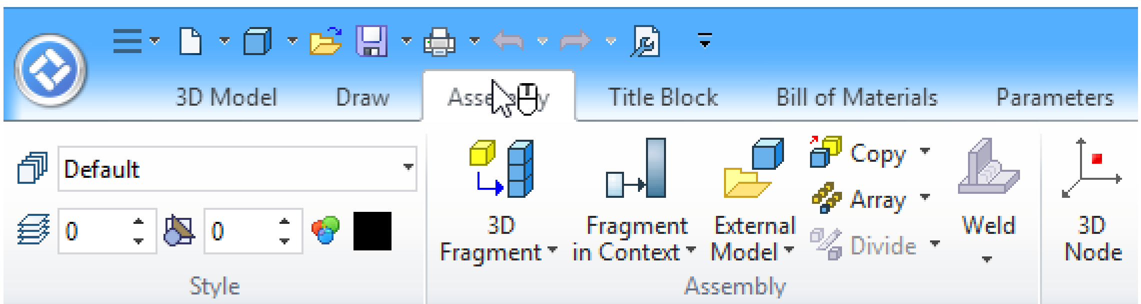
Ribbon adapts to the current mode. For example, when switching between 2D and 3D views, tabs are also switching between Draw tab and 3D Model tab.

Ribbon remembers the tab where the last selected command was located. If you choose a command from "Measure" tab in 3D view, and then continue operating in 2D view, the next activation of the 3D view will re-enable the "Measure" tab.

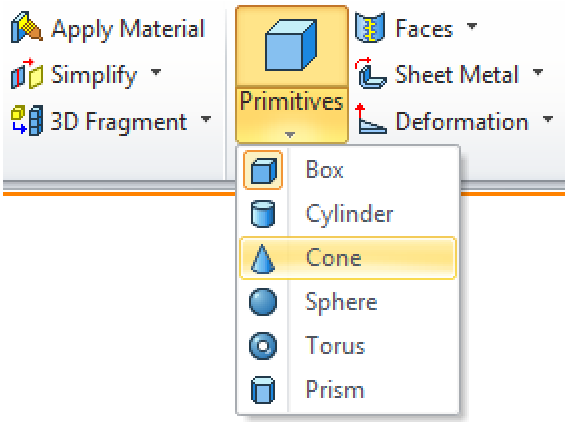
You can hide Ribbon by double-click on any tab to release a work space. Ribbon appears again in case of any of its tabs choice. It is possible to recover Ribbon by double-clicking on any of its tabs.

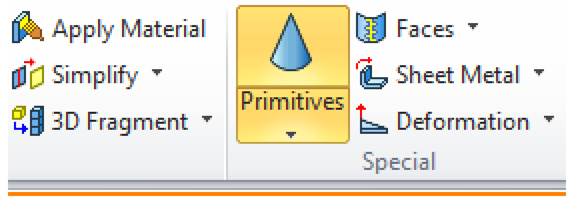


Tabs can be switched by means of a mouse wheel if you put the cursor at them and rotate the wheel.



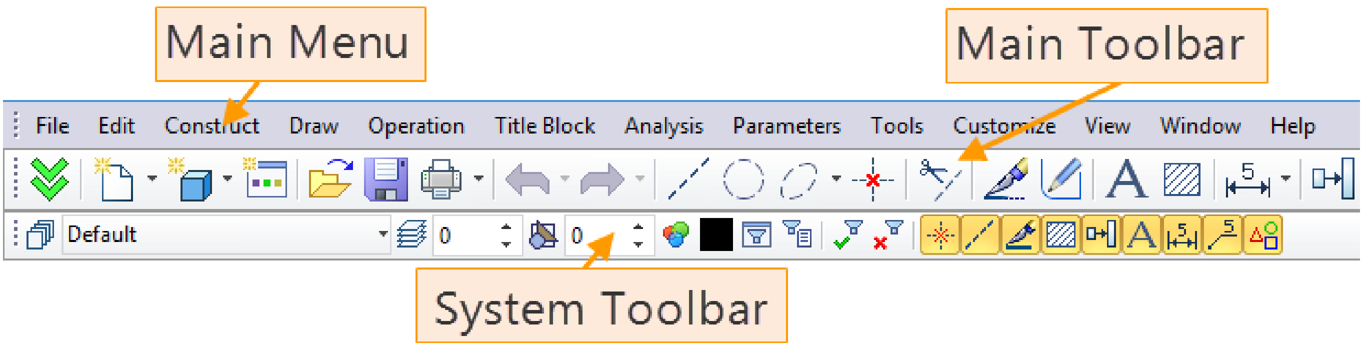
If some equivalent commands in Ribbon are united in the drop-down list, the last chosen command is remembered and displayed in the Ribbon.





For personal setup it is possible to edit existing tabs and to create your own tabs with necessary commands and operations.

Textual interface

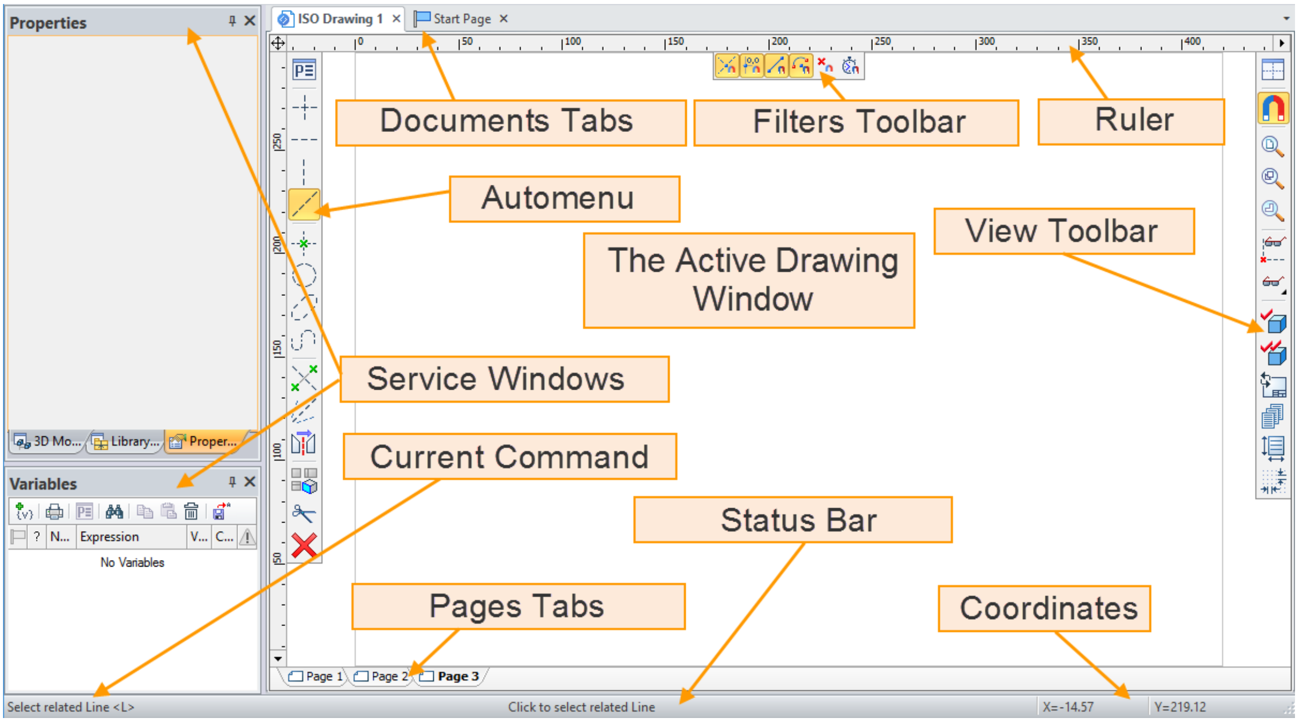


Main Menu contains the textual menu of the T-FLEX CAD commands by groups.

Main Toolbar contains icon buttons for T-FLEX CAD commands. Besides the main toolbar, the application window of the system can contain several toolbars (including the toolbars created by the user). Toolbars can be docked along one of the main window borders, or stand alone as floating windows.

System Toolbar contains the fields for modifying current settings of entities, such as color, line type, level, and layer. Also contains controls for modifying layer configuration, level configuration of the current document, and selector settings.

Elements of T-FLEX CAD Control



Ruler indicates current X and Y coordinates in the active drawing.

The active drawing window - the graphics window for displaying the drawing. Drawings can only be created and edited in this window.

Automenu - a menu of icon buttons for the options available within the current command. If no command is current, the automenu is empty.

Status bar contains the name of the current command, a prompt for the expected user action, the current X and Y coordinates, and the command-dependent auxiliary coordinate.

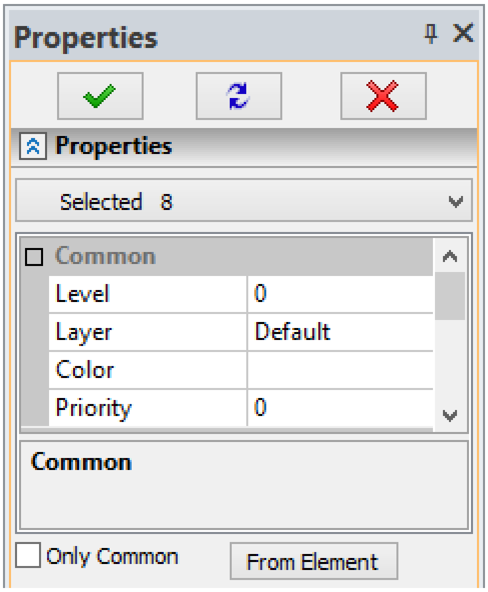
Pages Tabs provide quick access to the desired page in a multi-page document. To activate a page, select the respective tab. Tabs are not shown for the hidden pages.

Documents Tabs help quick navigation through the open documents. To activate a document, select the respective tab.

> The user can reconfigure the layout (position and visibility) of the dialog boxes and various control bars on the main T-FLEX CAD window. Use the menu **Customize** > **Tool Windows** or **Customize** > **Customize…**".

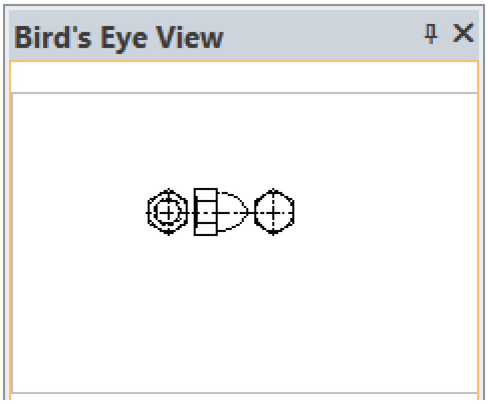
Tool Windows

Properties Window



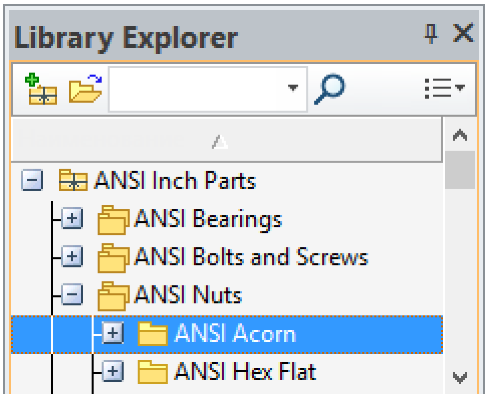
Is used for specifying parameters in transparent mode within most 2D and 3D commands. This window can be docked along one of the main window borders, or float.

Bird's Eye View Window



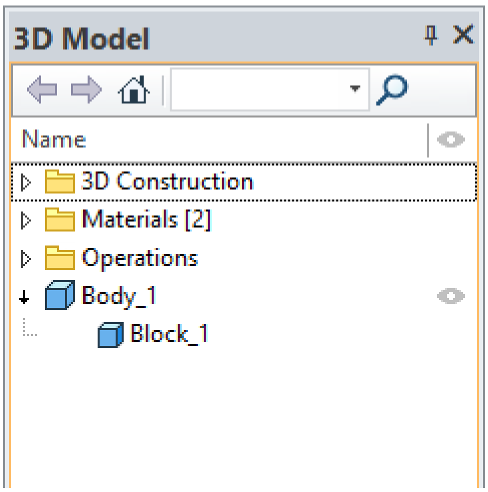
Displays the fitted view of the drawing, regardless of the current pan/zoom in the drawing window. Helps to quickly pan to any portion of the drawing. The window can be docked along one of the main window borders, or float.

Library Explorer Window



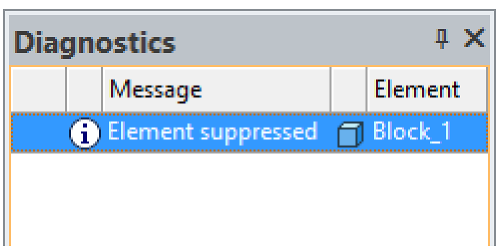
Contains graphical and textual representation of the libraries and the drawings in the current library configuration. Helps quick loading of a desired drawing and browsing drawing libraries. The window can be docked along one of the main window borders, or float.

3D Model (only for 3D release)



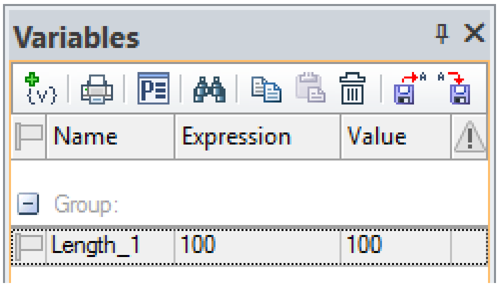
This window displays the structure of the 3D model, such as the existing workplanes and other auxiliary 3D entities and their dependencies, and the operations used for creating the model. The window can be docked along one of the main window borders, or float.

Diagnostics Window



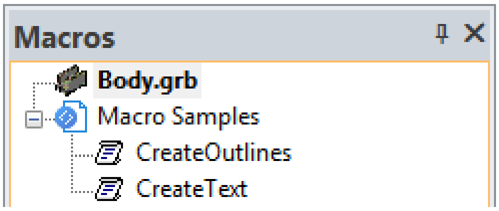
Displays messages about errors or failures that may occur during T-FLEX CAD operation. The window can be docked along one of the main window borders, or float.

Variables Window



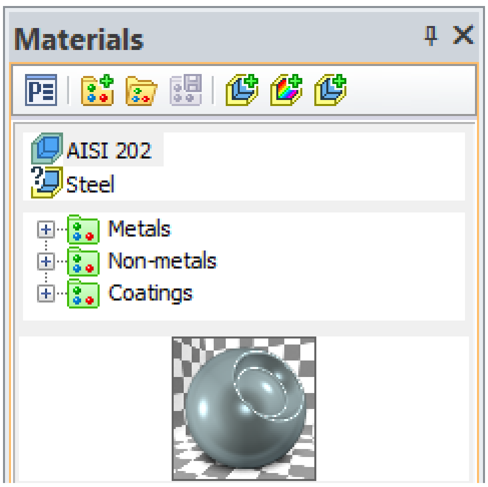
An additional window of variables editor which enables to work with the variables in an transparent mode, and simultaneously work with the drawing window or 3D model window. Upon changing the value of the variable, the model is regenerated transparently in the current window. All changes are immediately reflected on the drawing. This window can be docked along one of the main window borders, or float.

Macros Window



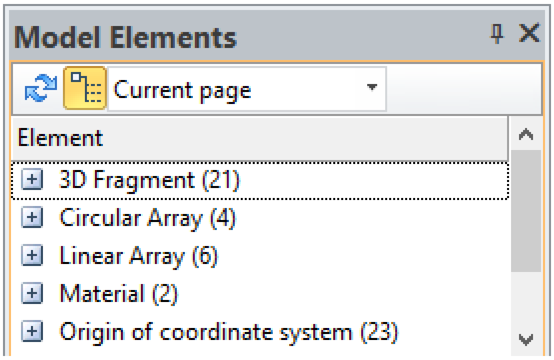
This window displays macros of the current document and macros from T-FLEX installation folder "...\Program\ Macros". The window helps to start macros for execution.

Materials Window



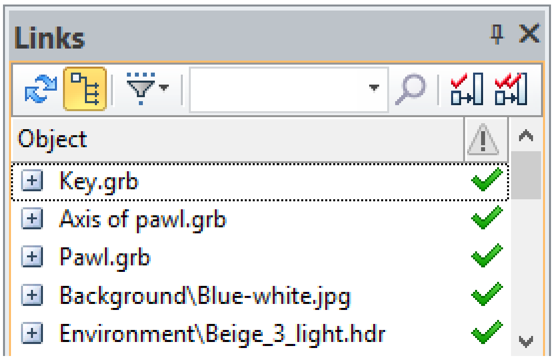
Window for working with the materials of 3D model and also with the material libraries of T-FLEX CAD.

Model Elements Window



All created in document elements are shown in the Model Elements window. Context menu is available for each of them.

External Links Window



External Links window allows to manage existing files links used in the current document.

Window "Product's structure"

Displays the model's structure. It allows us to add annotation elements to the product's structure, transforming it to the form of a single or group bill of materials.

Studies Window (only for 3D release)

The window displays data of the current document FEA and Dynamics studies. This window can be used for operations with studies.

Weld Window

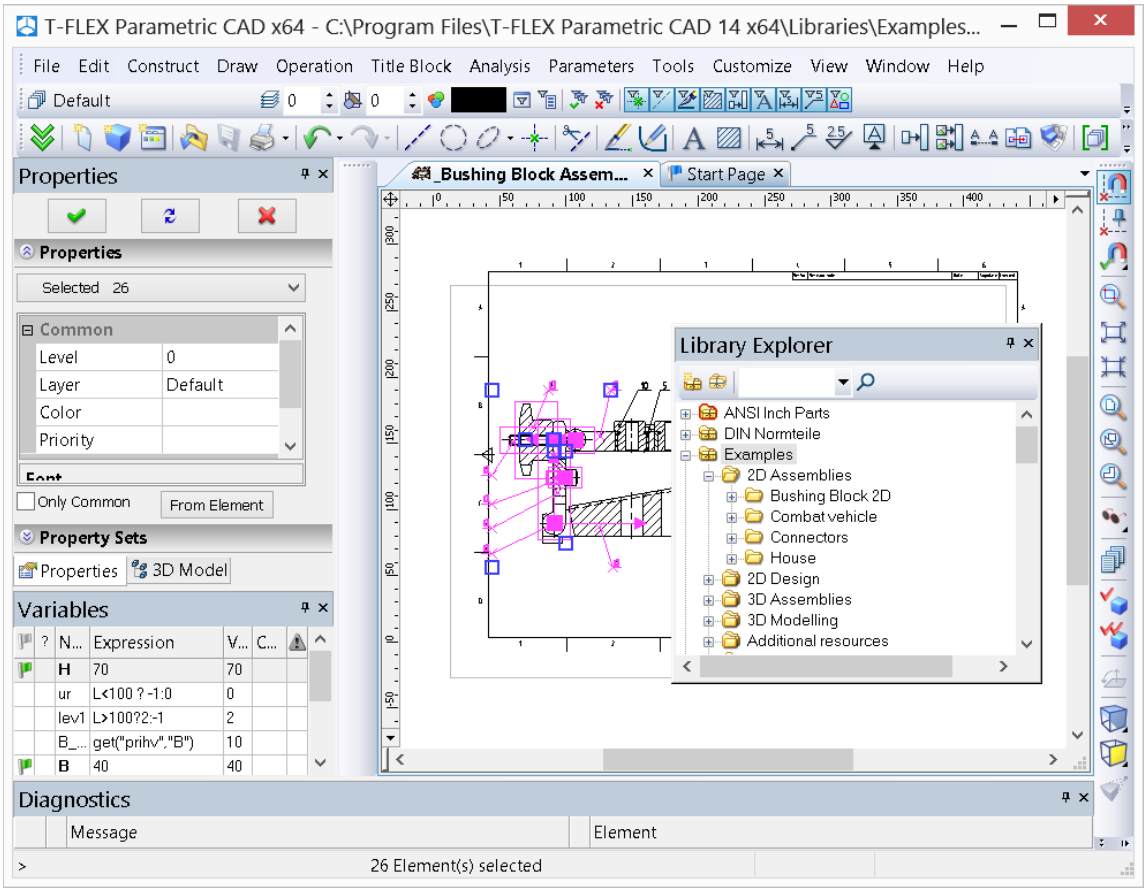
This window contains lists of welds created in the current document.

Structural Elements Window

Used for existing structural elements displaying.

Working with Tool Windows

The system tool windows (the properties window, "3D model", "Library Explorer", the Bird's eye view window, "Macros", the diagnostics window and other windows) can be positioned in the main application window in various ways. Those can be "docked" at the side of the working window, made "hideable" or set to “floating” mode. To save the workspace, some windows can be joined in one group window. Unused tool window can be turned off.

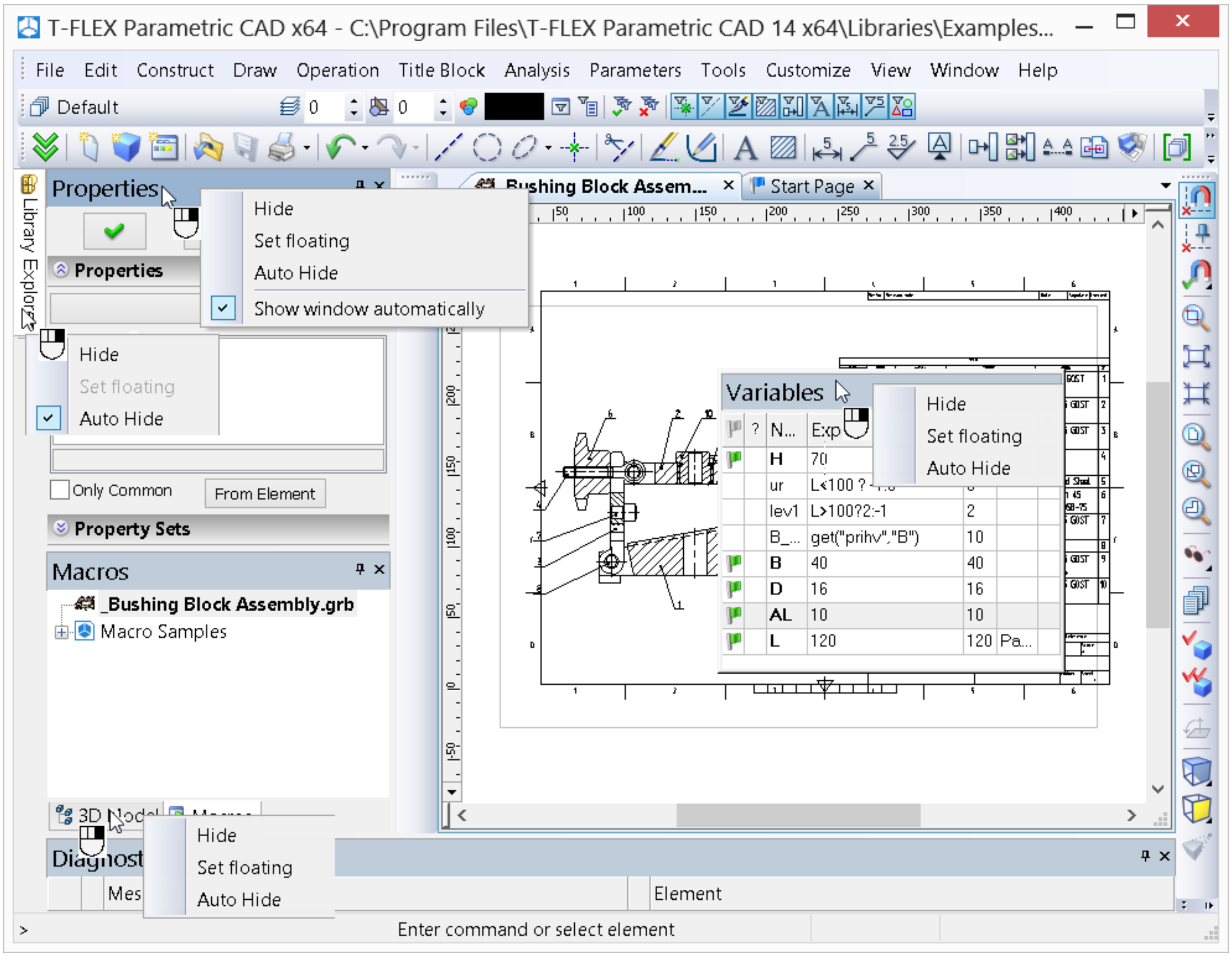


To engage a tool window, use the menu **Customize** > **Tool Windows**. The same dialog can be accessed by right clicking over an automenu of any other toolbar. Windows are closed by the button located on the title bar of the tool window.

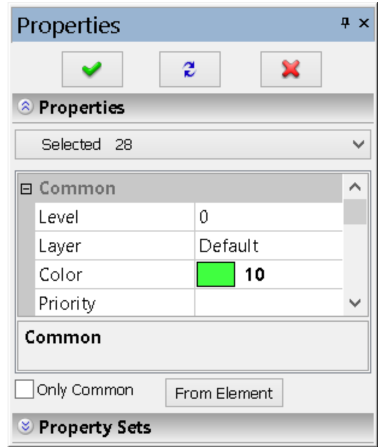
In controlling the service windows, the context menu accessed by clicking on the heading or the tab of the window can be used. The menu has several commands for controlling the state of the window:

- **Hide**. Remove the window from the screen;
- **Set floating**. Turn on the “floating” mode for the window (see below);
- **Auto Hide**. Turn on/off the auto hide mode for the window.

A set of commands available in the context menu is dependent on the state of the current window.



Upon the first launch of the system, the "3D Model", "Library Explorer" and "Properties" windows are already present in the application workspace. Those are placed in the **docked** mode along the left border of the workspace and are joined in one group window. If necessary, the two windows can be moved to any location along the perimeter of the application workspace. To display one of the joined windows separately, grab that window at its tab by pressing 🖱 and "drag" to the desired position.

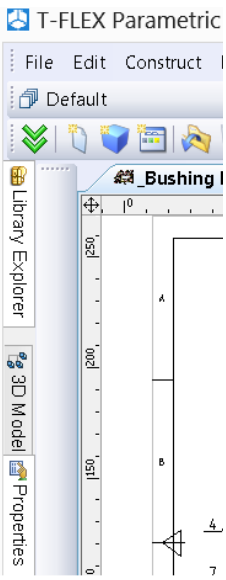


To add a tool window to an already existing or a new group window, grab the intended window by pressing and dragged to the title area of the other window or to the tabs area of an already existing group window.

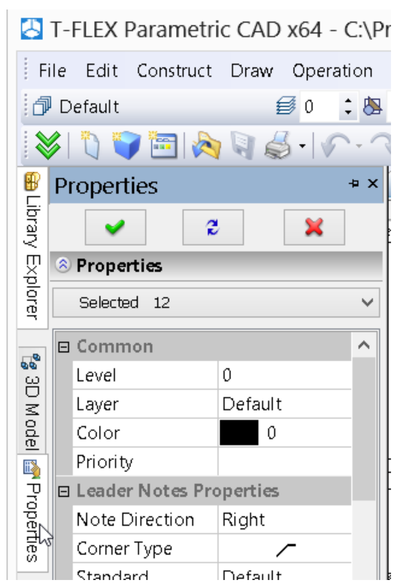
Upon dragging the windows several prompt signs will emerge showing where the window will be placed when the mouse is released.

In the cases when most of the workspace is needed, you can set the "auto hide" mode for the tool windows. In the auto hide mode, the window will appear as a tab located along the perimeter of the main application window. The window will appear automatically as you point the mouse to this tab. Once the pointer leaves the window area, it will automatically collapse.

To turn on the auto hide mode for the window, the context menu **Auto Hide** accessed by clicking on the header or the tab of the window can be used. Moreover, when the service window is in fixed position on one side from the main window of the program, the button appears on the header of the window. Pressing this button also turns on the auto hide mode for the window.



"Properties" window tab in the autohide mode

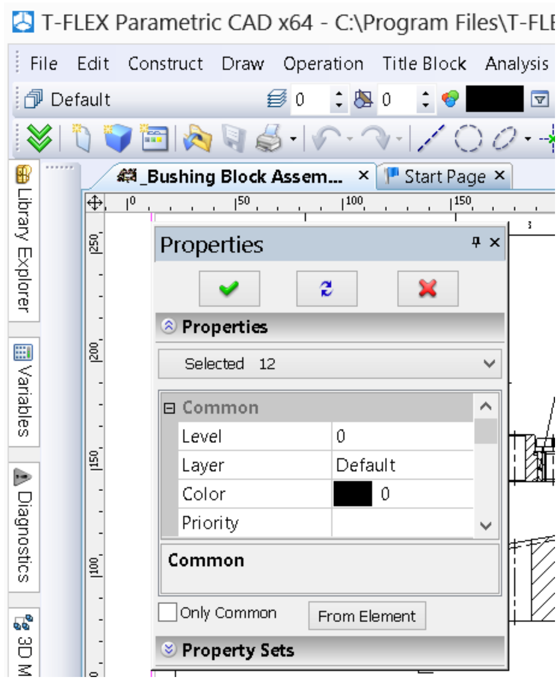


"Properties" window expands when pointed by mouse

The auto hide mode can be canceled by right clicking on the window tab and clearing the flag of the **"Auto hide"** parameter. This mode helps save significant space on the screen while maintaining benefits of the tool window functionality. Also, to turn off the auto hide mode, the button on the header of the window can be used.

It is often convenient to set some of the tool windows or a whole group window into the **floating** mode. In this way, the tool window can be placed anywhere within the application workspace without being docked.

Setting a tool window into the floating mode is done by grabbing the window title or tab in the group window by pressing and dragging into the drawing area of the application window. You can set to this mode not only separate windows, but group windows as well. To do this, grab a group window at the title by pressing drag into the drawing area of the application window in the same way.



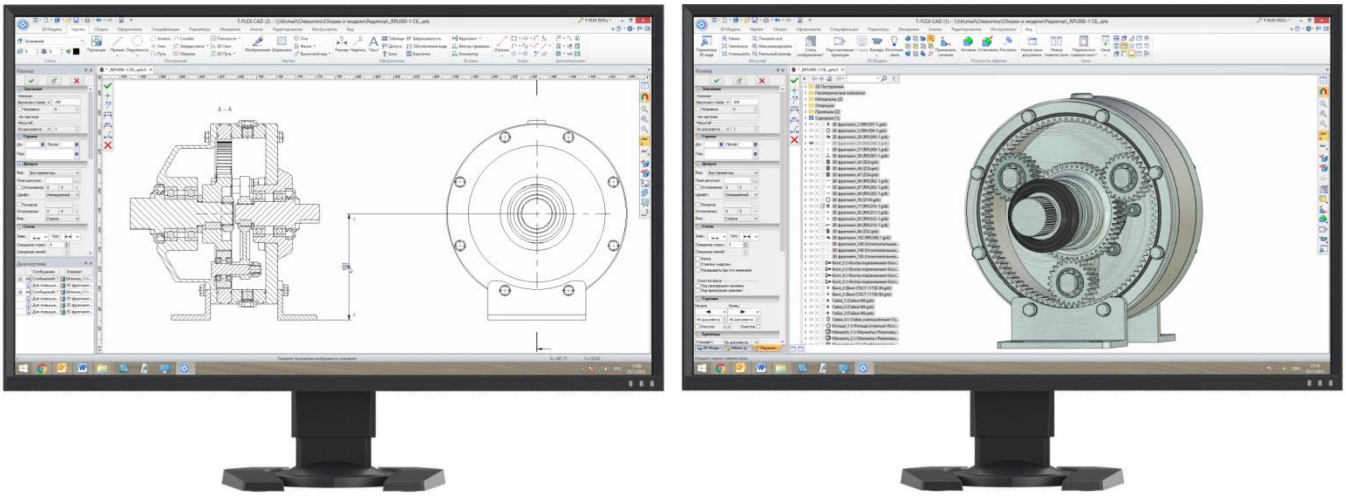
To turn on the floating mode for a window the command **Set floating** in the context menu of the given window can be used. Note that if a window, for which the context menu is called, was grouped with other service windows into a group window, then the floating mode will be applied to the whole group window.

To cancel the floating mode, grab the window at the title and by pressing drag it to a side of the drawing window. As you do this, the outline of the dragged window will be changing depending on available snapping: separately (right, left, bottom, etc.) or in a group window. To suppress snapping to sides, while moving the window hold <Ctrl> the key.

WORKING WITH MULTIPLE MONITORS

You can open multiple windows for one document and operate with it on multiple monitors.

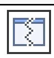
For example, if your computer is connected with two monitors, you can display a 3D model on one of them and a drawing with the same model projections on the other. Changes in the model are displayed in all main windows that are associated with the same document. So that, changes in the model made on the first monitor are immediately displayed on the projections at another monitor after the **Shift+f7: Regenerate all** command activation.



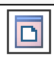
New main windows are independent from each other. Each of the windows has commands, input parameters, and other user interface elements.

> The system will ask you about saving of the document only when you close the main window with its last open copy.

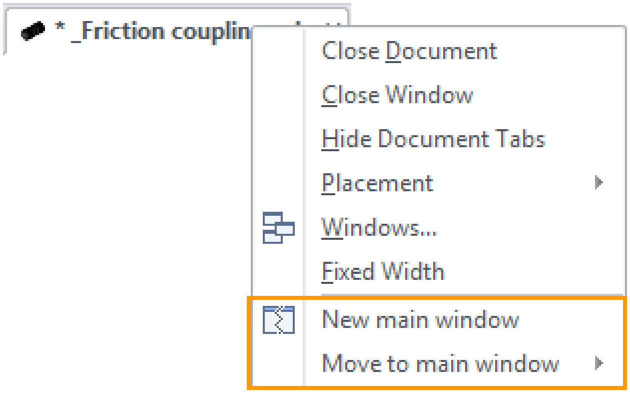
You can copy document to a new main window via command **New Main Window**:

Icon	Ribbon
	View → Window → New Main Window
Keyboard	Textual Menu
	Window > New Main Window

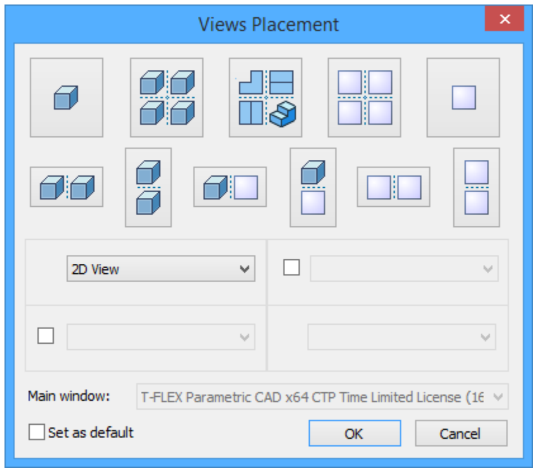
You can transfer document to a new main window via command **Move to Main Window**:

Icon	Ribbon
	View → Window → Move to Main Window
Keyboard	Textual Menu
	Window > Move to Main Window

The commands are also available in the document tabs context menus.



Before new main window opening, you can specify the **Views placement** for it.



The main windows related to a single document are numbered. Number of a window is displayed in its title in parentheses. It helps to choose the window if you want to copy or transfer the document.

Drawing Basic Terms

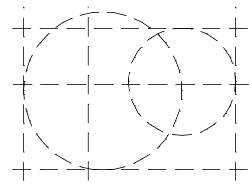
Drawing in T-FLEX CAD involves using several types of entities.

Construction entities. These make the framework of a drawing. The graphic entities of the actual drawing are drawn over the construction entities. The construction entities include construction lines and nodes. These construction lines and nodes are the principal elements for defining the parametric layout of the drawing. The analog for these in the conventional drawing is the thin pencil lines to be later marked in ink. The parametric behavior of the drawing will be driven by the relationships between the various-type construction lines and the nodes. This will result in a particular way in which the drawing geometry will adjust to changing parameters. The construction entities are displayed solely for user reference. They do not appear on printouts or plots, and are not exported.

Graphic Entities. These constitute the actual drawing of the drawing. The graphic entities include the graphic lines, dimensions, text, hatches, GD&T symbols, etc. These entities may be “snapped” to respective construction entities. In this case, modifications in the construction entities and nodes propagate on the corresponding graphic entities. This is the main technique for parametric design in T-FLEX CAD. The graphic entities constitute the drawing image on a printout or a plot.

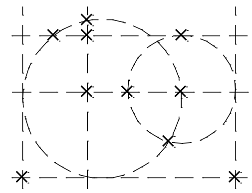
The Auxiliary Entities of T-FLEX CAD are variables, databases, reports and other certain system data.

Construction Entities



Construction Lines are the core elements of the T-FLEX CAD parametric model. These are “thin” base lines that define the parametric framework of a drawing. The construction lines include infinite straight lines, circles, ellipses, splines, offset lines, function curves, and paths. They are displayed as dashed lines.

The in-depth description of the construction line types and their creation techniques is given in the following chapters. The particular ways of creating construction lines define the behavior of the drawing as the user modifies location of any construction line. This is due to interdependencies among the construction lines that are established at their creation.



A Node is a point whose placement is defined by a particular way of creation and by interdependencies with other entities in the model. Nodes are also the core elements of the T-FLEX CAD parametric model.

Typically, nodes are created at construction line intersections.

The nodes are directly involved in defining the parametric model that will drive other construction entities. Examples of such situations are: a line passing through a node at a specified angle to another line, a circle passing through two nodes, etc. Modifying the location of one of the lines defining the node will cause the node to adjust. This change will propagate on other construction entities related to the node. The nodes are also used for defining the ends of the graphic line segments and other graphic entities.

Besides the nodes that are defined by intersections of pairs of construction lines, T-FLEX CAD supports several other types of nodes whose creation techniques are described below. For now, let’s consider only the difference between the “snapped” and “free” nodes.

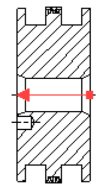
The typical technique of creating a parametric model implies creating nodes at construction line intersections. This technique is called **constrained drawing mode**. While in “constrained drawing” mode, creating a node at some location will undergo automatic snapping to the nearest to cursor pair of construction lines and their intersection.

Creating “free” nodes is a special drawing technique used in non-parametric drawing, such as sketching. This will further be referred to as **free drawing mode**. While in “free drawing” mode, the nodes are created exactly under the cursor, without snapping to construction line intersections.

The “constrained drawing” mode is indicated by the icon  of the T-FLEX CAD automenu.

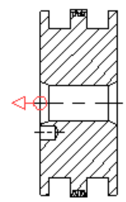
The “free drawing” mode is indicated by the icon  of the automenu. Switching between these modes is done with <Ctrl><F> or by picking the respective automenu icon.

The recommended drawing technique is using the “constrained drawing” mode. Avoid using mixed modes on the same drawing, as this may cause errors in parametric modifications of the drawing.



Fixing Vector is a construction entity that helps defining the location and orientation of the drawing that is used as a fragment in an assembly drawing.

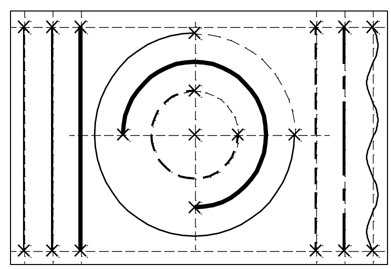
Connector is a construction entity that provides a placement reference for 2D fragments. Besides the geometrical location (the origin of the coordinate system and the axes orientation), a connector can keep additional data (both the dimensional and non-dimensional) that is necessary for “plugging in” the 2D fragments. These data are stored as a list of named values that can be either fixed constants or modifiable parameters. As for the parameters, their names within the connector are significant in the following way: assigning same names to the external parameters of the element to be connected makes these parameters assume the values of their counterparts in the connector.



Graphic Entities

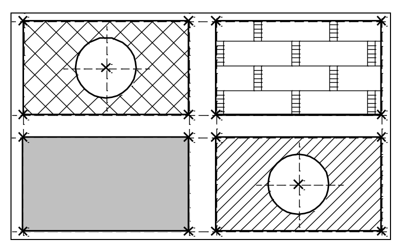
Graphic Lines are the lines constituting the actual drawing of the drawing. Graphic lines include straight segments bound by a pair of nodes, full entities, such as circles, closed splines and so on, except for the infinite straight lines, and the portions thereof bound by pairs of nodes, also splines through nodes.

The graphic lines may be of various types (main solid, thin solid, dashed, dotted etc. They are snapped to nodes and construction lines.



Hatches and Fillings are closed-contour single-connected or multiple-connected areas filled with various patterns or colors.

Hatch contours are snapped to nodes and construction lines. They adjust to node location modifications. The filling pattern also regenerates automatically as the contour changes.



Text is a single-line or multi-line textual data input via a text editor or directly in the drawing window. Either way of input supports various fonts. Besides, T-FLEX CAD supports use of paragraph formatting and other operations. A text can either be located in absolute coordinates and thus independently from the construction entities, or be snapped to construction lines and nodes.

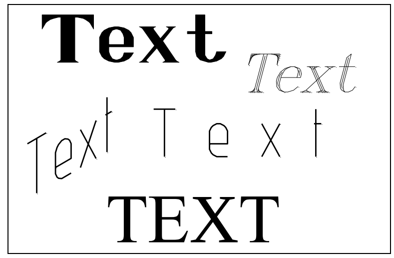
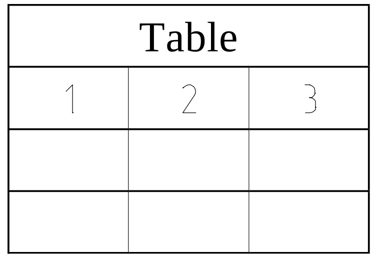
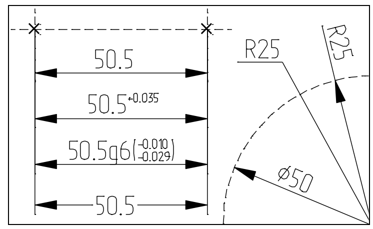


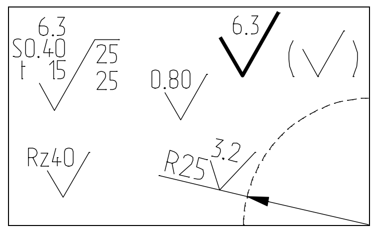
Table is an element of drawing layout. It is composed of lines and textual data. Tables are created by the same command as text. A table can either be located in absolute coordinates and thus independently from the construction entities, or be snapped to nodes.



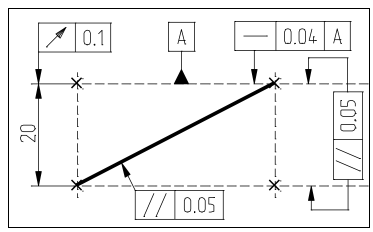
Dimension is a standard element of drawing layout. It is composed of lines and textual data. A dimension is created with respect to construction lines and nodes. T-FLEX CAD supports several dimensioning standards, including ANSI and Architectural ANSI. Dimensions automatically adjust to parametric modifications of the drawing.



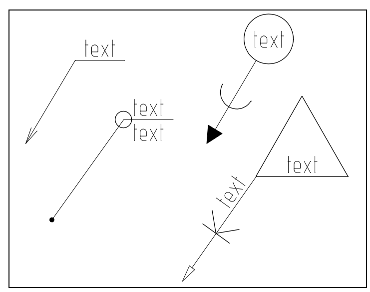
Roughness Symbol is a standard element of drawing layout. It is composed of lines and textual data. A roughness symbol can either be located in absolute coordinates, or be snapped to a node, construction or graphic line, and to a dimension.



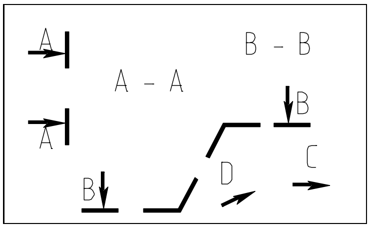
Geometric Datum and Tolerance Symbol (GD&T Symbol) is a standard element of drawing layout. It is composed of lines and textual data. A GD&T symbol can be snapped to a node, construction or graphic line, and a dimension, or located in absolute coordinates.



Leader Note is a standard element of drawing layout. It is composed of lines and textual data. A leader note can either be located in absolute coordinates, or snapped to a node, construction or graphic line.

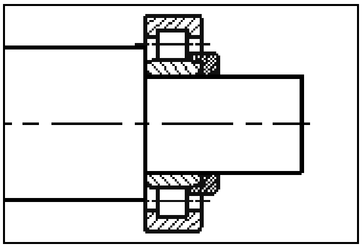


Section symbol is a standard element of drawing layout. It is composed of lines and textual data. This symbol marks various views, sections and cuts. The element can either be located in absolute coordinates, or snapped to a node.

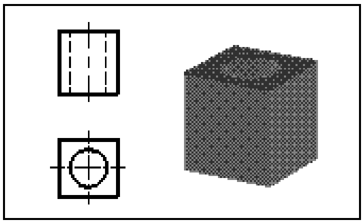


Fragments are T-FLEX CAD drawings that are used in other drawings in subassemblies and assemblies. Any T-FLEX CAD drawing can be used as a fragment.

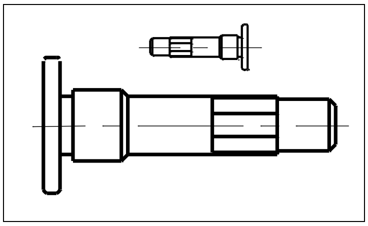
A parametric fragment in T-FLEX CAD is a drawing that can be inserted (assembled) into another drawing to a specified location and with modified parameters. The fragment appearance shall change to satisfy the parameter values. In order to create parametric fragments, the user needs to follow certain rules described below.

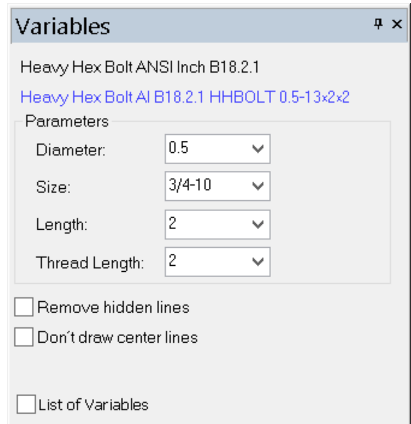


Pictures are graphic images saved in various file formats.

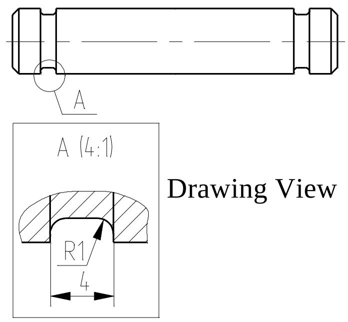


Copy is an element duplicating the original, except for the different transformation parameters.





Controls are special elements in T-FLEX CAD used for creating user-defined dialog boxes customized for controlling external parameters of a parametric model.



Drawing View

Drawing View is a T-FLEX CAD entity that displays the content of one drawing page on another page, appropriately scaled. This is a rectangular area of specified size that will contain the other page image. The main purpose of this element is combining in one drawing several elements of different scale. A common use of the Drawing View is for creating enlarged detail views.

Auxiliary Elements

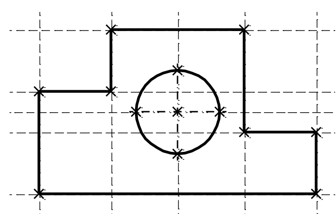
Variable is a system element for specifying non-geometrical dependencies between the various parameters. One main use of the variables is assigning their values to the construction line parameters. Consider, for example, a line parallel to a given line, at a certain distance. This distance can be defined not only by value, but via a variable as well.

Database is a table of information ordered in a certain way. Databases are used for storing information required in the drawing.

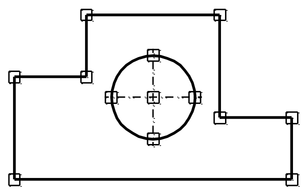
Reports are textual documents that are created with the T-FLEX CAD text editor. Reports can include the system variables and are used for creating various text documents.

Drawing Techniques

A T-FLEX CAD drawing can be created in one of the following ways:

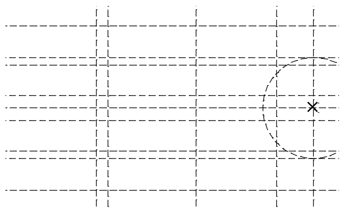


Parametric Drawing. This is the recommended drawing technique in T-FLEX CAD. Take the advantage of parametric design capabilities of T-FLEX CAD to create a drawing that can be easily modified according to your design intent. Such a drawing can also be added to a parametric model library to be later used in other, more complex drawings. In the latter case, one can specify a new location for the drawing as a fragment, and modify parameters to obtain a desired shape.



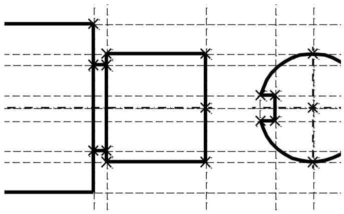
Non-parametric Drawing (Sketch). This is a conventional drawing similar to those created by most CAD systems. This drawing is created by using the standard set of functions for plotting different basic entities (straight lines, arcs, circles, ellipses, splines etc.) and by using the mechanism of objects snaps. These drawings do not have advantages of parametric drawings as far as efficient modification of parameters (dimensions) is concerned, however, in certain cases creating these drawings saves time and can give the benefit when significant subsequent modification is not required.

Creating Parametric Drawing in T-FLEX CAD



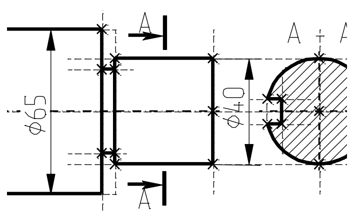
Creating a drawing in T-FLEX CAD begins with creating construction entities. Construction entities can be created by various means. First, create the base construction lines that will be used as a reference for additional construction lines. The base lines can be vertical or horizontal. Next, create straight lines and circles dependent on the base lines. For instance, construct parallel lines, tangent circles, etc. The way in which additional lines are created is stored in the model. The line intersections provide reference locations for nodes that need to be created for further construction.

More straight lines and circles can then be created referencing the earlier ones in various ways. A line, for instance, can be created through two nodes; a circle can be drawn through a node and tangent to a line. All these construction steps are stored, and in future the thus created entities will be adjusting to the base and other entity modifications according to their creation history.



Thus, the early stage of creating a drawing involves building parametric dependencies among construction entities that become the parametric framework of the drawing.

Once the construction framework is built, proceed with drawing the graphic entities. Create line segments, arcs and circles by drawing over the construction lines, snapping to nodes.



Once the actual drawing graphics is complete, proceed with the drawing layout arrangement. Create dimensions referencing construction lines and nodes. Define hatch contours, their filling patterns and other particulars. Add text entities. When placing text use snapping to nodes and construction lines if desired. This would be necessary if a text is supposed to move together with the drawing graphics.

Further, define GD&T symbols, roughnesses and leader notes. Finally, a complete parametric drawing is created and can further be modified. One can vary construction entity parameters, such as distances between parallel lines, angles between lines, radii of circles.

The graphic entities will subsequently adjust with the construction ones they reference. Thus, a family of variations of the original drawing can be created. All the rest of the drawing layout will also adjust accordingly, all done in an instant.

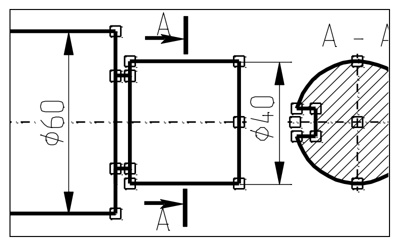
> Note that the above scenario for creating a parametric drawing in T-FLEX CAD is just one recommended technique. One can create construction entities and graphic entities in an arbitrary sequence. What is important is that the graphic entities are constrained to the construction ones.

The following chapters will tell how to use variables as drawing parameters, how to create an assembly from fragments, and much more.

Creating Non-Parametric Drawing (Sketch) in T-FLEX CAD

This technique implies quick sketching of the drawing graphics, completely avoiding preliminary creation of the construction entities.

Sketching supports object snapping and provides dynamic hints that make the drawing process simple and slick. However, thus created drawings do not share the advantage of parametric drawings in the capability of parameter (dimension) modifications. Creating non-parametric drawings may be somewhat preferable in the cases when no significant modifications are expected.



Fast Drawing Creation. Automatic Parametrics

Another method of drawing creation combines the previously described methods – it is used for creating construction-based parametric drawings using commands of non-parametric sketch. The user creates only image lines, using object snapping. T-FLEX CAD automatically “puts” necessary geometrically related construction lines under these image lines. The program defines construction types from the snapping used on creation. For example, for a straight image line parallel to another line the program creates construction line parallel to the construction line of the original image line. The resulting image line will lie on the new construction with parametric relation to the original image line.

Quick Reference on User Interface

This section provides quick reference to T-FLEX CAD while assuming user familiarity with PC operation in general, and some CAD experience as well.

Getting Help

The answers to the questions arising during operation can be got by the following means:

- The current command help can be invoked by pressing <F1> key, or by selecting menu “**Help > Current**. Pressing <F1> key when no command is active, or selecting “**Help > Contents** invokes the help contents.
- While within a command, the status bar displays hints and prompts.
- Pop-up help appears when the mouse is placed over an icon, a toolbar or other control element for a brief time. This help message tells the name of the element pointed at, or other related information.

Mouse Interface. Context Menu

T-FLEX CAD operation is mainly performed by mouse. The keyboard is used for inputting numerical values, names, and, in certain situations, for keyboard command accelerators (see below).

Using Left Mouse Button

- Pointing cursor at an icon and pressing invokes the respective command.
- Pointing cursor at an item of the textual menu and pressing also does the command call.
- Pointing cursor at a 2D construction or graphic entity in the drawing window and pressing selects this entity and activates its editing command.
- Pointing cursor at a 2D entity and double-clicking invokes the “Entity Parameters” dialog box.
- Pointing cursor at an entity and depressing and holding while moving the mouse (“dragging”) moves the entity.
- Subsequent clicking on 2D or 3D entities while holding *left* <Shift> key selects a group of entities.
- A group of 2D entities can be selected by “box selection” that occurs when the mouse with the depressed is dragged across the drawing window. The entities will be selected that are entirely within the selection box.

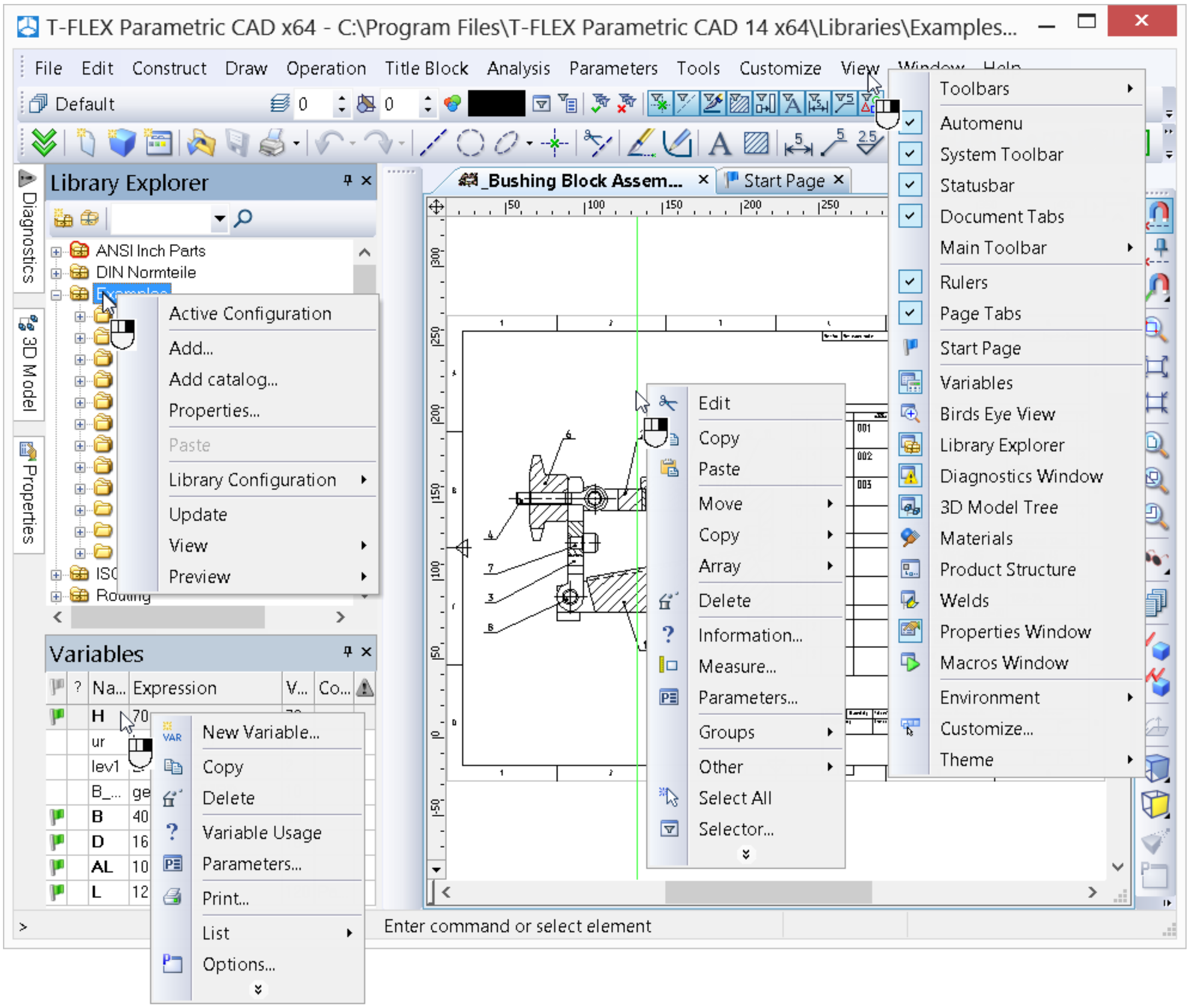
  If the mouse is moved from left to right the entities will be selected that are entirely within the selection box. The box is drawn with continuous line.

  When mouse moves from right to left, the entities are selected with the “cutting” box. This means that the elements both entirely and partially within the selection box will be selected. The box is drawn with the dashed line in this case.
- To unselect one entity in a group of selected, click on it with while holding *left* <Ctrl> key.
- Pointing cursor at a selected group of entities and clicking or double-clicking starts moving the selected entities.
- Managing libraries and arranging toolbars can be done using Drag&Drop mode. This is done by pointing cursor at an element, depressing and holding , and moving to a new location.

For more information, refer to the appropriate volumes of the documentation.

Using Right Mouse Button

- While within most commands, pressing cancels the last action or quits the command. Certain commands, as, for instance, the spline creating command or the hatch creation, allow user customization of the action performed by the command on the click. This could be quitting entity creation, canceling last selection, or completing a sequence of inputs.
- If no command is active, pressing invokes **context menu.** This menu consists of the currently available commands for the given entity. The set of items of the context menu will depend on elements the cursor is pointing at. Thus, it will be different when the cursor is pointing at drawing entities from when the cursor is over a menu area, or toolbar area, or control window area of T-FLEX CAD, etc. To launch a command, point the cursor at the desired line of the context menu and press .



- The context menu can also be invoked while working with dialog boxes (see the topic "Context Menu for Dialog Box Items" in the chapter "Customizing Drawing").

The described right mouse button actions are set as defaults, but can be customized. To do so, go to **Customize** > **Options...** (**Preferences** tab). For more information, refer to the chapter "Customizing System".

Additional Functions:

If the mouse has a wheel middle button then zooming in/out on the drawing can be done by scrolling the wheel, and panning – by dragging the mouse with the wheel button depressed.

Calling a Command

A command call in T-FLEX CAD can be performed by the following means:

- Using an icon on a toolbar;
- Selecting an item in the textual menu;
- Typing a keyboard accelerator sequence.

In this volume, any T-FLEX CAD command description will begin with a table describing these three ways of calling the command. For instance, consider the command **ESA: Select all elements in current View**. The table will appear as follows:

Icon	Ribbon
	Edit → Edit → Select All
Keyboard	Textual Menu
<ESA>, <CTRL><A>	Edit > Select All

The three columns of the table contain the respective calling instructions.

The first column indicates the keyboard accelerator for the command for inputting the command from the keyboard. All key strokes are shown together within one pair of angle brackets. Also, if defined for the command, a standard function key combination is entered next. Each key in the function key combination is shown in its own angle brackets.

The second column contains the access sequence for the command via the textual menu. The name before the dividing line is the name of the appropriate group item in the menu bar. It is followed by the item name in the pull-down menu that stands for the command. The menu item name may be different (abridged) from the full command name, as is, for instance, the item name "Select All" versus the command name "Select all elements in current View".

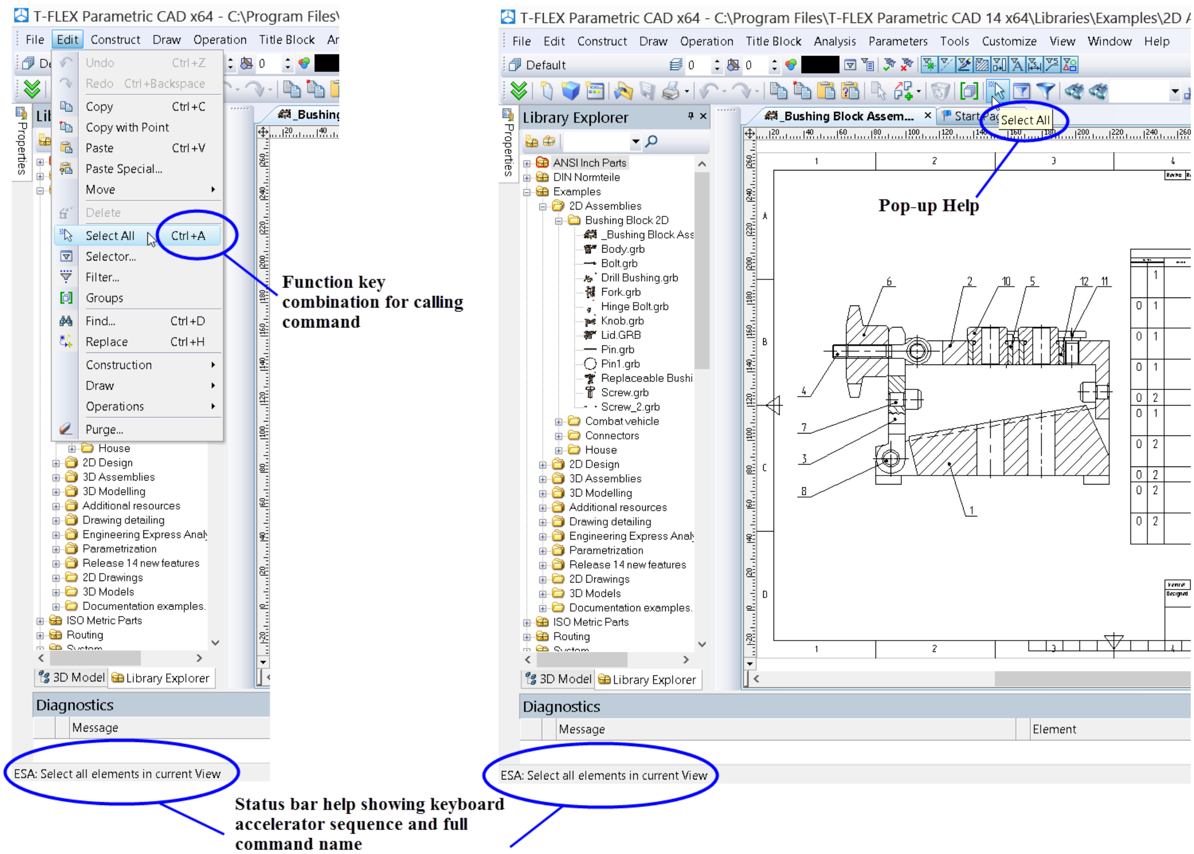
The third column of the table contains the icon image for the command. Normally, the particular toolbar containing the icon has the same name as the menu bar group item. For user convenience, a popup with the command name appears when the cursor is briefly held over an icon. Once a command is activated by pressing  on its icon, the icon stays "pushed" up until completing the command or switching to another command.

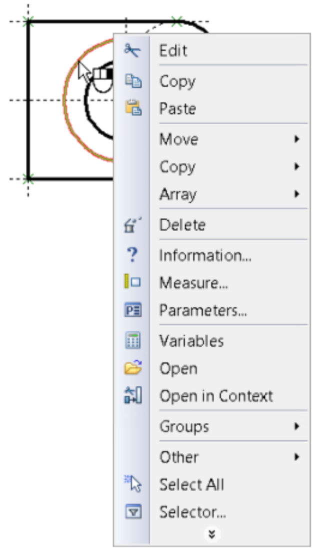
> Note: the keyboard accelerator combination is input by pressing the keys sequentially, while the function key combination is pressed simultaneously, i.e. the first key is depressed and held while pressing the second key.

The accelerator sequence for a command can be watched in the prompt field of the status bar when selecting the command in the T-FLEX CAD menu bar or a toolbar. If a function key combination is defined, it is shown on the textual menu item button at the right of the name. Any command allows defining or modifying such combination.

> See "Customizing System" chapter, "Customizing Toolbars and Keyboard" topic, "Keyboard" tab.

When inputting a command by typing, make sure the system is not within another command, and the status bar is empty.





Each command has an additional set of options and subcommands that can be accessed via the automenu or from keyboard. The keyboard accelerators appear on the pop-ups by the respective commands.

Some commands can be conveniently accessed from the context menu. The context menu is invoked by pressing after selecting one or several elements. The context menu contains a list of commands available with the given selected group.

Canceling a Command

The last action can be cancelled by pressing in the drawing area or <Esc> key. Repeated pressing quits the command. Alternatively, use the  icon of the automenu. Canceling a command clears the command field in the status bar and the automenu.

Starting System, Saving Drawing, Exiting System

Upon the start of the system the dialog box “Start Page” appears on the screen. It has been explained how to work with this dialog box at the beginning of this chapter. It is worth mentioning again that this dialog box allows creating new documents on the basis of templates already existing in the system, and it shows the list of the recently used documents (with the possibility of opening them). Also, this dialog box has various links, which can be useful in working with the system.

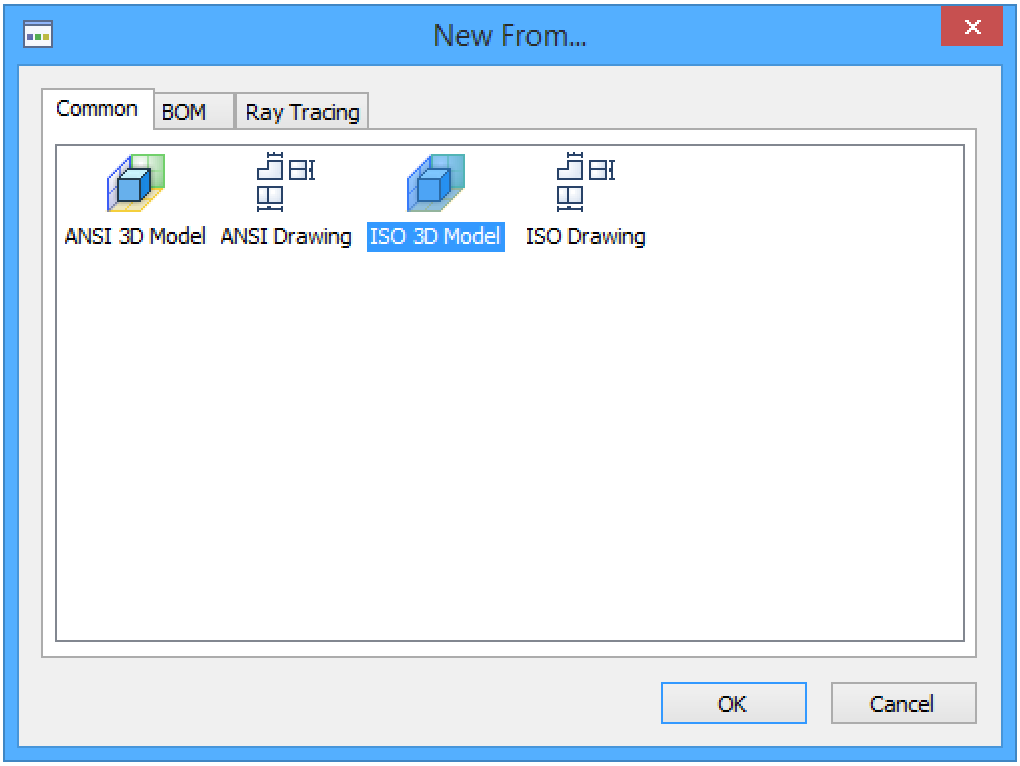
In addition to the dialog box “Start Page”, to create new documents and open already existing ones, the system commands gathered in the textual menu **File** can be used.

FN: Create New Model command allows to create a new document:

Icon	Ribbon
	Get Started → Files → Drawing
Keyboard	Textual Menu
<FN>, <Ctrl><N>	File > New > Drawing

FP: Create New Document Based on Prototype command displays a dialog box that allows to select a prototype file for the new document:

Icon	Ribbon
	Get Started → Files → New From Prototype
Keyboard	Textual Menu
<FP>	File > New > From Prototype



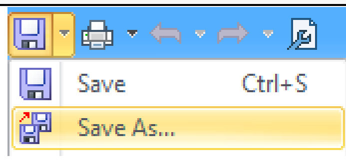
O: Open Model command brings up the standard "Open" dialog box to open a document for editing:

Icon	Ribbon
	Get Started → Files → Open
Keyboard	Textual Menu
<O>, <Ctrl><O>	File > Open

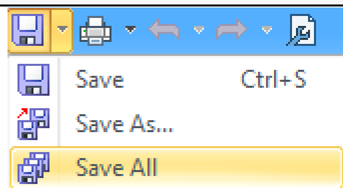
SA: Save Model command saves the current document:

Icon	Ribbon
	
Keyboard	Textual Menu
<SA>, <Ctrl><S>	File > Save

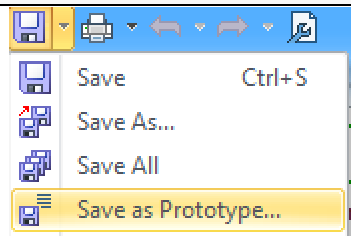
SV: Save Model As command allows the user to save the current document into a new file with a different name without changing the original document:

Icon	Ribbon
	
Keyboard	Textual Menu
<SV>	File > Save As

SL: Save All Modified Models command saves all currently open documents:

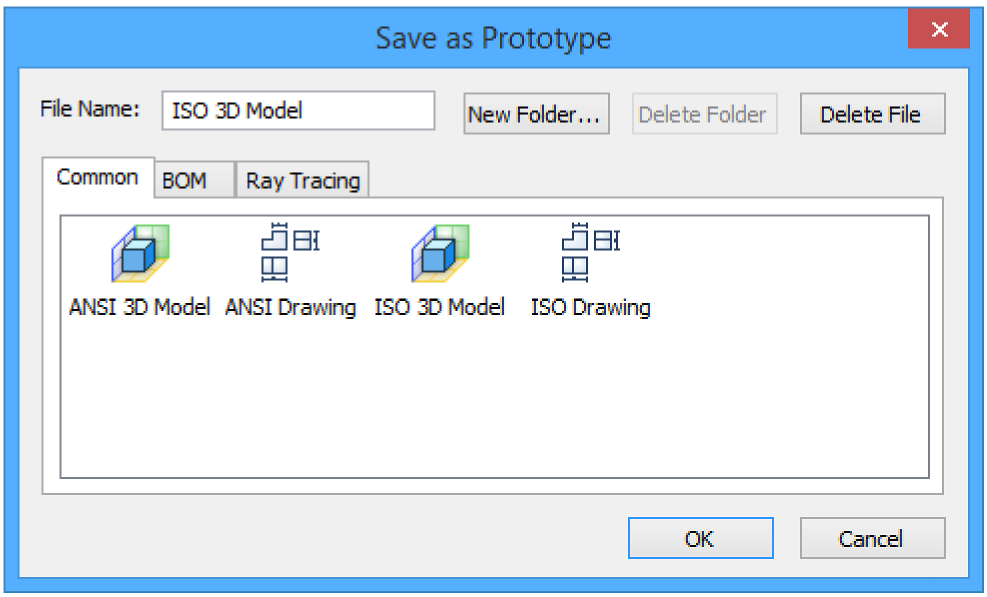
Icon	Ribbon
	
Keyboard	Textual Menu
<SL>	File > Save All

SY: Save current document as prototype for new documents command saves the current document as a prototype for creating new documents:

Icon	Ribbon
	
Keyboard	Textual Menu
<SY>	File > Save as Prototype

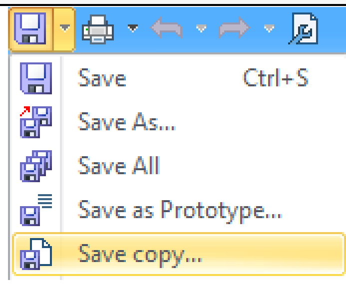
Once this command is called, a dialog box appears on the screen. This dialog allows the user to specify the name for the prototype file, specify the tab in this dialog box for this prototype or create a new tab if desired, and also delete unnecessary files and tabs.

The prototype files are located in the "Prototypes" folder under the "Program" folder off the T-FLEX CAD home. This is exactly the folder whose content is displayed in the dialog box by default.



A prototype folder can be specified by the command **SO: Set System Options**, **Folders** tab.

Save document copy to file with different name command allows to save copy of the document without opening the copy. In this case you can continue working in the source document.

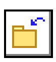
Icon	Ribbon
	
Keyboard	Textual Menu
	File > Save Copy

PS: Show Model Properties command displays all properties of the current document, and allows to input a brief comment:

Icon	Ribbon
	 → Properties
Keyboard	Textual Menu
<PS>	File > Properties

File > **Recent Files** displays the list of files open during previous sessions. Select a file name in the list to open. The number of displayed recent files can be set via the **Customize** > **Options** > **Preferences** command.

"**FCL: Close Model**" command closes the current document:

Icon	Ribbon
	 → Close
Keyboard	Textual Menu
<FCL>	File > Close

A document can also be closed using the button ☒, located in the top-right corner of the document window.

"**FI: Exit system**" command closes the T-FLEX CAD session:

Keyboard input	Textual Menu	Icon
<Alt><F4>	File > Exit	

The system queries the user whether to save modified documents (if any) before exiting.

Function Keys

Certain frequently used commands are bound to function key combinations, as follows:

<F1>	Get reference information (help) on the current command
<Alt><F1>	Get information on the selected element(s)
<Ctrl><S>	Save document
<Ctrl><O>	Open document
<Ctrl><N>	Create new document
<Ctrl><P>	Print document
<Ctrl><F7>	Recalculate parameters of the current document
<Alt><F7>	Regenerate 3D model
<F3>	Call **ZW: Zoom Window** command. This is an instant command that can be called while within another command. The previously active command continues after this command.
<Ctrl><Shift><PgUp>	Zoom in
<Ctrl><Shift><PgDown>	Zoom out
<Ctrl><Shift><Left>	Pan left (moves the model left)
<Ctrl><Shift><Right>	Pan right (moves the model right)
<Ctrl><Shift><Up>	Pan up (moves the model up)
<Ctrl><Shift><Down>	Pan down (moves the model down)
<Ctrl><Shift><Home>	Fit to page
<Ctrl><Shift><End>	Fit all objects
<F7>	Call **RD: Update Model Windows** command
<Alt><BackSpace> or <Ctrl><Z>	Call **UN: Undo Changes** command
<Ctrl><BackSpace> or <Ctrl><Y>	Call **RED: Redo Changes** command

Please note that the above command bindings can be changed via the **Customize > Customize... > Keyboard** command.

BRIEF INTRODUCTORY COURSE

This chapter introduces various drawing techniques. The manual describes all necessary steps in the drawing process. Once you start drawing with T-FLEX CAD, you will have an opportunity to fully appreciate the advantages of this system. Further you will learn the basic commands and principles of creating a drawing with the aid of T-FLEX CAD.

T-FLEX CAD supports creation of two types of drawings: parametric and nonparametric (sketches). The mainly used type is the parametric drawing.

It takes somewhat more time resources to create a parametric drawing; nevertheless, later on such drawing will be easily modifiable as you desire. A nonparametric drawing (sketch) can be created faster. Its creation method is similar to the ways of drawing in some other CAD systems. However, nonparametric drawings do not possess the advantage of effective parameter (dimension) modification. Therefore, this method is recommended to use in the cases when no significant modifications to a drawing are expected.

To speed up creation of parametric drawings, the system supports the use of automatic parameterization. This mode allows constructing not too complicated parametric drawings just like nonparametric ones: all you do is create graphic lines using object snapping. The construction lines constrained by the parametric relation will be automatically “slipped under” the graphic lines by the system.

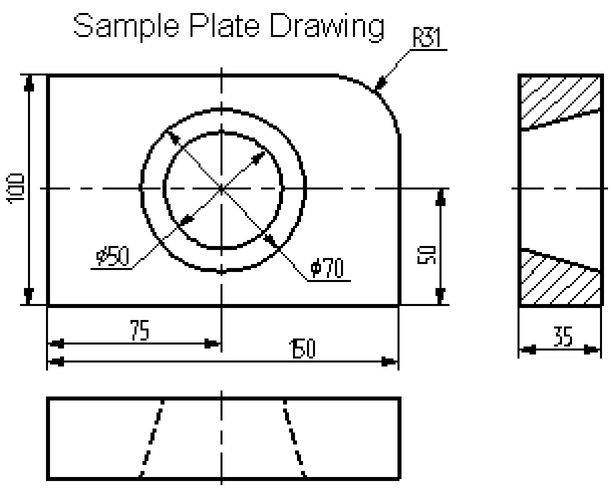
Three approaches to creating a T-FLEX CAD drawing will be reviewed below: creating a parametric drawing by the traditional method (that is, with the manual creation of construction elements), creating a nonparametric sketch drawing, and creating a parametric drawing in the automatic parameterization mode.

CREATING PARAMETRIC DRAWING

The following diagram shows a drawing to be created. It is a plate with a through hole of conical shape. The drawing will be defined parametrically so that any modifications will automatically reflect on all projections.

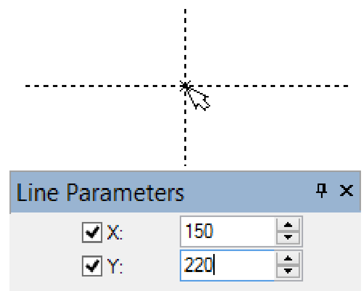
Let’s begin with the main (elevation) view of the plate. First, create the necessary “thin” construction lines, and then draw the graphic lines on top. Next, create the other two views using the construction lines of the main view. This creates a dependency between the views so that the two views automatically adjust to the main view modifications. Finally, apply text and dimensions.

As was mentioned, any command can be called by a number of ways. It can be typed on the keyboard, selected from the textual menu, or picked on a toolbar.



Let's begin with the command **L: Construct Line**. To invoke the command, use:

Icon	Ribbon
	Draw → Construct → Line
Keyboard	**Textual Menu**
<L>	Construct > Line



Pick the icon at the top of the automenu. A crosshair appears that follows cursor dynamically. The current coordinates of the crosshair crossing point are displayed in the status bar. There are several ways to define the crossing point. One is to simply place the cursor near the center of the drawing window and press . To define the crossing point more precisely, specify its coordinates in the property window. The coordinates can also be specified via a parameter dialog invoked by typing <P> key or picking the icon in the automenu.

As a result, two crossing construction lines will be created. Besides, a node is created at the intersection point. These lines make the basis of the view being created. The line parameters represent the absolute coordinates. The view can be moved around on the drawing by moving the base lines.

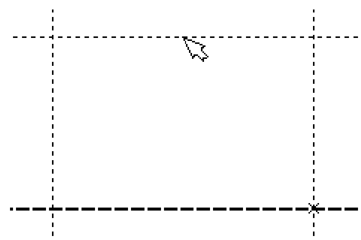
> Do not use more than two base lines on the main (independent) view, and more than one base line on the views defined by projections. This will insure freedom in placing the drawings.

A T-FLEX CAD command stays active up until it is quit or another command is called. Quitting the crosshair mode (as by pressing once) cancels the crosshair rubberbanding, but the line creation command stays active. After canceling the crosshair mode, move the cursor close to the vertical line. The line will get highlighted, and a pop-up help will appear next to cursor displaying the name of the highlighted entity. This is object snapping in action. This behavior relieves the user from typing on keyboard or using the automenu buttons.

> The object snapping is on by default when starting the application. To set or unset this mode manually, use the button on the "View" toolbar.

Pressing now starts rubberbanding of a line that follows the cursor while staying parallel to the selected one. We are now creating a line parallel to a vertical line. Such a relationship between the two construction lines, established at the creation time, is an example of an important feature of T-FLEX CAD system. This defines behavior of a set of construction entities under parametric modifications.

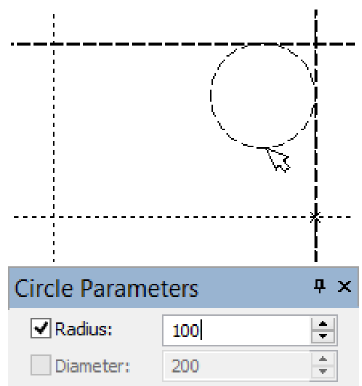
Place the new line at the left of the highlighted vertical line by pressing . The exact value of the distance can be specified in the property window or parameter dialog box. The newly created line will become the left-hand side of the part.



Pressing once cancels the parallel line creation mode, yet the line creation command stays active. (Otherwise, call it again.) Next, move the cursor toward the horizontal line and press . The line is selected as a reference for a parallel line to be created. Move the cursor up, specify an exact value of the distance, if desired, using the property window, and press fixing the top side of the part.

The next step is to round a corner of the plate with a fillet. For this purpose, let's use the command **C: Construct Circle**. Call the command via

Icon	Ribbon
	Draw → Construct → Circle
Keyboard	Textual Menu
<C>	Construct > Circle



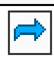
To draw the fillet at the upper-right corner of the plate, construct a circle tangent to the top and the right-hand-side lines. Move the cursor to the top line and press or ***<L>***. This starts rubberbanding of a circle whose radius adjusts as the circle follows the cursor while the line tangency stays intact. This means a circle is being constructed that is tangent to the top line. Any future modifications of the top line location will not break the circle tangency condition.

Next, move the cursor to the right-hand-side line and again press or <L>. Now, the circle becomes "tied" to the two construction lines and keeps the tangencies while being rubberbanded. Pressing fixes the current circle radius. The exact value of the radius can be specified in the property window.

If the resulting construction does not match the illustration, use **UN: Undo Changes** command,

Icon	Ribbon
	
Keyboard	Textual Menu
<UN>, <Ctrl><Z>, <Alt><BackSpace>	Edit > Undo

Each call to this command brings the system one step back. If this command was called mistakenly, its action can be reversed with the command **RED: Redo Changes**,

Icon	Ribbon
	
Keyboard	Textual Menu
<RED>, <Ctrl><BackSpace>	Edit > Redo

This command restores the action that was mistakenly undone.

One can remove all construction lines and start creating a drawing from the beginning with the command **PU: Delete Unused Construction**:

Icon	Ribbon
	Edit → Additional → Purge
Keyboard	Textual Menu
<PU>	Edit > Purge

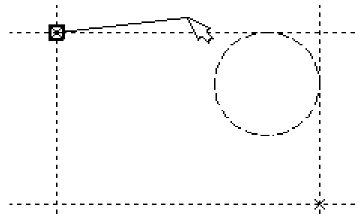
This command will delete all construction entities and allow to start drawing anew. A specific construction entity can be deleted using command **EC: Edit Construction**:

Icon	Ribbon
	Draw → Additional → 2D Construction
Keyboard	Textual Menu
<EC>	Edit > 2D Construction

Once the command is called, select the entity and delete it by pressing <Delete> key on the keyboard or by picking the icon  in the automenu.

Now, draw the graphic lines on top of the completed construction portion of the drawing. To do so, let's create graphic lines by calling **G: Create Graphic Line**. Call the command via

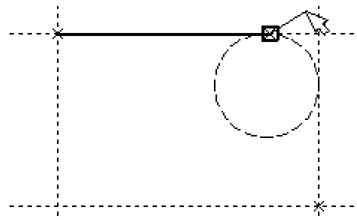
Icon	Ribbon
	Draw → Draw → Graphic Line
Keyboard	Textual Menu
<G>	Draw > Graphic Line



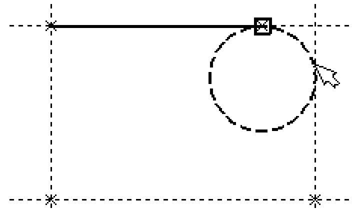
Note that the previous command is automatically terminated when calling another command via the toolbar icon button or the textual menu (no need to cancel the previous one explicitly).

Start drawing solid lines from the upper-left corner of the plate. The graphic lines snap automatically to a closest intersection of the construction lines. Simply move cursor to an intersection and press . The line will be rubberbanded after the cursor. Just keep selecting nodes or construction line intersections.

> It is not recommended to select multiple (more than two) line intersections neither by pressing <Enter> nor by . In this case, we recommend creating nodes at such intersections first. The graphics can then be applied using the <N> key. When using the <Enter> key in “free drawing” mode, a “loose” node will be created that is not constrained to any construction line. Following these tips insures correct parametric function of the drawing under modifications.

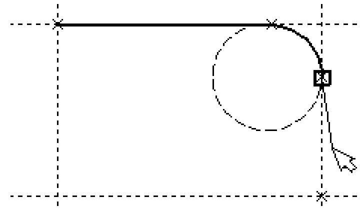


Move cursor to the tangency point between the top line and the circle, and press . What you see on screen now should be similar to the illustration at right. Note that T-FLEX system automatically adds nodes to the end points of the graphic lines, unless already created.

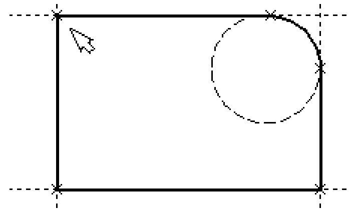


Now let’s draw a graphic line along the circle to construct an arc between the two tangency points. To do so, move the cursor to the circle and press ***<C>*** key. The circle will then get highlighted. The direction of arc creation depends on the position of the cursor when selecting the circle. To change the arc direction, press the ***<Tab>*** key.

Place the cursor just above and to the left of the second tangency point as shown.



Then press , and the graphic arc will be created in the clockwise (CW) direction, spanning to the second tangency point. The result should look like on the diagram.



Continue drawing. Select with the lower-right corner of the plate, then the lower-left one, and finish the construction in the upper-left corner where the drawing started. To complete the command press .

The drawing should look as shown here.

If applying graphic lines did not come out as desired, edit the graphics using the command **EG: Edit Graphic Line**. Call as follows,

Keyboard	Textual Menu	Icon
<EG>	Edit > Draw > Graphics	

Move the cursor to one of the lines to be edited, and press . This selects the line. It can then be deleted by pressing <Delete> key or picking the icon in the automenu. Repeat for each line to be edited. If a whole area is to be edited, use box selection. To select by box, press where one of the box corners should be, hold and drag to the desired location of the opposite corner, then release the button. As you drag the cursor, it rubberbands a rectangle of the selection box. The elements will be selected that are fully within the box. All these elements can be deleted at once.

To apply graphic lines again, call the command **G: Create Graphic Line**. To redraw the screen at any moment use the <F7> key, in case not all lines are displayed properly after editing.

Once the desired image is obtained, proceed to the next step of drawing creation. The drawing can be saved preliminary with the help of **SA: Save Model** command:

Icon	Ribbon
Keyboard	Textual Menu
<SA>	File > Save

Congratulations! You have accomplished your first drawing in T-FLEX CAD. Now let us briefly describe the system editing capabilities.

The current drawing uses five construction entities that define the shape and size of the part. These are the left-hand side, the right-hand side, the top, the bottom and the fillet radius. To modify construction entities call the command **EC: Edit Construction**:

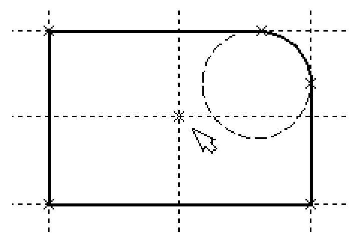
Icon	Ribbon
	Draw → Additional → 2D Construction
Keyboard	Textual Menu
<EC>	Edit > 2D Construction

Move the cursor to the left-hand-side vertical line and press . The line gets highlighted. As you move the cursor left to right, the line will move along. Specify the new position of the line by pressing . The width of the plate will change. Note that modifying locations of construction entities causes instant update of their respective "snapped" graphic lines. If you try to move the right-hand side of the plate then the whole plate will move. This is because the left-hand side was created as a dependent of the right-hand one, and the dependency stays as the right-hand side is modified. However, the left-hand side can move independently of the right. Try such manipulations with other construction entities, including the circle. As the construction entities move the size and shape of the plate will be changing while maintaining the dependencies defined at the construction time.

After testing modification capabilities of the system please bring the drawing back into an approximately original configuration as shown on a diagram above. Let's proceed with the next element of the drawing, which is the conical hole in the middle of the plate.

First, let's define the center of the circle to be constructed. To do so, let's do auxiliary construction to define the center point of the plate. T-FLEX CAD provides a handy command to create a line in the middle of two others. For two parallel lines, this command creates a parallel line in between at equal distances to the two. For intersecting ones, the resulting line passes through the intersection at equal angles to the two original lines. Thus, the new line appears as the symmetry line for the two.

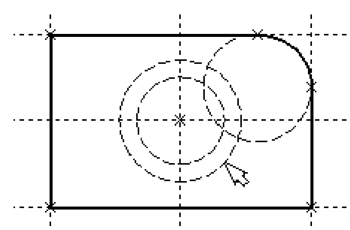
Call the command **"L: Construct Line"** and choose the icon in the automenu. Move the cursor to the right-hand side of the plate and select the vertical line by . A parallel line appears rubberbanding after the cursor. Move the cursor to the left-hand side of the plate without fixing the rubberbanded line. Now, select the left-hand-side vertical line with . This creates a new vertical line on the drawing that is the symmetry line for the two selected ones.



Follow same way to create a horizontal line as the symmetry line for the top and bottom sides of the plate. The intersection point of the two new lines will be the center of the hole to be constructed.

Next, call the circle creation command, move cursor to the intersection of the symmetry lines, and press . This starts rubberbanding of a circle with the fixed center, with the radius adjusting to the cursor position.

The circle center snaps to the node created automatically at the intersection of the symmetry lines. Fix the circle with . Just like line-to-line distances, the circle radius (diameter) can be defined approximately by mouse operation, and exactly in the property window. Note that after pressing the command **"C: Construct Circle"** stays active.

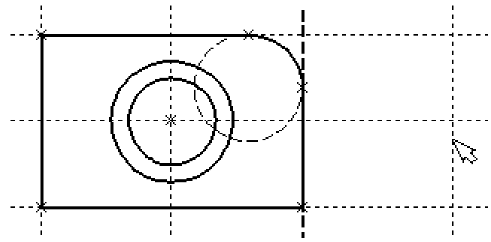


The second circle of the conical hole can be constructed as concentric to the first one. To do so, pick the icon in the automenu or type <O>. Then select the existing circle with . The new circle starts rubberbanding after the cursor. Place the cursor so that the rubberbanded circle is slightly bigger than the original one, and fix with . The exact radius difference can be managed via the property window.

Proceed with the command **G: Create Graphic Line**, move cursor to the bigger of the two circles, and press or <C>. The circle gets drawn in solid. Then, move the cursor to the smaller circle, and again press or <C>. Now both circles are drawn solid. From this point, we can proceed with the two other views of the plate.

> The two other views are not required for constructing a parametric drawing in T-FLEX CAD. In this example, creating the side and the plan views simply help demonstrating additional advantages of parametric modeling using T-FLEX CAD system.

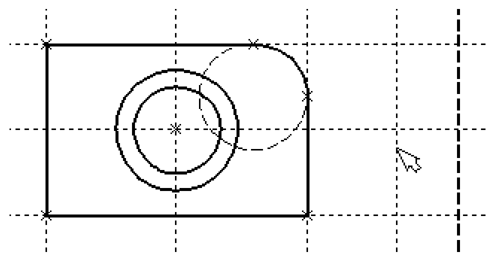
Since the straight lines are considered infinite, one can see that the other views (side and plan) are already partially created. To finalize the drawing, we will need to establish additional dependencies between the construction lines. These additional steps are described next.



Activate the line creation command and move the cursor to the construction line defining the right-hand side of the plate. Select it with . This highlights the vertical construction line and starts rubberbanding of a new line parallel to the selected one. This line will be the right-hand side of the plate side view. Fix it at a desired location by pressing . As before, the exact distance from the selected line can be specified in the property window.

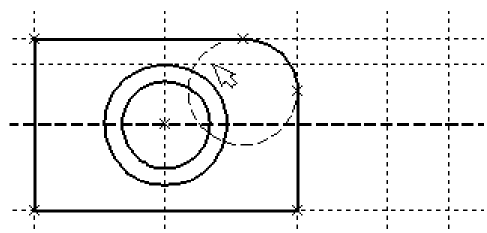
The new line is created relative to the right-hand side of the plate on the main view. Therefore, when the right-hand side of the plate is moved, the new line will follow, staying at the same distance. To place the new line at a different distance, use the command for editing construction lines. After that, moving the right-hand side of the plate will again preserve the new distance. The dependencies between construction entities stay valid until redefined in the construction line editing command.

The next step is creating the line of the left-hand-side edge of the part on the side view. After completing one line, a new line rubberbanding began automatically.



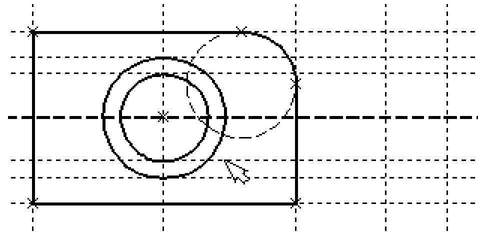
Note that the currently rubberbanded second line is also a dependent of the plate right-hand-side line as the latter is still highlighted. This is not our intention; therefore, press to start line creation anew. Select the last created line – the one marking the right-hand side of the side view - by clicking on it. Rubberband the new line up to the approximate location, and fix with , or enter exact value in the property window.

We recommend using specifically the right-hand side of the part main view as the base line, and construct all the rest vertical lines with respect to it. In this way, the line-to-line distances will be positive which is preferable in some situations.



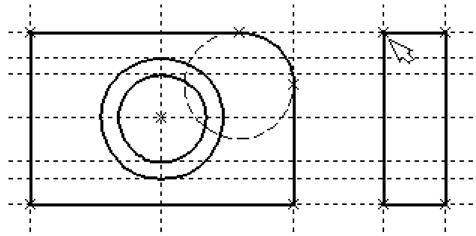
Now let's proceed with constructing the projection of the conical hole. First, let's create horizontal lines tangent to the top and bottom of the inner and outer circles of the hole. These lines will be used as guides for the side view of the hole.

Press once to restart line creation, move the cursor to the horizontal symmetry line, and select it by pressing or typing <L>. The line gets highlighted. Rubberband the new line by moving cursor to the outer circle and type <C>. The line is created parallel to the horizontal symmetry line and tangent to the circle.



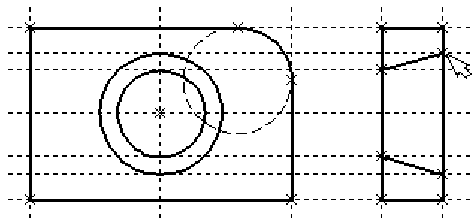
Repeat the same sequence of actions three more times: for the top tangency with the inner circle, and the two bottom tangencies.

Now we have the necessary guides for applying graphics on the side view.



Call **G: Create Graphic Line** command and apply solid lines between the four corners of the plate side view. Simply move the cursor from corner to corner clicking on each corner node, and then quit with .

Next, apply the two lines representing the conical hole. The view is now almost complete, with only the hatch yet to be created.



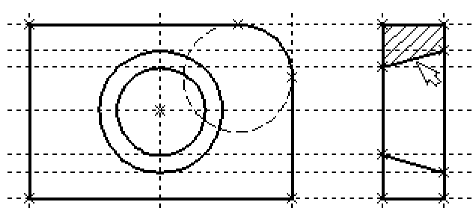
The hatch is created by **H: Create Hatch** command. Use

Icon	Ribbon
	Draw → Draw → Hatch
Keyboard	Textual Menu
<H>	Construct > Hatch

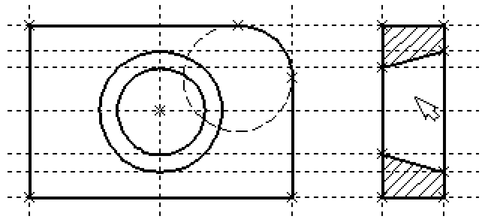
Set the following option, unless on by default,

	<A>	Automatic Contour search mode

Then move the cursor to the top portion of the side view, place within the area to be hatched, and press . The top contour gets highlighted. Next, type <P> to invoke the hatch area parameters dialog. Specify the type and scale factor of the hatch. Pressing in the automenu completes hatching of the selected area.

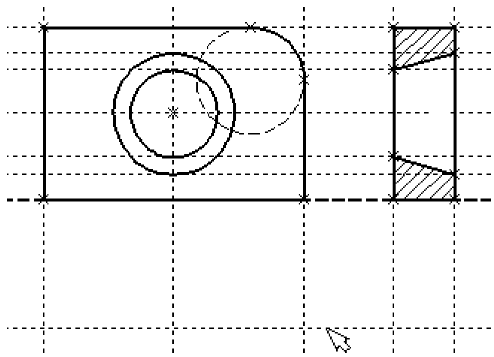


Repeat the same actions to hatch the bottom portion of the plate.



It is also possible to create a single hatch consisting of two contours, instead of creating two separate hatches. To do so, one could select the second contour right after the first one, and then press <End> key or pick icon in the automenu.

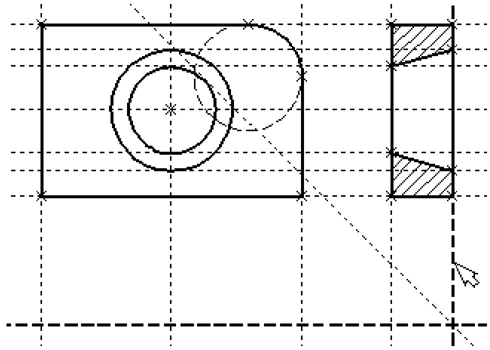
Once the hatch is created, proceed with the plan view. Call the line creation command "L: Construct Line". Select the bottom line of the main view in order to create dependency of the plan view on the main view. Rubberband the new line to a location below the main view and fix with . Then quit next parallel line creation with .



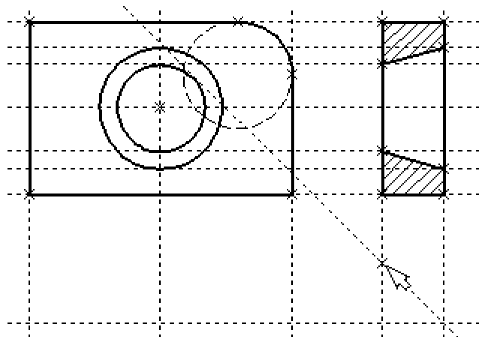
Let's try creating the plan view in such a way that modifications of other views propagated on the plan view via the established dependencies. The simplest way to create a dependency in projective drawing is creating a slanted line at 45-degree angle to the side lines of the side and plan view. The rest of auxiliary construction is done with respect to this slanted line.

Let's again use the symmetry line creation functionality, this time with a slanted symmetry line in mind. Since the lines of the side and the plan views are orthogonal, the resulting symmetry line will pass at the intended 45-degree angle.

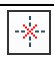
Call the option . Point at the right-hand-side line of the side view and select with or <L>. The line will get highlighted. Next, select the bottom line of the plan view by same means. A new line will be created passing through the intersection of the two selected lines at 45 degree to each.



Let's create all necessary nodes at intersections while within the line creation command. The relevant intersection points are those on the right-hand-side line of the side view and the newly created slanted line. To create a node, place the cursor at an intersection and press the <Space> bar.

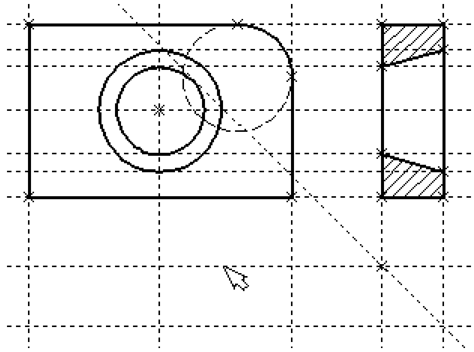


Another way of creating nodes is using command **N: Construct Node** via:

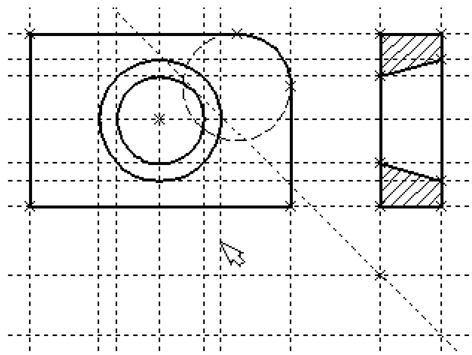
Icon	Ribbon
	Draw → Construct → Node
Keyboard	Textual Menu
<N>	Construct > Node

You should still be within the command "L: Construct Line". Point the cursor at and select the bottom line of the plan view. This way we can create a line parallel to the bottom-side one. Now, move the cursor to the newly created node and type <N>. This creates a line parallel to the selected one and passing through the specified node. Thus, the top and the side view become parametrically related.

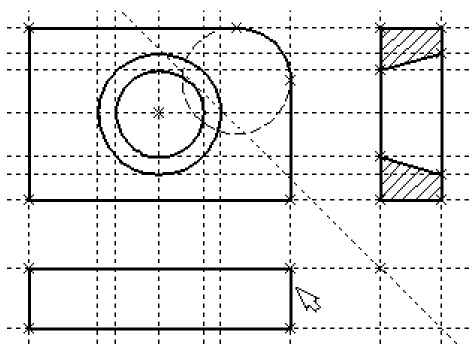
To witness this, call the construction line editing command **EC: Edit Construction.** Try changing location of the left-hand-side line of the side view. To do so, select it, move and fix in the new position. Note now that the corresponding line on the plan view moves accordingly.



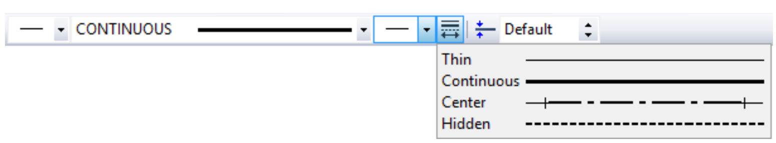
Construction of the conical hole on the plan view follows the same steps as on the side view. Select a vertical line while in the construction line creation command, and create four parallel lines tangent to the two circles.



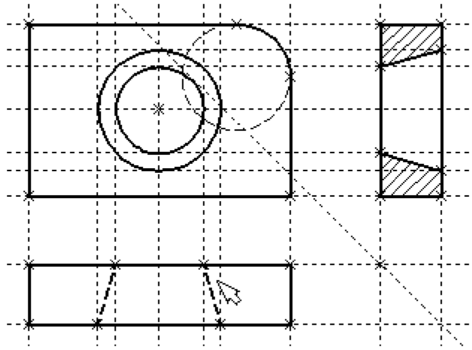
Now one can draw all graphic lines on the plan view. Use the command “G: Create Graphic Line” to draw the perimeter of the plan view.



Next step is to apply the two dashed lines corresponding to the conical hole. Set the “HIDDEN” line type in the system toolbar.



Then create the two dashed lines representing the conical hole.



Now, let's create centerlines. Call "**AX: Create Axis**" command:

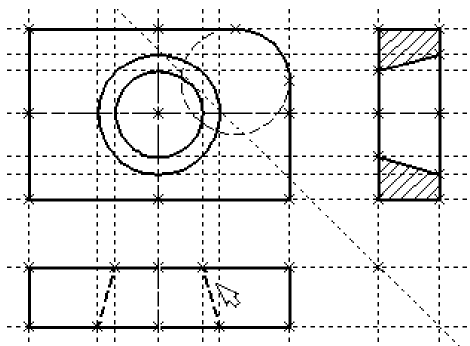
Icon	Ribbon
	Draw → Draw → Axis
Keyboard	**Textual Menu**
<AX>	Draw > Axis

Set the automenu option:

	<I>	Create Axis of two Graphics lines

Use to select the left and then the right-hand side of the elevation view. Push the automenu button. This creates a centerline on the elevation view. Similarly, create a horizontal centerline on the elevation view and those on the side and plan views.

One could notice that all construction lines created so far were infinite. For convenience, an option is provided in the command "EC: Edit Construction" for trimming construction lines at outermost nodes. This works as follows,

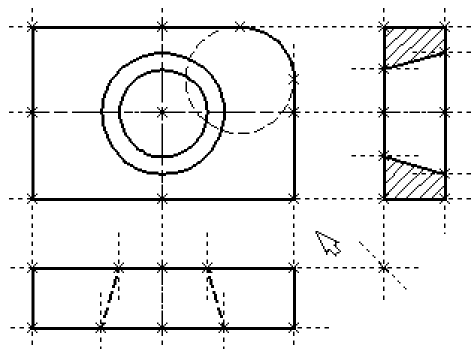


1. Call **EC: Edit Construction** command.
2. Selecting one particular line and typing <T> or pushing trims this selected line only.
3. Using option trims all lines.

4. If you want to revert to the infinite line setting, call the command **ST: Set Document Parameters**:

Icon	Ribbon
	Edit → Document → Document Parameters
Keyboard	Textual Menu
<ST>	Customize > Document Parameters

Go to the parameter View > Construction Lines > Length and set it to "Default infinite". Alternatively, enter the command EC: Edit Construction, select desired lines, type <P> and specify an appropriate setting.



The diagram shows a drawing with construction lines trimmed. It appears less crowded, although all necessary construction entities are present. By default, construction lines are not output to the printer or plotter, regardless of their length.

Next, let's create the necessary dimensions on the drawing as follows:

1. First, let's create linear dimensions. Call the command **D: Create Dimension:**

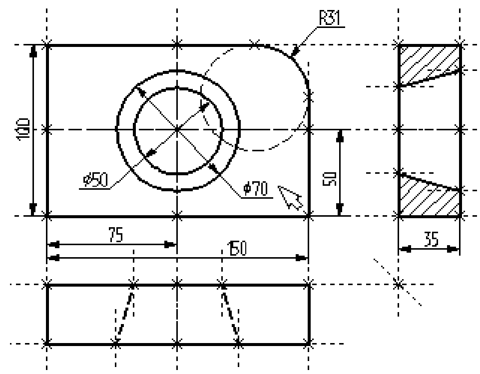
Icon	Ribbon
	Draw → Title Block → Dimension
Keyboard	Textual Menu
<D>	Construct > Dimension

One can select any pair of construction lines or graphic lines to create a linear or angular dimension. Select the two outermost lines on the main view by . This starts rubberbanding of a dimension following the cursor motion. To change any dimension parameter, type <P> or push the button in the automenu. The dimension parameters dialog box will appear on screen. Specify the desired parameters, close the dialog, and fix dimension placement with . To change the size of the dimension string font, use the command **ST: Set Document Parameters**, the **Font** tab. The font parameters can be specified on this tab for the elements that did not have such parameters set originally.

2. Repeat the steps of the item 1 for the rest of the linear dimensions.

3. Diameter and radius dimension creation is also straightforward. While in **D: Create Dimension**

command, move the cursor to a circle to be dimensioned, and type <C> or click . The circle gets selected, and a dimension begins rubberbanding after the cursor. Switch between the radius and diameter dimension types by typing <R> and <D> or picking and buttons of the automenu as appropriate. Typing <M> loops through the possible witness/leader line configurations for the entity to be dimensioned. The <Tab> key handles the direction of the dimension leader line jog. Point the cursor at the desired location and press . The newly created dimension will be displayed on the screen. Repeat this procedure to dimension all circles.

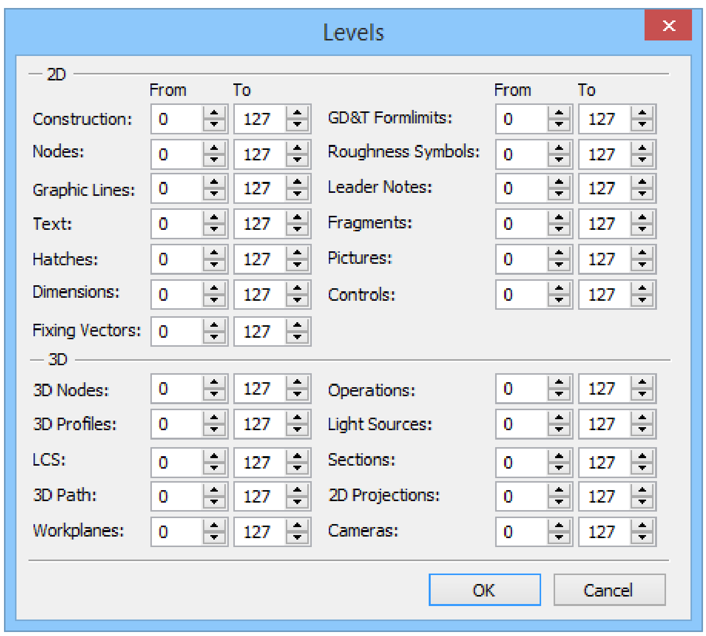


After finishing construction of all major elements, one can hide all construction entities using the command **SH: Set Levels**:

Icon	Ribbon
	Edit → Document → Levels
Keyboard	Textual Menu
<SH>	Customize > Levels

This command controls visibility of various elements. An element visibility depends on the “level” at which it is residing.

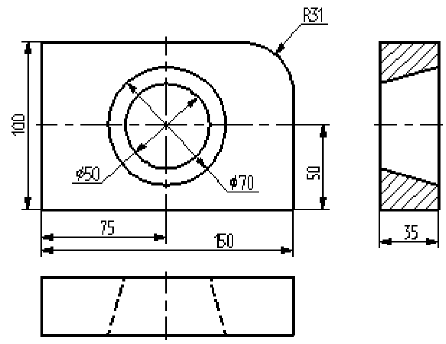
After calling the command, a dialog box appears on screen that allows setting a range of visible levels for each element type of the model.



Think of levels as transparent films with images drawn on them. The complete drawing consists of all of them overlapped. The system permits making one or more levels invisible, displaying only intended ones. A drawing may consist from up to 255 levels enumerated from -126 to 127.

All elements in T-FLEX CAD are automatically created on the level “0”. One can re-assign any element to another level at any time. In our example, we did not change levels of any element; therefore, all created elements fell in the level “0”.

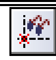
As appears on the diagram above, all elements are visible by default whose level is in the range from 0 to 127. Setting the low limit of the visible range to 1 for construction lines and nodes hides the construction lines and nodes, because they reside on the level 0 which is not within the new range.



A simpler way to hide construction lines and nodes is to use a dedicated command. This command hides or shows all construction entities in the current window. It is preferable in the situations when hiding construction should not affect the document data, rather, the current window only.

It is thus possible to open the same document in several windows, and have construction entities displayed in some windows, and hidden in others.

Call the command via

Keyboard	Textual Menu	Icon
<Ctrl><Shift><C>	View > Hide Construction	

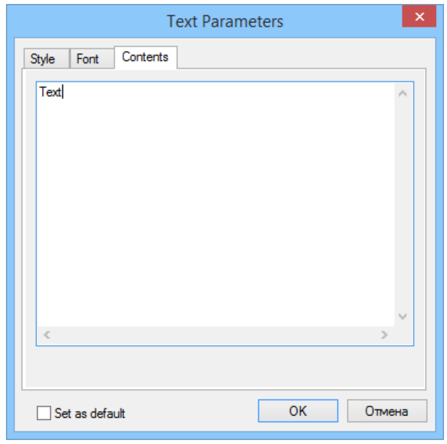
5. Let’s make a line of text containing the name of the drawing using command “**TE: Create Text**”. Call the command via

Icon	Ribbon
	Draw → Title Block → Text
Keyboard	Textual Menu
<TE>	Construct > Text

In the automenu of the command, turn on the option:

	<D>	Create string text

A text can be “snapped” to any construction entity on the drawing in order to have it move together with the drawing elements being modified.



Move cursor to the intersection of the vertical centerline and the top line on the main view. Type <N> in order to snap the text to the node at the intersection. Move the cursor to the text placement point and press 🖱. The text editor window appears on screen. Type a line of text "*Sample Plate Drawing*" and push **[OK]** button.

Should the text not be placed as intended, this can be corrected easily. Quit the text creation command. Point and click 🖱 at the text. This automatically starts the editing command **ET: Edit Text**. The selected text starts moving after the cursor. Locate it as desired and click 🖱.

To explicitly call the text editing command, use

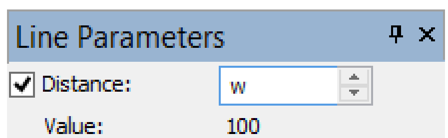
Keyboard	Textual Menu	Icon
<ET>	Edit > Draw > Text	

In this case, select the text to be edited after launching the command.

There is another way of creating a text, which is typing it directly in the drawing area. To do so, enter the **"TE: Create Text"** command and set the option <T> **-** "Create paragraph text" (icon). Move the cursor to the intended location of the text and press 🖱. A rectangle starts rubberbanding on screen that defines the text box. Define the intended area and click 🖱, then push the icon. A blinking cursor will appear in the box. Make sure of the correct input locale and enter the intended text. Then push the icon or <F5> key.

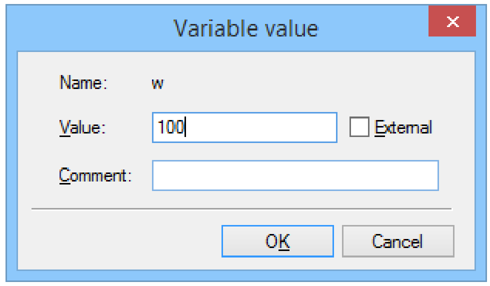
The drawing is now finished. One can try moving construction lines using construction editing command. When editing, fix line new placement by either using 🖱 or specifying exact line location in the property window or parameters dialog (the latter accessible via the pick). Note that the whole drawing, including dimensions, adequately responds to modifications. Changing diameters of the conical hole instantly reflects on the two other views. Hatches also adjust to their defining contours. Now one can easily witness the powerful capabilities brought in by the parametric technology.

From now, we will assign variables and expressions to the various drawing elements. Select the left-hand-side line on the main view by clicking 🖱.



The line will get highlighted, along with the one it is dependent on by construction. Line editing command will automatically activate as well. The two parameters are displayed in the property window. The first one is the original distance, and the second is the current value according to the cursor position.

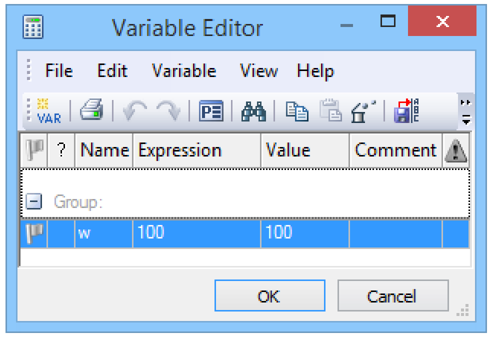
Since the line was originally created as parallel to the left-hand side of the plate, the displayed distance is the distance between the right and the left-hand side of the plate. Instead of a specific value, one can input a



variable. Type a variable name "*w*" instead of the value and press <Enter> or **[OK]**.

A new dialog window will appear requesting a confirmation for the new variable to be created.

Please note that the variable naming is case-sensitive. A variable "*w*" is not the same as "*W*".



The created variable *«w»* and the value assigned for this variable can be seen in the window "Variables" located, by default, under the properties window. Point with a cursor at the number in the column "Expression", press to enter the edit mode and specify the value for the variable, for example, *«170»*. The line will move to a different location corresponding to the new value of the plate width.

The same operations can be carried out in the dialog window of the command **"V: Edit Variables"**:

Icon	Ribbon
	Title Block → Additional → Variables
Keyboard	Textual Menu
<V>	Parameters > Variables

Similarly, define a variable "*H*" as the distance from the base line to the top side of the main view. Select the line on the drawing by clicking and enter the variable name in the property window. Now there will be already two variables in the window "Variables", and you can, by modifying their values, observe the change in the drawing.

Try making an expression. In the window "Variables" place the cursor in the field "Expression" of the variable *«H»* and press to enter the edit mode. Specify the expression *«w/2»* instead of the numerical value. This means that the value of "H" will be equal to the half of "*w*". From now on, changing just the value of "*w*" will automatically reflect on the value of "H".

Next, let's assign an "*R*" variable to the radius of the circle defining the fillet at the upper-right corner of the plate. Select the circle on the drawing by . In the property window specify the radius as *"R" variable*. After confirming its creation, in the window "Variables" set the variable to the following expression: *w < 100 ? 0 : 6*

This expression means that *"R"* equals 0 when *"w"* is less than 100, and equals 6 otherwise.

Let's briefly explain the syntax of the expression. Its members are described as follows.

< - is the "less than" sign

? - means "then", "in such a case"

: - means "else", "otherwise"

The complete expression is written as

R = w < 100 ? 0 : 6

The value of *"R"* equals 0, if *"w"* < 100, and equals 6 for any other value of *"w"*. Therefore, there are only two possible values of *"R"* - either *"0"* or *"6"*.

Check this on your drawing. Try setting *"w"* values greater or less than 100, and watch what's happening. Note that when the radius of the fillet equals *0*, then the radial dimension automatically disappears. The system does it for the user.

Therefore, one can create a variety of relations between variables, including quite complex ones, using just a few basic terms. You will get to know all capabilities of the variables functionality in later chapters.

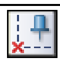
Creating Sketch, Non-parametric Drawing

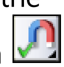
We will use the same familiar drawing example of the plate with a conical hole. Let's begin with constructing the main view. Thereafter, we will create two projections, the "Left Side View" and the "Plan View", using object snapping mechanism.

In this case, all construction is done using the command **SK: Create Sketch.** Call the command via

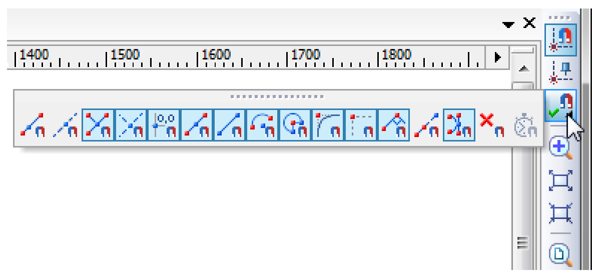
Icon	Ribbon
	Draw → Sketch
Keyboard	Textual Menu
<SK>	Construct > Sketch

This command can be used to create either a sketch (nonparametric drawing) or a parametric drawing in the automatic parameterization mode. Since we are going to create specifically a sketch, please make sure that the automatic parameterization icon is switched off on the "View" toolbar

	Automatic parameterization on/off

When creating a sketch, object snaps are widely used. The control over snaps is performed with the "Snaps" toolbar. To access this toolbar, press the icon  on the toolbar "View".

All snaps turned on by the present moment correspond to the toolbar icons which have been pressed.



To turn off a pushed option, point at and click on the respective icon. To turn off snapping completely, set the option:

		Clear all sketch Snaps

Unsetting this option sets all snaps on. In our exercise, the following snaps will be used:

		Line Midpoint
		Horizontal / Vertical
		Orthogonal
		Line Intersection
		Horizontal/Vertical tangent

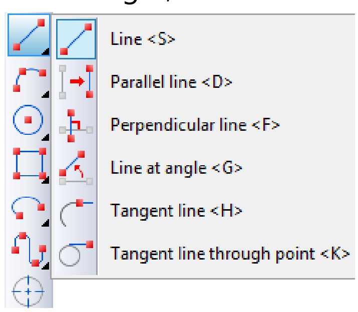
Push these icons on the **Snaps** toolbar. The object snaps can also be managed within the command **SO: Set System Options**, using **Snap** tab.

When creating line segments, arcs and circles, the point coordinates can be defined by simply clicking in the drawing area. To specify exact node coordinates, one can use the property window.

The two options are turned on automatically in the automenu while within the sketch creation command:

	<J>	Continuous creation
	<S>	Line

The first icon allows drawing continuously, so that the end of a just created line becomes the start of the new line. This mode will stay active until the user turns it off by pointing at the icon and clicking . We recommend keeping this option on for speedy sketching. The other option sets the segment input mode. A black triangle in the lower-right corner of the icon marks availability of more options. To access these encapsulated options, press and hold a bit longer, and a menu of options will appear.



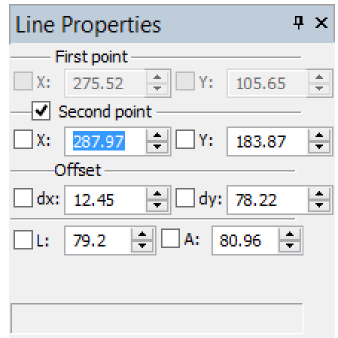
> Attention: the automenu may display any of the encapsulated option icons in the given position. Usually this is the icon of the last used option among the encapsulated set.

In “Sketch” automenu, each set of encapsulated options corresponds to a group of actions related to same-type element creation, such as creating segments, arcs, or circles.

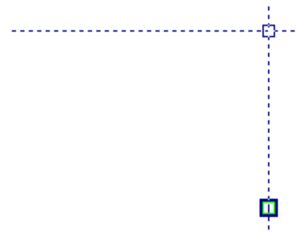
The cursor on screen appears as a little square. Move the cursor to the intended location at the lower-right corner of the main view, near the middle of the screen, and press . This creates the first node of a line segment and starts rubberbanding the segment to be created. At the same time, the fixed coordinates of the first point are displayed in the property window.

> While sketching, consider leaving sufficient margins. This space will later be used for creating dimensions.

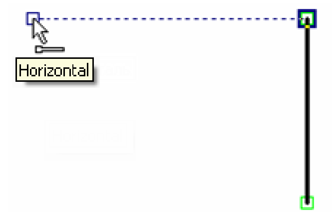
Move the cursor upward. Note that the coordinates of the cursor are dynamically updated in the property window, along with the vertical and horizontal shifts from the start node of the segment. We can use the property window for specifying exact location of the segment end. The end node can be defined in several ways. One way is to enter absolute Cartesian coordinates (X, Y) of the segment end node. Another way is specifying X and Y shifts of the end node from the segment start (dx, dy). Yet another way is to define the end of the segment in polar coordinates, or as a combination of the other ways.



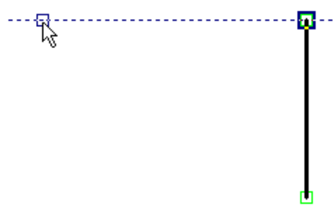
Let's create the end of the segment by specifying its shift from the start point. Make "dx" value equal to 0, and "dy" parameter equal to 100. The parameters "X" и "Y" will instantly update with the absolute coordinates of the segment end and get checkmarked. Checkmarking locks the value of the respective coordinate in spite of cursor movements. The end node of the segment will be displayed in the drawing window per the entered coordinate values.



To complete the end node input, press **[Enter]** or click in the drawing area. The first segment will be created. Move the cursor leftwards and place it so that it snaps to the horizontal constraint with respect to the last created node. The snap will be indicated by the specific glyph next to the cursor, and a pop-up help message saying "Horizontal".

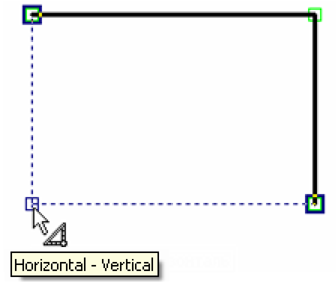


To lock this snap, press <Space> bar. Then, a horizontal helper line will be displayed passing through the node snapped to. The cursor will keep sliding along this helper line as an unattached node. The same effect can be achieved by setting the "dx" shift to 0 in the property window and locking the X coordinate with the checkmark.

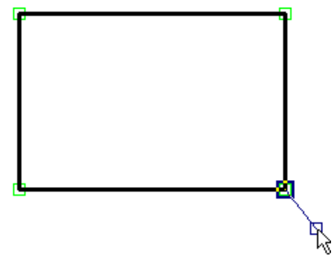


Place cursor on the side of the segment intended direction. Type the value of the “dx” shift for the end node of the segment being created in the property window. In our case, this value represents the width of the part and is equal to -150, while “dy” equals 0. The new segment will be created upon confirmation by pressing **[Enter]** or .

You are still within the segment creation mode. For further construction, you need to move the cursor downward to a point where snapping occurs to both horizontal and vertical constraints simultaneously. This will be indicated by a special glyph next to cursor and pop-up help message. Press . A new segment is created.



Move the cursor rightward to snap at the very first created node, as indicated by a glyph and the pop-up help, and press . We have thus completed the perimeter of the main view. We are still in line rubberbanding mode, with a new line stretching from the last created node to the cursor. Quit this line rubberbanding by clicking .



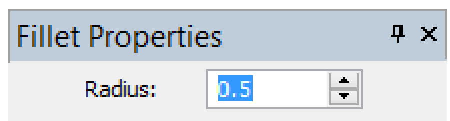
Now you are still within continuous segment input mode, with snapping active but no line rubberbanding after the cursor.

The next step is to round the corner of the plate. To do so, set the option:

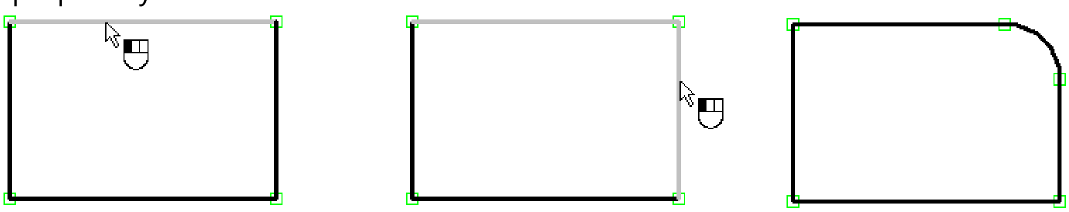
	<Ctrl+A>	Fillet

This option belongs to an encapsulated set and may not necessarily be displayed in the automenu. Instead, it may be under the fillet/chamfer icon group (see explanation above).

Once the option is set, the property window changes appearance. Now it provides the input box for the fillet radius. Set the radius value equal to 31.



What is left now is to select two segments to be filleted. In our case, it is the top and the right-hand-side segments of the plate. Once the second segment is selected, the fillet is created, and the segments trimmed appropriately.



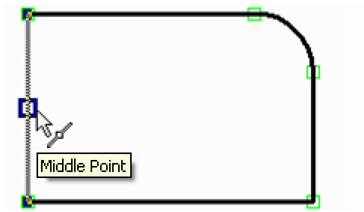
Now let’s draw the conical hole on the main view. To do so, let’s create two centerlines whose intersection will define the exact location of the center of the hole. Set the option

	<S>	Line

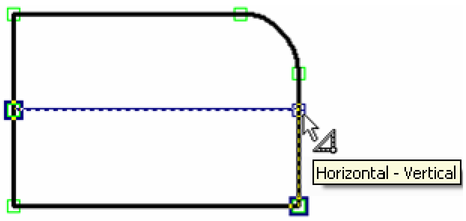
Once this is set, rubberbanding resumes with a line attached to the last created node. Reject this line by clicking . To create centerlines, set the appropriate line type. Set the line type to DASHDOT in the system toolbar or in the graphic line parameters of the dialog box. Call the dialog box by

	<P>	Set Graphic line Parameters

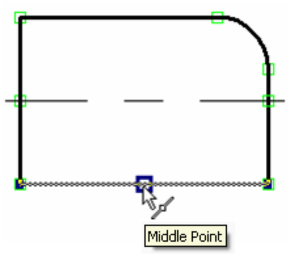
Then move the cursor to the left-hand-side graphic line segment to get vertical snapping to one of the segment nodes, and slide the cursor along the segment to its midpoint. When the cursor reaches the midpoint, the glyph beside the cursor will change to indicate this, along with the pop-up help.

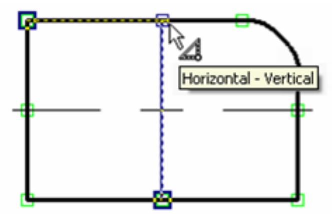


Press here. A node will be created at the midpoint, and a line will start rubberbanding from this node. Move the cursor horizontally to the right-hand-side graphic segment and stop at the intersection of the horizontal and vertical projections of the two nodes as shown on the diagram.



Press , creating a centerline and a node. Rubberbanding resumes from the latest node. As we do not intend to construct another line through this node, quit rubberbanding with . Then follow the same steps to create the vertical centerline, beginning at the bottom segment.

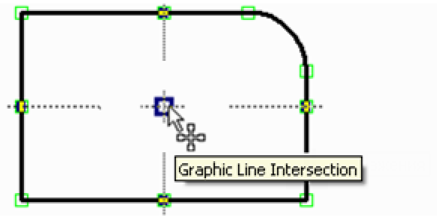




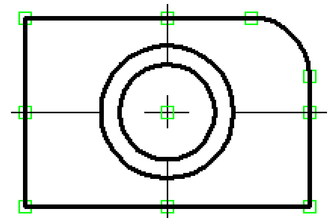
Now, let's create circles. First, reset the line type to CONTINUOUS by selecting in the system toolbar or in the command parameters dialog box. Call the dialog by typing <P>. Then pick the option:

	<O>	Circle By Center And Radius

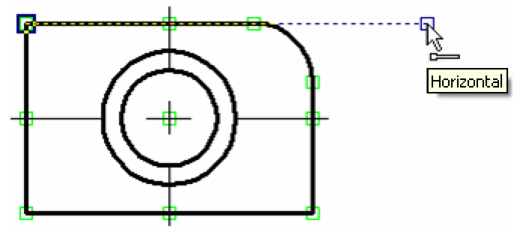
This is also an encapsulated option that may not be shown on the automenu, rather, be within a group of options.



After activating the option, move the cursor to the intersection of the two centerlines. Both centerlines will get highlighted, and the cursor will gain a glyph and a pop-up help of the graphic line intersection snap. Press here. A circle starts rubberbanding on screen.

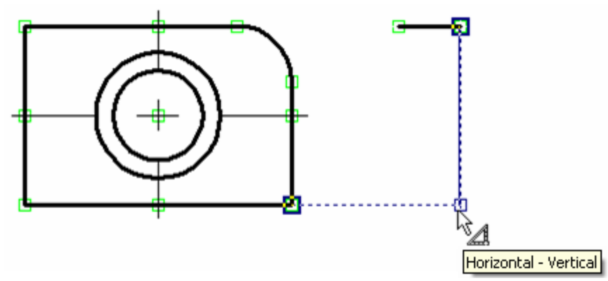


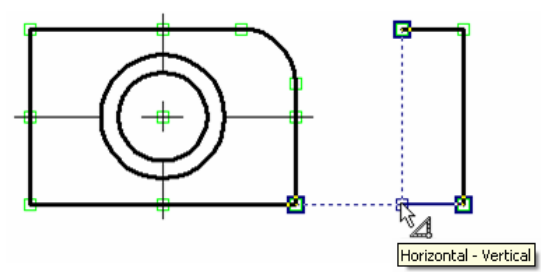
Enter the value of the radius of the smaller circle of the cone equal to 25 in the property window, and press **[Enter]** key. A full circle is now fixed on screen. Now you are still in the circle creation mode. Select the node at the two centerlines intersection and create a circle of a bigger radius, 35. This mostly completes creation of the main view of the part.



Now, let's construct the left side view. Again, set the line creation mode via the option. Line rubberbanding resumes on screen with the line attached to the end node of the last created segment. Since we are not making a line from this node, quit with . Move the cursor to the right of the drawing area and place it so to snap to the horizontal constraint with respect to a node of the top line of the main view.

Click here , and move the cursor horizontally rightwards. In the property window, set the end node shifts. Set the X equal to 35, Y to 0. Press **[Enter]** or . The new segment will be fixed on screen, and rubberbanding resume from the last created node. Next, move the cursor vertically downward maintaining the "vertical" snap, up to the point when snapping occurs with a node of the bottom line of the main view. Click there , then move the cursor leftwards to snap against the left-hand-side end of the top segment.

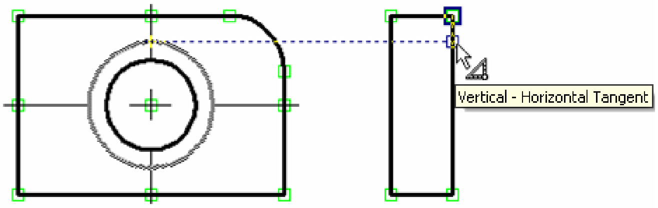




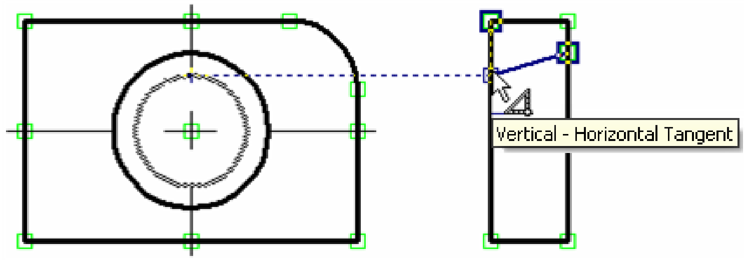
Click . Now close the perimeter of graphic lines by moving cursor to the first created node on this view, and clicking , and then .

> One-degree-of-freedom snaps can be locked by pressing <**Space**> bar.

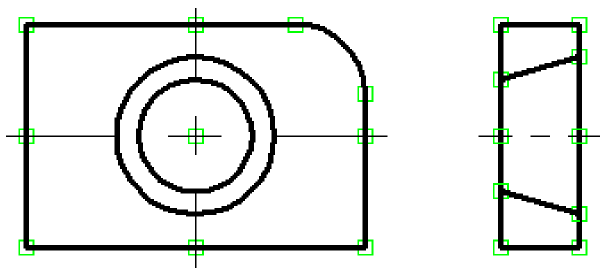
Next, we need to create the image of the conical hole on the side view. Without quitting the current command, move the cursor to the right-hand-side segment of the side view, and move along the segment until it snaps to horizontal tangency against the top of the bigger circle



Click at this spot, then move the cursor to the left-hand-side segment of the side view and locate so to get it snapped against the smaller circle.



Click , and a segment will be created, with rubberbanding resuming from its end node. For now, quit rubberbanding by clicking . Then construct the lower segment of the hole view in the same way. Next, using already familiar snapping constraints, construct the centerline, setting the line type to DASHDOT in the graphic line parameter dialog box (the key ***<P>***) or in the system toolbar.

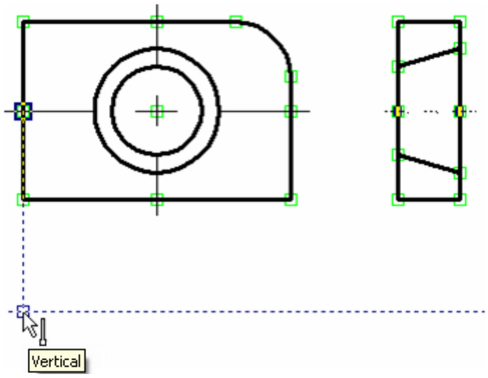


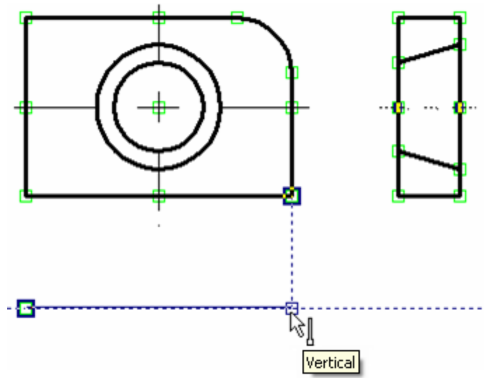
Proceed with the plan view. This view can be created in the same way as the side view. However, for deeper exploration of non-parametric drawing capabilities, we will follow a different approach. Set the option:

	<D>	Parallel Line

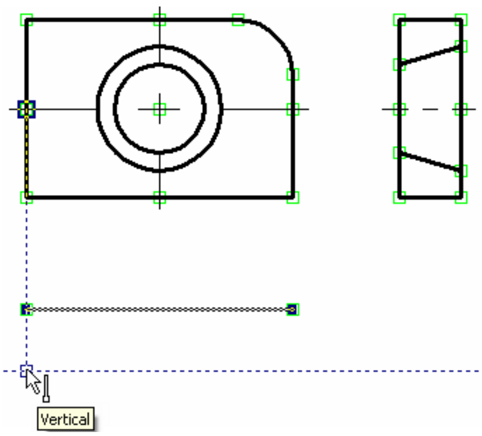
This is an encapsulated option in the segment creation group. If this icon is not displayed in the automenu, find it under one of the group icons marked with a black triangle (see explanation above).

Once this option is set, the cursor starts rubberbanding an auxiliary infinite line parallel to the last created segment. The reference segment is highlighted. The current reference suites us. Otherwise, we would reject the system-selected segment with and select an intended one to construct a parallel line. Make sure the line type is back to CONTINUOUS in the graphic line parameter dialog (the key <P>) or in the system toolbar. Move the cursor to get a snap against a node of the main view, and click . A node will be created at this point, and the auxiliary line will get fixed. Slide the cursor along the line up to the point of another vertical snapping, and again click .

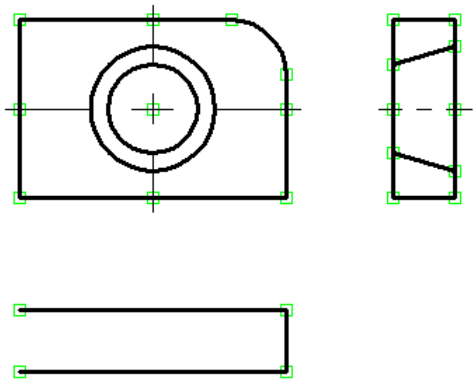


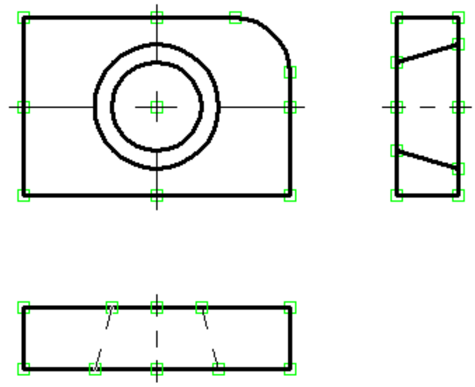


Thus, we have created the top segment of the plan view. A new auxiliary line starts rubberbanding after the cursor, parallel to the newly created graphic segment, as indicated by highlighting. Move the cursor down, and set the desired distance, equal to 35, in the property window, thus defining the thickness of the plate. This will fix the auxiliary line with respect to the reference segment. Slide the cursor along the line, locating as shown on the diagram.



Click , fixing the start node of the segment being created. Slide the cursor rightwards to get vertical snap against the end node of the reference segment, and again click . The bottom segment will be created. At this moment, parallel line rubberbanding resumes. Now, set the option , thus switching to normal continuous line input mode. A line will start rubberbanding, attached to the last created node. Move the cursor upward to the top segment node, and click , and then . Next, connect the left-hand-side ends on the plan view with another segment. Create the centerline and the lines of the conical hole projection in the same way as on the side view. Doing so, maintain the appropriate line types.





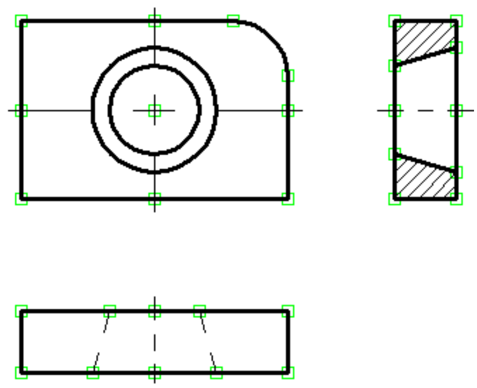
What is left now is to apply hatch on the side view. Call the command "**H: Create Hatch**":

	<H>	Create Hatch

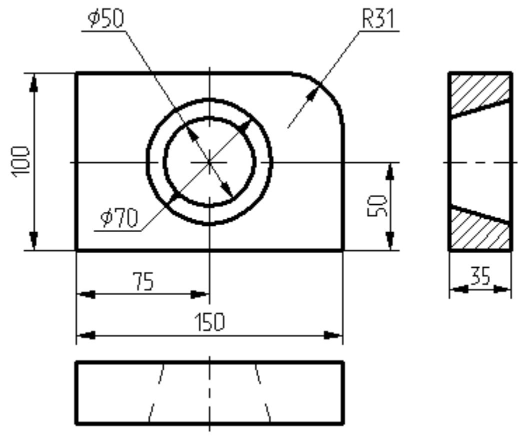
Set the option:

	<A>	Automatic Contour search mode

Then move the cursor to the top portion of the side view, place it in the middle of the area to be hatched, and click . The closed contour will be highlighted. Now move the cursor to the lower portion of the view, and similarly select the other contour to be hatched. Then pick the button.



Now, let's create the necessary dimensions on the drawing. Dimensions are created on a sketch in the same way as on a parametric drawing. One can select graphic lines instead of construction lines in this case. Let's skip the detailed description of this functionality, as it was described in depth as part of the main drawing technique.



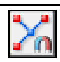
This completes creation of the non-parametric drawing. Further modification of its elements will not affect the whole drawing. One would have to modify each view separately. The elements of such a drawing cannot be related by variables. All other functionalities such as use of visibility levels, layers, hide/show construction entities, etc. are fully supported.

Creating a parametric drawing in the automatic parameterization mode

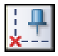
We will use the same drawing as an example. The sequence of constructions will be the same as described in the previous section of this chapter.

When working in the automatic parameterization mode, we will be creating only graphic lines (as when constructing a sketch). Meanwhile, the system will be automatically "slipping" nodes and construction lines with parametric relations underneath those graphic lines. What constructions to create and what dependencies to use in relations is determined by the system based on the user-selected snaps and parameters defined in the command's property window when creating sketched lines.

Call the command “**SK: Create Sketch**”. Make sure that the following snaps are enabled:

	Construction
	Line Midpoint
	Horizontal/Vertical
	Orthogonal
	Graphic Line Intersection
	Horizontal/Vertical tangent

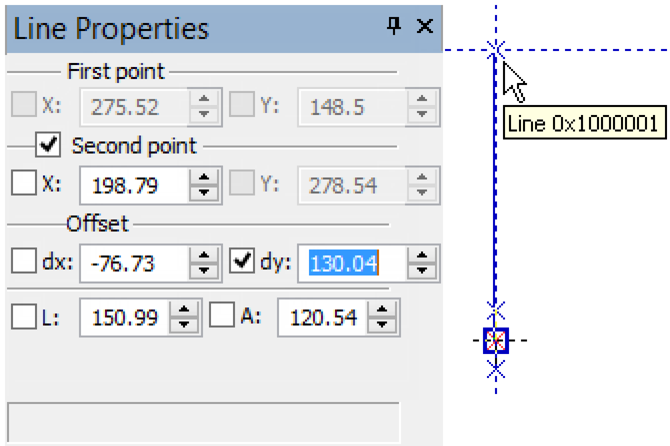
To create a parametric drawing, the automatic parameterization mode must be turned On in the system. This mode is enabled with the icon on the "View" bar:

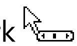
	Automatic parameterization On/Off

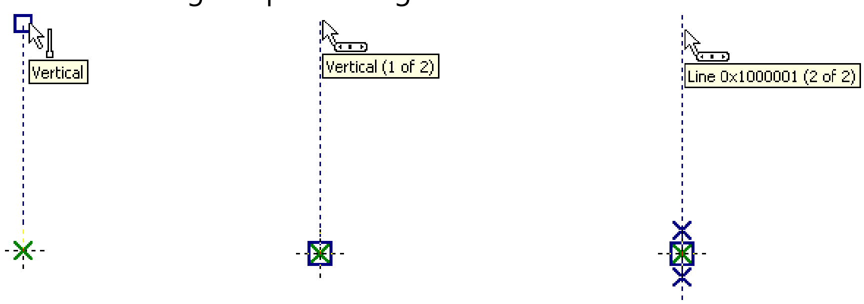
We will start with creating the main view of the plate. If necessary, enable the line segment creation option  in the command's automenu (if desired). Create the first point of the segment, corresponding to the lower-right corner of the plate's main view. Please note that it was not a free node that was created at the location of the click, rather, there are two crossed lines (vertical and horizontal), and a node at their intersection.

For the second node of the segment, define the Y-axis offset (100) in the properties window. The cursor will start moving along a horizontal auxiliary line. Move it to the vertical construction line going through the first segment node. When the snap to the latter line engages ("Line ..."), click .

As a result, the second node of the created segment will also be constructed as snapped. It will lie on the intersection of the vertical line created at the time of constructing the first node, and a line parallel to the horizontal line of the first node.

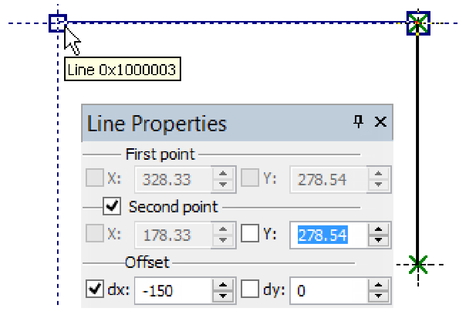


Please note, that, when selecting a snap, the system may offer the vertical snap to the first segment node rather than snapping to the line (the order of displaying snaps is determined by the settings in the command **SO: Set System Options**). To select the desired snap, do the following: briefly rest the cursor at the location of the snap activation. After a brief while, the cursor will change its shape: the mark  and a tooltip will appear next to it, showing the total number of object snaps found at this point. Using the mouse wheel, you can scroll through those snaps. In the ongoing construction, select the desired snap from the list of possible ones at a given point using the same method.

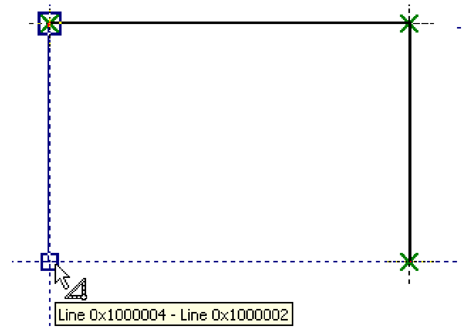


Create the second, horizontal, line segment with the length 150. When constructing it, specify the desired offset along the X-axis and use a snap to the line again.

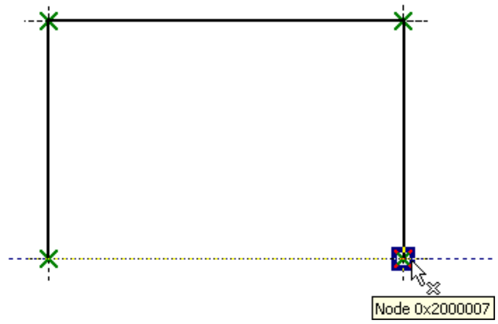
If all was done correctly, then the created line segment will lie on the line created at the time of constructing the second node of the previous segment. Meanwhile, the second node of the current segment will be constructed as one lying on the intersection of the same line and a new line parallel to the very first vertical line.



The third line segment, again vertical, is constructed by snapping to two existing lines at once.



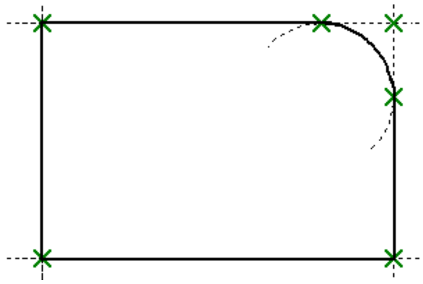
The fourth segment must be closing the described rectangle. After that, move the cursor rightward up to the first created node, as indicated by a special mark in the dynamic tooltip, and click . This completes the base for the part's main view.



Please note, that the resulting drawing we obtained is the same as when constructing a parametric drawing by the conventional approach (as was described in the first section of this chapter).

Just like when creating a sketch, to create a fillet one would need to quit the continuous line input mode (using).

To create the fillet, let's use the option . After activating the option, set the fillet radius equal to 31 in the properties window.



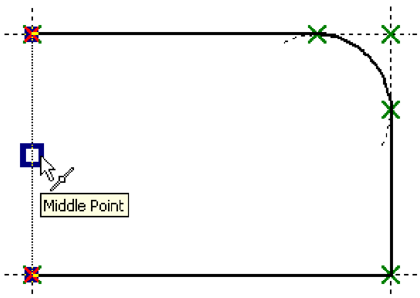
After that, select two segments, at whose intersection the fillet needs to be constructed (the top and the right-hand-side segments of the plate) or the node (the rectangle vertex) at their intersection.

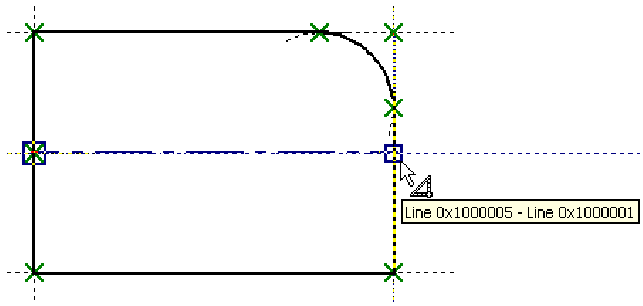
This will result in constructing a graphic line – a circular arc with a "slipped underneath" construction line-circle. Just like when creating a common sketch, the extra pieces of the fillet segments will be automatically trimmed.

Next, we will create the image of the conical hole.

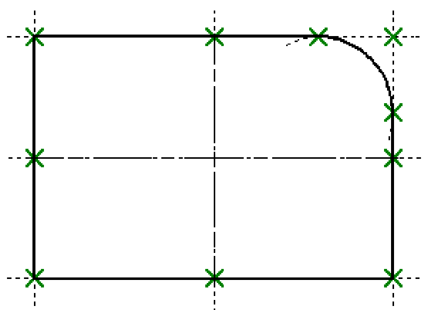
We will start with creating the axes. To create the axes, enable the option again. Set the “Axis” line type in the system panel or in the graphic line parameters (the option).

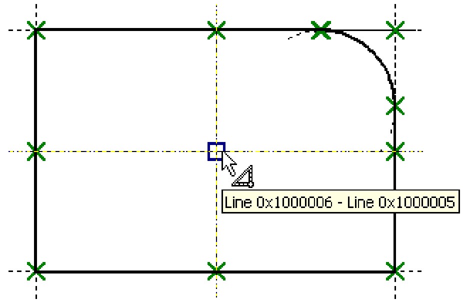
Move the cursor to the middle of the left-hand-side segment of the plate's main view image. Construct the first node of the axis using the line midpoint snap. Move the cursor horizontally to the right-hand-side segment of the image and stop it at the intersection of the two lines as shown on the figure. Click . The created segment will lie on the line that divides the segment (the left-hand side of the main view) in the 0.5 ratio.





Similarly construct the vertical axis.

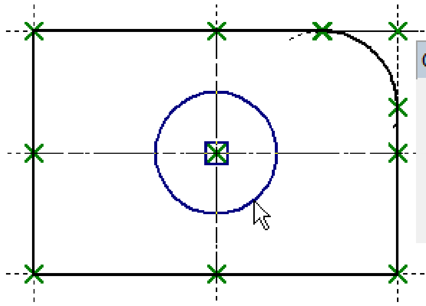


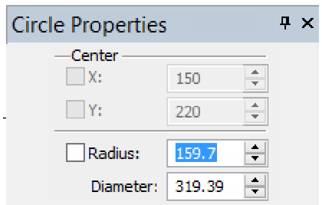


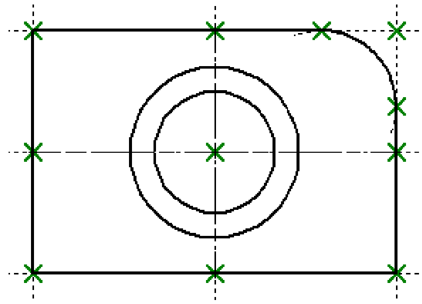
Now, let's create the circles. Set the normal graphic line type. After that, select the option . Move the cursor to the intersection of the axial lines, wait until the tooltip appears indicating the available snapping to the intersection lines. Click right there. A rubberbanding circle will appear on the screen.

Set the radius value equal to 25 for the smaller circle of the conical hole and click or press the button **[Enter]**. Similarly construct the second circle with the radius equal to 35.

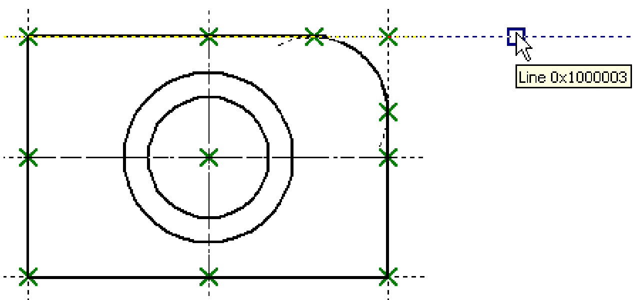
Please note that the construction result is not just free graphic lines representing the circles. The system constructed them as lying on the construction lines-circles.





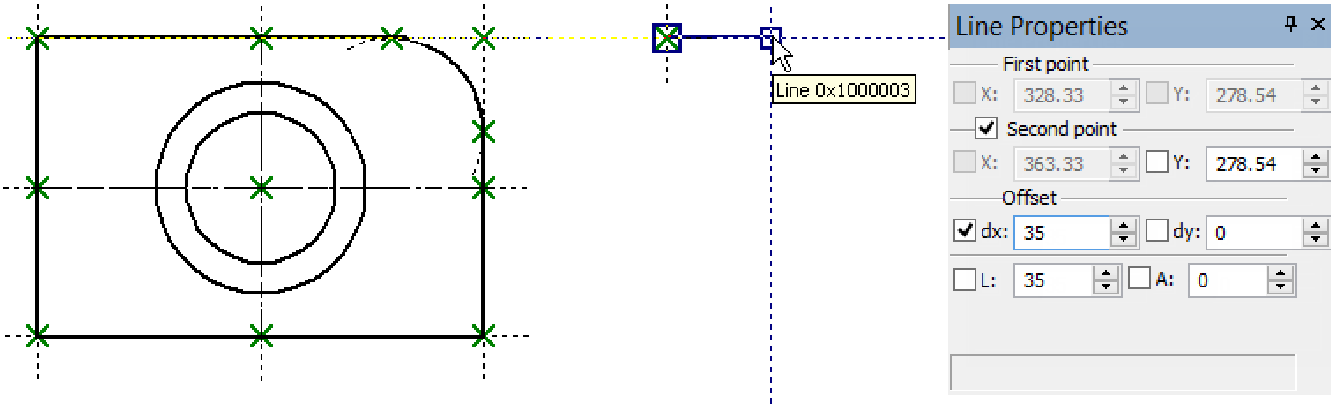


Now let's create the left view. To do that, enable the segment creation mode again (the option). If the system offers creating a segment from the last created node, refuse that by right-clicking . Move the cursor to the right-hand side of the drawing and set it so as to maintain the snap to the top line of the main drawing view.

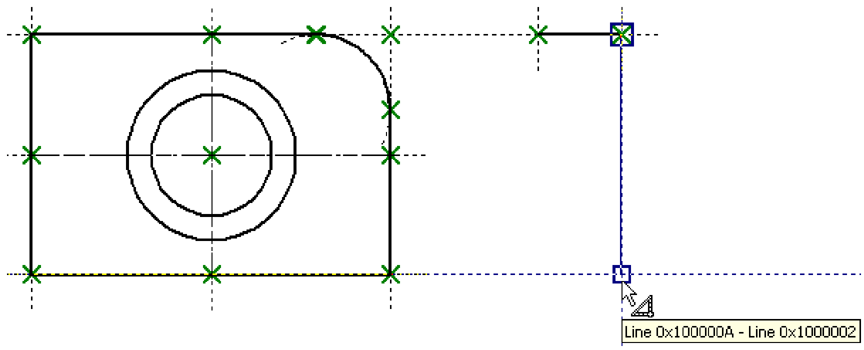


Click there . The first node of the new segment will be constructed as lying on the selected construction line.

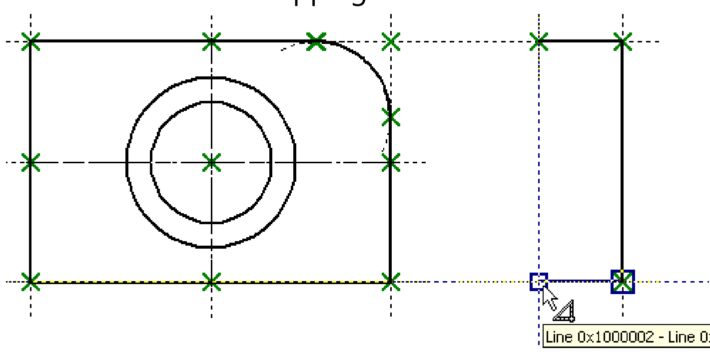
Move the cursor horizontally rightward. In the properties window set the offset of the second point along the X-axis equal to 35. Then move the cursor so as to pick the snap to the top line of the main view. Click . As a result, the second segment node will also lie on the top line of the main view at the distance 35 from the first node.



Then move the cursor downward vertically to the last created node up until a snap to two lines appears on the screen.

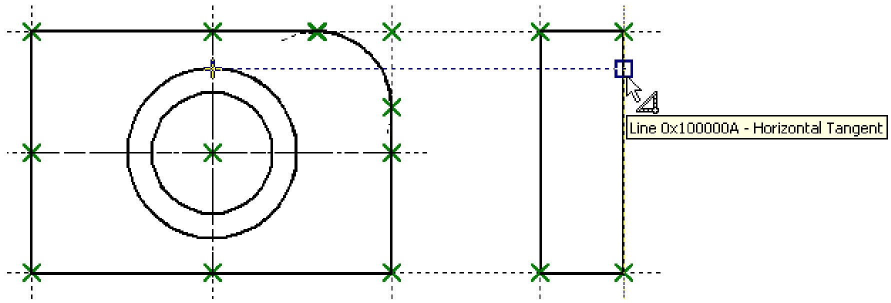


Click and move the cursor leftwards till snapping to two other lines.

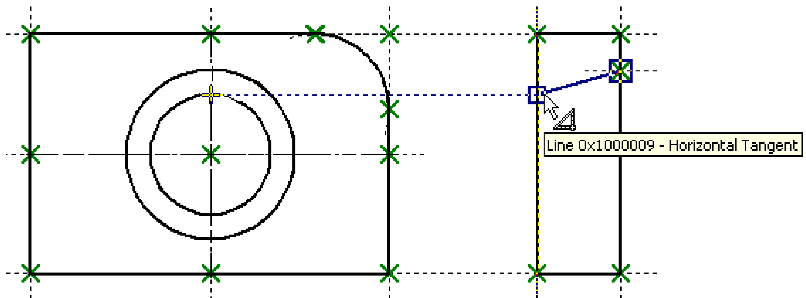


Click . Now, close the created graphic lines by moving the cursor to the first created node on this view, and click , then (to cancel the mode of continuous line input).

After that, we will create the lines belonging to the conical hole, on the left view. Move the cursor to the right-hand-side segment of the left view, and then move it along that segment up until establishing the relation between the line underlying that segment and the greater circle. Click right there.

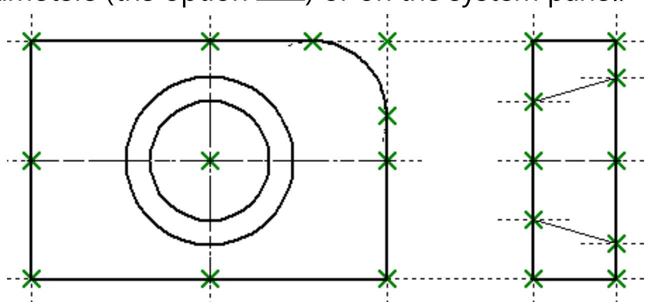


Move the cursor to the left-hand-side segment of the same view so as to establish the relation between the smaller circle and the line underlying that segment. Click .



As a result, a segment will be created. No construction line will be underlying that segment. Nevertheless, each segment node will be constructed as a node at an intersection of the selected line and the line tangent to the circle.

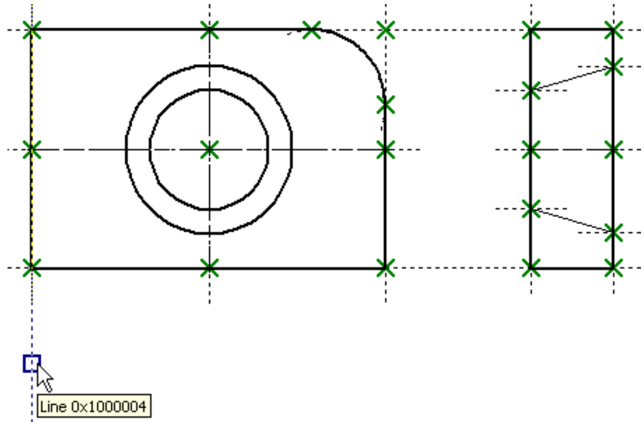
Next, use the same method to construct the lower line of the conical hole. Then create the centerline by snapping to the middles of the lateral sides of the left view. Do not forget to also set the dash-dot line type in the graphic line parameters (the option) or on the system panel.



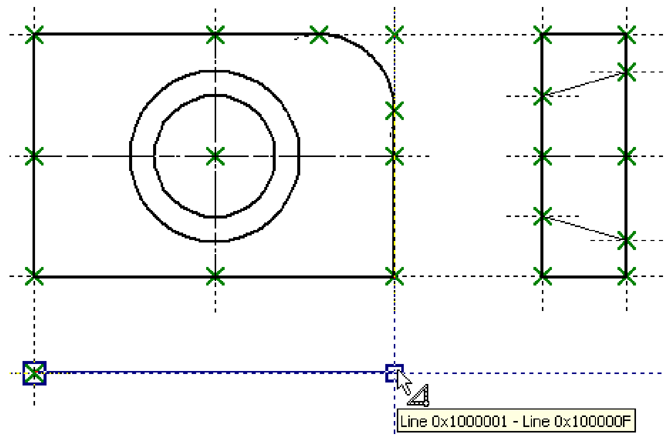
Let's proceed to creating the top (plan) view. We will create it a little bit different then when creating a simple sketch. We will not use the option of constructing a parallel segment. Now there is no practical necessity in defining that particular relation. When using the automatic parameterization mode, the use of that option will make the system try to create a construction line beneath the segment parallel to a line beneath another segment. As a result, relations would be created that we didn't need. Therefore, we will continue using the option .

> Do not forget to reset the normal line type after finishing the creation of the centerline (in the graphic line parameters or on the system panel).

Move the cursor to the drawing area below the main view so as to invoke the desired relation with the line of the main view. Click . The first segment node will be created as lying on the intersection of the main view line and the horizontal line.

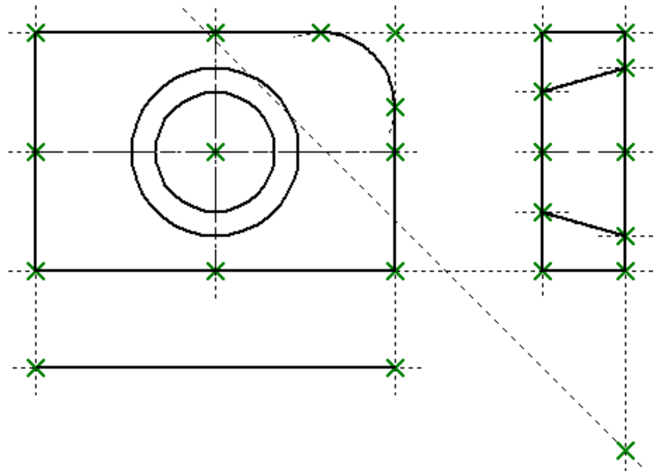


Next, move the cursor rightward up until hitting the snap to another line of the main view. Click again. We have just created the upper segment of the top view.

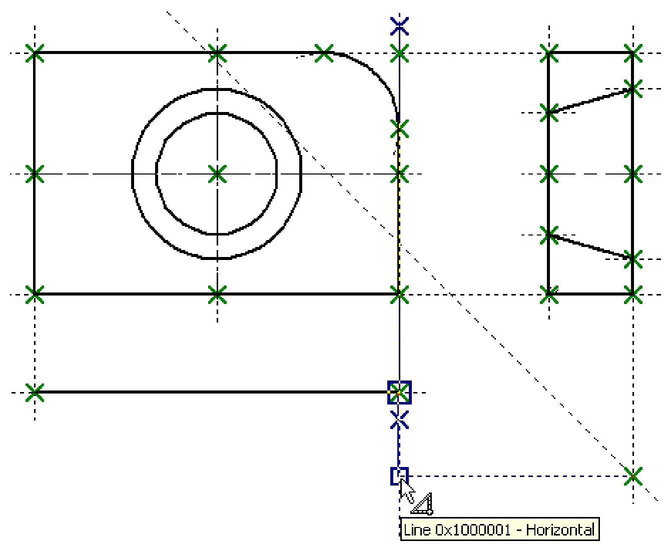


Next, we will have to temporarily quit the sketching command. The reason for that is that it is impossible to create a relation between the left view and the top view by the common means of automatic parameterization. Such relation can be achieved only by various workaround methods (for example, introducing variables as the parameters of the segments being created). But we will simply use the command **L: Construct Line** and create an auxiliary line at the angle of 45° to the outer lines of the left view and top view (just as we did in the conventional creation of the parametric drawing).

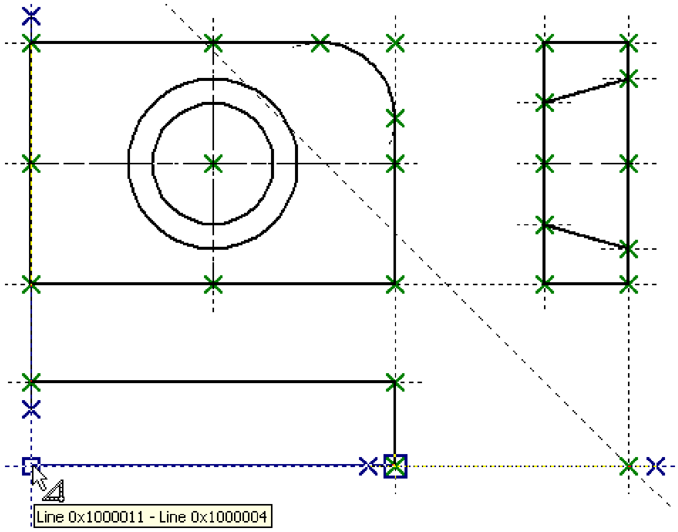
So go ahead and call the command **L: Construct Line**. With the help of the option , construct a line – the symmetry axis between the left-hand-side vertical line of the left view and the horizontal line of the top view. Place the cursor at the intersection point of the created line and the right-hand-side vertical line of the left view, and then press the button <Space>.



After that, call the command **SK: Create Sketch** again. We will create the next segment of the top view. Select the end node of the last created segment as the first node of the next segment. Then move the cursor up until reaching the intersection between the line and the horizontal through the node as shown on the figure. Fix the achieved point by clicking .

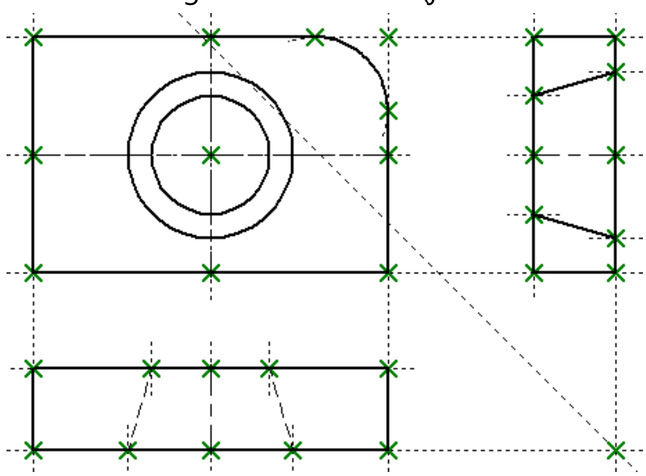


The next segment is constructed by snapping to two lines.

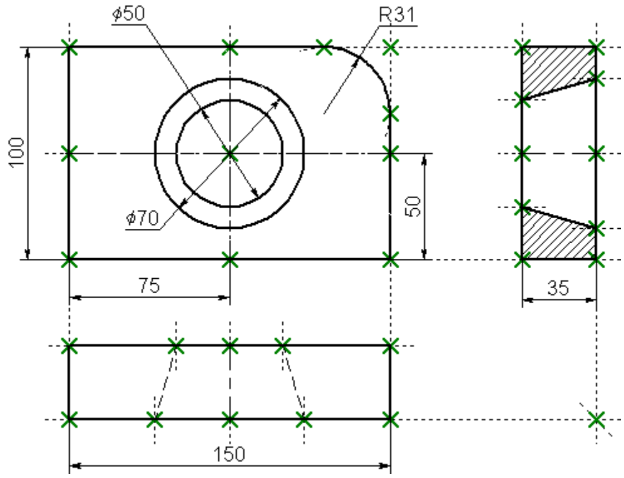


Next we need to close the created graphic lines of the top view by moving the cursor to the first created node of the view and clicking , followed by (to cancel the continuous line input mode).

Create the centerline and the lines defining the conical hole (just like that on the left view).

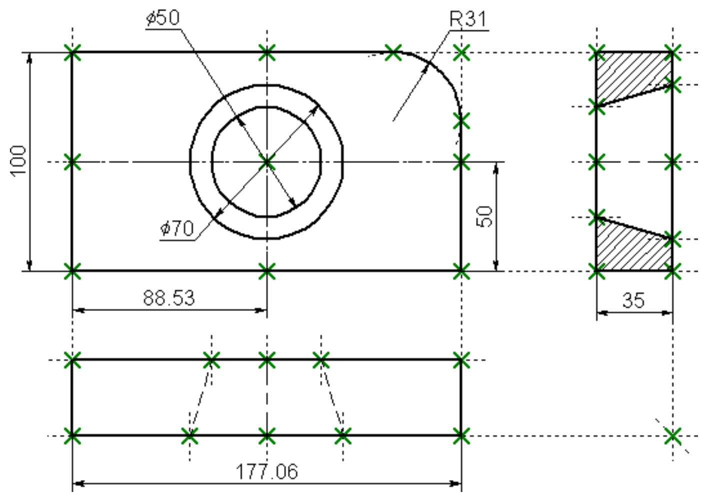


Create the hatch on the left view and the dimensions in the same way as in the previous cases.



This completes the creation of a parametric drawing in the automatic parameterization mode. From now on, such drawing will behave just as a common parametric drawing.

To test, move the cursor to the segment that makes up the left border of the main view, and click . The command will be launched to edit the selected graphic line. If you click on the line once again, the system will automatically go into the command of editing the construction line underlying this graphic line. Move the line around, define the new position by . The width of the plate main view shall automatically change. Besides that, the top view should change as well, since it was constructed by snapping to the lines of the front view.



Similarly, try to edit the position of the right boundary of the main view. In this case, the entire plate drawing will move. Try the same with other drawing elements, including circles. As construction elements are moved, the shape and dimensions of the plate will be changing so as to maintain the relations defined by the construction.

This completes the brief introductory course. Please feel free to refer to the rest of T-FLEX CAD documentation for detailed description of various system functionalities.

MAIN CONCEPTS OF SYSTEM OPERATION

DOCUMENT MANAGEMENT

Creating New Document

For creating new documents the dialog box "Start Page" can be used (see the chapter "Getting Started"). This dialog box is always present on the screen when the standard settings of the system are used. This dialog box enables to create new documents on the basis of the templates and also open already existing documents from the list of recently used ones. In addition to the dialog box "Start Page", the commands grouped in the textual menu File can also be used for creating new documents.

To create a new drawing, use the command FN: Create New Model:

Icon	Ribbon
	Get started→ Files → Drawing
Keyboard	Textual Menu
<FN>, <Ctrl><N>	File > New > Drawing

A command F3: Create New 3D Model allows to create a 3D model:

Icon	Ribbon
	Get started→ Files →3D Model
Keyboard	Textual Menu
<F3>	File > New > 3D Model

To create a new assembly drawing, use the command:

Icon	Ribbon
	Get started→ Files → Assembly Drawing
Keyboard	Textual Menu
	File > New > Assembly Drawing

To create a new 3D assembly, use the command:

Icon	Ribbon
	Get started→ Files →3D Assembly
Keyboard	Textual Menu
	File > New > 3D Assembly

> Product structures of assembly documents include records for an assembly forming. It's their only differ from the standard documents.

A created document name depends on the document prototype. Forstly created drawing will be called "Drawing 1", firstly created detail – "Detail 1". You can assign any other name upon saving the document.

> T-FLEX CAD does not distinguish between the 2D drawing and 3D drawing files. In the document created as a 2D drawing, the 3D model can be generated afterwards. In the document created with the use of the command F3: Create new 3D model the new 2D drawings could be generated.

To create new documents, template files are used that are defined in the command Customize > Settings..., the tab "Files". They contain elements and settings for the new document.

You can change prototype manually by editing of the respective template file or enter another template file name.

> The prototype files should be placed in the folder "...*T-FLEX CAD\PROGRAM\Template*". The name of the directory for the template files is set in the command Customize > Options..., the tab "Folders".

> A user on his own can create an arbitrary number of prototype files.

A new file can be created from an existing prototype using the option "New document" of the dialog box "Start Page". Otherwise, use a similar dialog "New From..." by calling the command FP: Create New Document Based on Prototype.

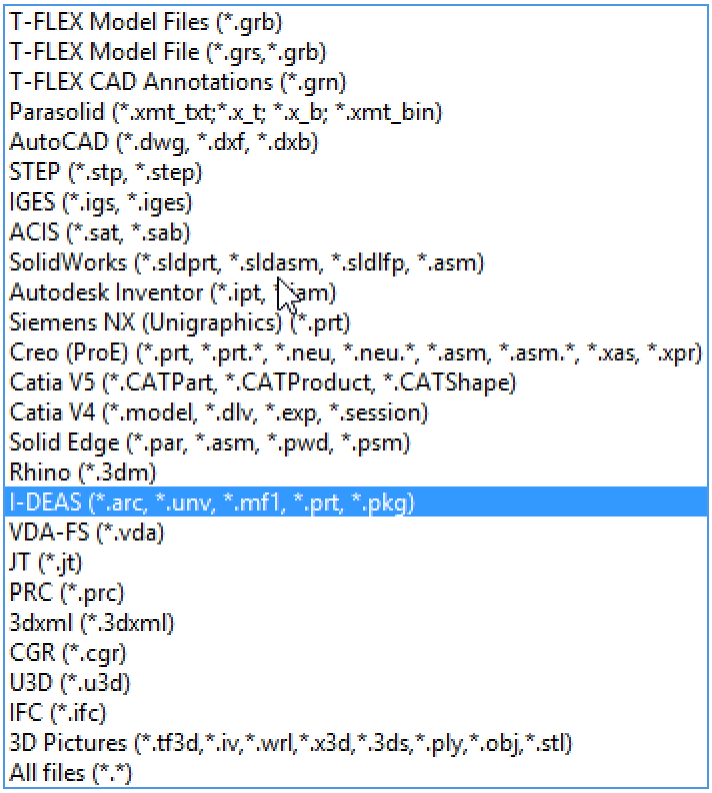
Opening Document

A T-FLEX CAD document can be opened using the command **O: Open Model**. Call the command using:

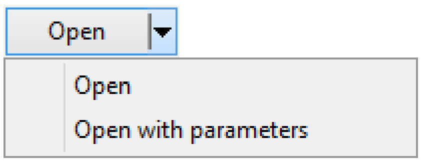
Icon	Ribbon
	Get started→ Files → Open
Keyboard	Textual Menu
<O>, <Ctrl> + <O>	File > Open

The new window **Open** will appear on the screen. This is a standard Windows dialog box for opening application files.

The command allows you to open not only *.grb format files, but also open files in formats of other systems.



When you open the file of another system format, you can select the **Open** or **Open with parameters** option.



If you select **Open with parameters**, a dialog similar to the import dialog for the specified format opens. In this way, you can open the model with the specified parameters.

> More details about Import can be found in the "Exporting And Importing Documents" chapter.

Panning and Zooming in Active Drawing Window

The drawing image can be panned and zoomed in and out in the active drawing window. Zooming effectively changes the size of the working window of the drawing. The easiest way to do these manipulations is using a mouse with a middle wheel. Alternatively, the working window size can be changed by using the rulers as described below. Besides, a command is provided for this purpose, ZW: Zoom Window. Call the command via:

Icon	Ribbon
	View → Scale → Zoom Area
Keyboard	Textual Menu
<ZW>, <F3>	View > Scale > Zoom Area

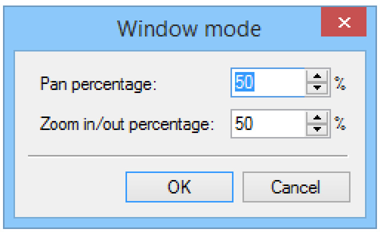
The following options are provided with the command:

	<P>	Set command options

Selecting this option brings up a dialog box on screen with the following parameters:

Pan percentage. Defines the percentage of the working window shifting left/right and up/down.

Zoom in/out percentage. Defines the percentage of the working window magnification.



	<A>	Zoom All

This option redraws the working window according to the drawing format size. The latter is set in the command **ST: Set document parameters**.

	<M>	Zoom Limits

This option calls the command ZM: Zoom Limits that fits the full image to the drawing area.

	<T>	Actual Size

This option calls the command ZT: Actual Size the drawing and 3D model in accordance with their real dimensions.

	<I>	Zoom In

	<O>	Zoom Out

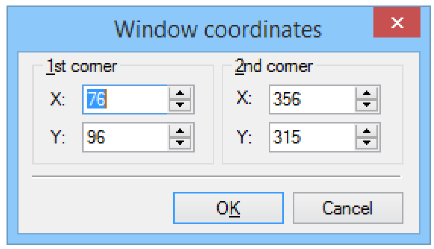
These options respectively magnify and shrink the drawing image each time by a fixed percentage ratio specified in the command parameters.

	<L>	Pan Left
	<R>	Pan Right
	<U>	Pan Up
	<D>	Pan Down

These options move the drawing image by a fixed percentage ratio specified in the command parameters.

	<W>	Set absolute window coordinates

Calling this option brings a dialog box on screen for inputting window coordinates. The user can type in the coordinates of the two opposite corners of the working window.

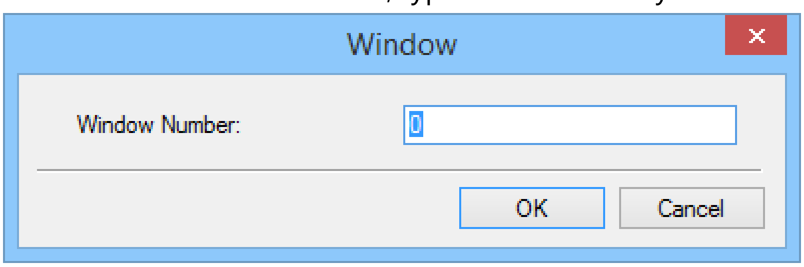


	<<Backspace>	Zoom previous

This option resets the active drawing window coordinates to the previous settings.

	<S>	Save current window coordinates

This option allows saving the working window coordinates and assigning an Id to the saved configuration. A dialog box appears on screen for entering an Id from 0 to 9 to be assigned to the saved window configuration. To return to a saved window, type the number key of the desired Id (<1>, ...).



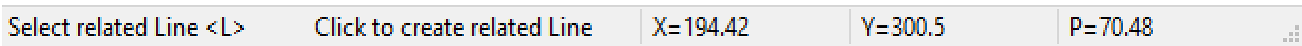
		Set the working window size by dragging selection box

An arbitrary area of the drawing can be zoomed on by specifying two opposite corners of a box. Move the cursor to one corner of the area to be zoomed on, and press and hold . A rectangle starts rubberbanding after the cursor. Drag it, selecting the desired area by box, and then release the button. The selected area will be displayed zoomed on. A one-time call to any option of the ZW: Zoom Window command can be done asynchronously from within any element creation or editing command by typing <F3> function key or selecting the icon.

The options of the ZW: Zoom Window command are also accessible via the View > Scale menu and the View toolbar.

Status Bar

The status bar is located at the bottom of the application window.



The status bar has the following fields (left to right):

The current command name field. This field displays the full name of the current command.

The field can also be used as the input box for calling a command by typing on keyboard. Type the reserved keyboard accelerator sequence or press a function key combination. This can be done only when the field is empty, containing only the prompt symbol ">". Typing a sequence that is not part of any command name automatically clears the field. If so, type again. A correctly typed accelerator sequence causes the full command name and a brief description to be displayed. For instance, type the following sequence, <R><O>. Once typed <O>, you entered the command for creating roughness symbols, and the field will display *RO: Create Roughness Symbol*. In the command descriptions down the manual, the respective keyboard accelerator sequences will be printed in a single, common pair of triangular brackets, as in <RO>. We will thus distinguish those from simultaneous double- or triple-key combinations for calling commands, as, for example, <Ctrl><O>.

Help field. This is an information field displaying help messages and prompts for user. If the cursor is within the active drawing window, this field displays suggested user actions. When the cursor is pointing at other fields on screen, information is displayed about their purpose. While within a command, pointing at an icon of the automenu brings a help message in the field, describing the action performed by the option.

X-coordinate field.

Y-coordinate field.

Auxiliary coordinate field.

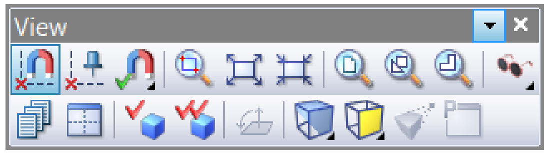
Toolbars

A toolbar is a set of icon buttons for calling the application commands. There can be several toolbars on screen simultaneously.

In the standard package of the T-FLEX CAD there are five toolbars: “Main toolbar”, “System toolbar”, “View”, “Full screen mode”, “Context” (toolbar which appears when entering fragment editing mode in the assembly context). In addition to that, any number of user-defined toolbars can be created (via the command SB: Show Toolbars). With the help of the same command the structure of user-defined and several standard toolbars can be modified as well (for example, of the toolbar “View”).

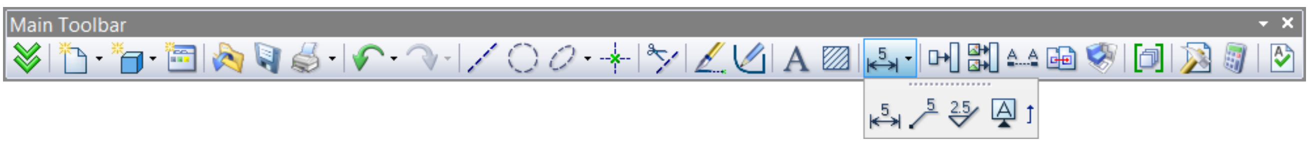
What toolbars to display is defined in the command “Customize|Customize…” on the “Toolbars” tab. Besides, a desired toolbar can be accessed via the context menu by pressing right mouse button while over any toolbar.

Any toolbar can be docked along any of the application window borders, or floating within. When floating, a toolbar window is titled, and can be resized.

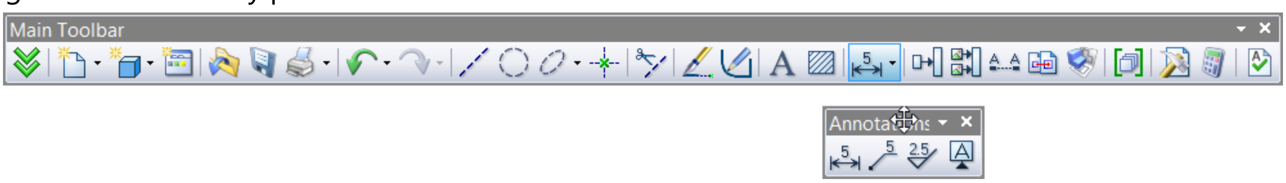


Embedded toolbars

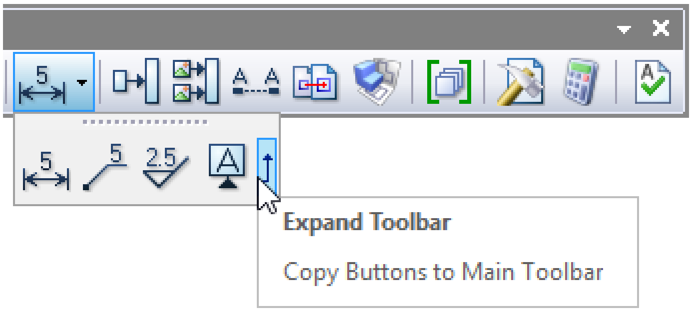
Several command pictograms in the instrument toolbars can be grouped on the principle of similarity of performed functions. In this case the instrument toolbar will show only one pictogram of the given group (the rest are not shown), and the button will be placed on the right side from the pictogram. By pressing this button the “embedded” toolbar with the rest of the pictograms of the given group will emerge.

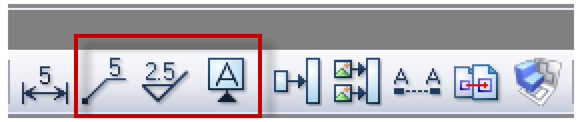


The embedded toolbar can be converted into the ordinary instrument toolbar. To do that, it is necessary to place the cursor on the header of the embedded toolbar, press and, without releasing the mouse, drag the toolbar to any place of the T-FLEX CAD window.



The buttons of the embedded toolbar can be placed directly on the main toolbar. To do that, it is sufficient to press the button on the right end of the embedded toolbar.

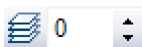
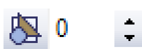




System Toolbar

The system toolbar is a set of tools for quick definition of element parameters at creation and editing time.

Following is the description of the functions of the system toolbar fields and buttons.

- ✓ Layer configuration command button .
- ✓ Layer name box . Displays the layer name of the model elements being created and edited.
- ✓ Button for setting visibility levels of the model elements .
- ✓ Level input box . This box displays the current level of the model elements. Changing levels can be done within element creation and editing commands. Clicking inside the box sets a text cursor. Type in the element level. Confirm the input by pressing <Enter> or clicking within the drawing window.
- ✓ Priority input box . This box displays the current priority of the model elements. Changing priority can be done within two-dimensional element creation and editing commands.
- ✓ Color selection box . This box displays the color of the element being created or edited.
- ✓ These are the main items that are always present on the system toolbar. The rest of elements replace depending on the application state.

When in command waiting mode, the system toolbar contains selector controls, as follows:

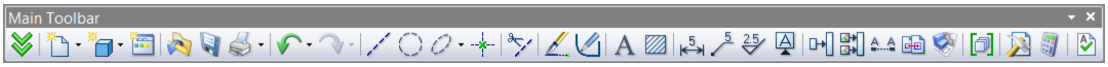
- ✓ Button for calling selector configuration dialog . Used for specifying exact settings of the selector and defining named selector configurations.
- ✓ Button for calling named selector configuration . Brings a pull-down list of available configurations.

Quick selector setup buttons:

- ✓ The buttons and help to quickly allow/disallow selection of all types of elements;
- ✓ The buttons , , , , , , , и define and edit the current set of elements allowed for selection. The pushed icons represent the elements allowed for selection. The set of buttons will be different when working in the 3D window.

> Other various items may be added to the system toolbar while working with various 2D commands. For instance, a line type box appears on the system toolbar while creating lines, along with the line start and end arrow type boxes. When creating text, the font name and size boxes are displayed.

Main Toolbar

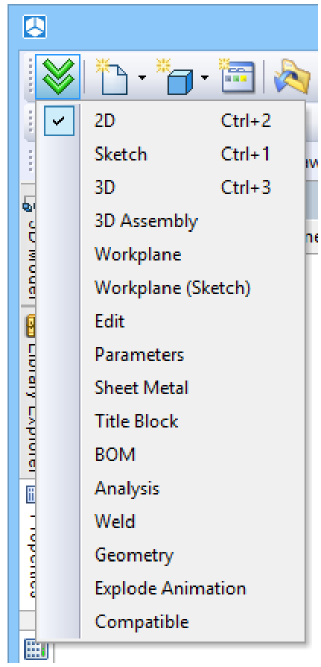


The main toolbar has a set of buttons which, depending on the currently solved problem and the settings of the system, can be selected by the user or automatically activated.

> The button sets in the main toolbar are aimed at solving different problems – geometric construction, 3D modeling, analysis, geometric construction on the workplane, operations with sheet metal, editing specifications etc. Internal specialized modules, which are included into the T-FLEX CAD package, can add their own button sets into the main toolbar. For example, the application "T-FLEX CAM" adds to the main toolbar a set of buttons which perform the functions of this particular application.
>
> Switching between the button sets in the main toolbar occurs automatically depending on the operations performed in the working window of the T-FLEX CAD.
>
> For example, upon opening of the 2D document the set "2D" is turned on automatically, and upon transition to the 3D window – the set "3D". When a drawing is made on the workplane, the set "Workplane" or "Workplane (Sketch)" becomes active (depending on what has been used last time in the given situation).
>
> At the beginning of the BOM editing, the button set "BOM" is activated. Upon the exit from the BOM editing, the set, which was active before the editing was started, turns back on in the main toolbar.

Some of the standard sets of the main toolbar are invisible by default and shown only upon activating the corresponding command of the T-FLEX CAD. For example, the set "Text" is by default not present in the list of the main toolbar modes, but upon entering the mode of creating/editing the text, this particular set will appear on the main toolbar.

When the set "Compatible" gets active, the main toolbar itself represents a copy of the standard toolbar existing in the earlier versions of the T-FLEX CAD.



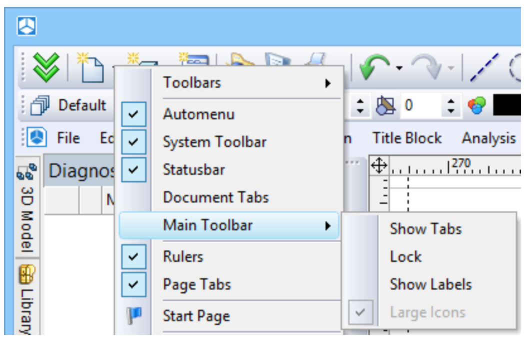
Switching between the button sets can be done manually by using the button on the left side of the toolbar . Upon pressing this button the list of available sets pops up.

The desired set can be chosen with the help of . In addition, several sets can be activated from the keyboard with the help of the specified for them key combinations.

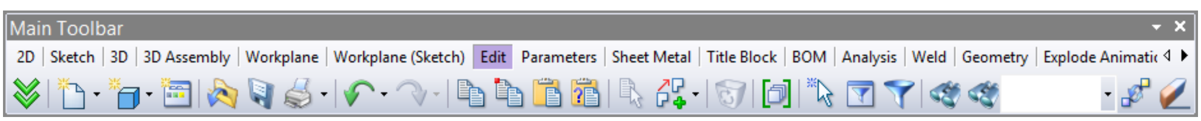
By default, the key combinations are assigned only for the sets “2D”, “Sketch”, “3D”. In the dialog of the command SB: Show Toolbars it is possible to assign key combinations for other sets of the main toolbar as well.

The user selected set is stored in the window of the current document and automatically recovered when the window becomes active. The given setting is stored in the document and gets activated when the file is open.

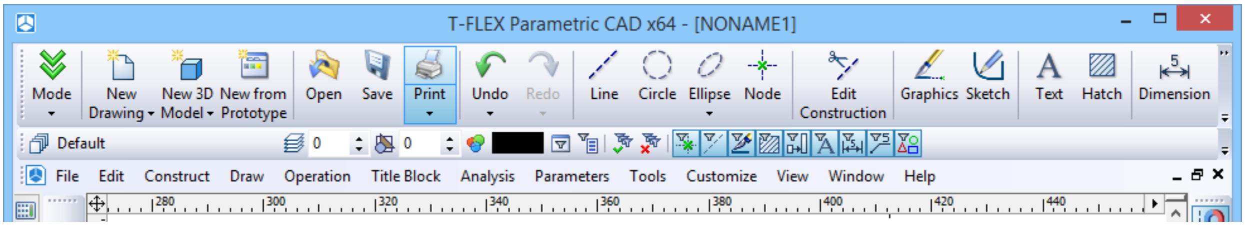
It is possible to decline automatic switching between the main toolbar sets by setting on the flag Lock. This flag can be found in the context menu, called with the help of in the auto-menu field or any other instrument toolbar. After turning on the flag, the main toolbar state is going to be modified only upon manual switching between its sets.



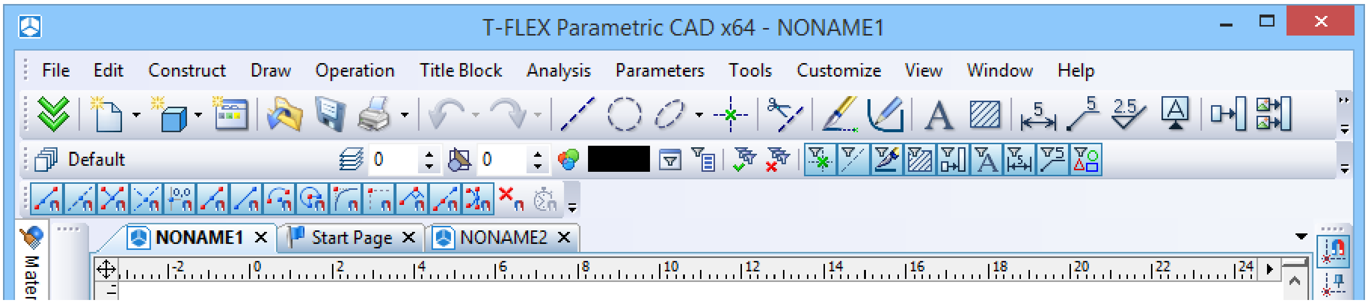
One more flag is available in the same context menu which allows controlling the view of main toolbar – Show Tabs. It controls visibility of the tabs on the main toolbar. The tabs allow quick switching between the button sets of the toolbar. The tab of the active set is marked with the color.



The flag Show Labels enables to add annotations to the buttons of the main toolbar. This can be convenient at the first acquaintance with the system or while working with the high resolution monitor.



The flag "**Large icons**" allows turning on the mode of the large icons for the main toolbar (no matter what the size of the icons in other system toolbars is).



The command **SB: Show Toolbars** provides with additional possibilities for controlling the main toolbar. Via the tab "Main Toolbar" of the dialog of this command, it is possible to do the following:

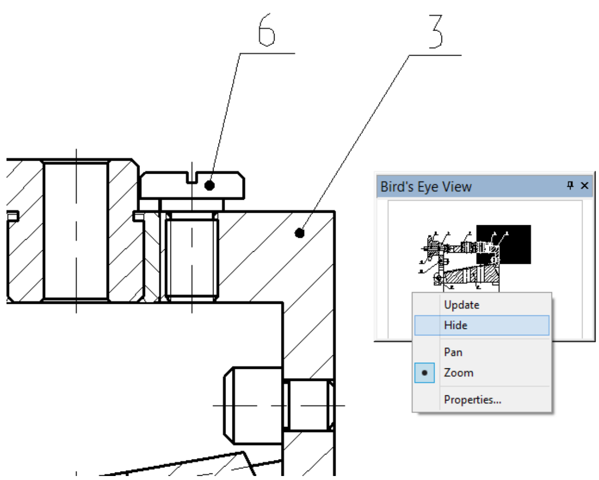
- ✓ Hide/show main toolbar sets in the list of sets (displayed upon pressing the button );
- ✓ Rename the main toolbar sets;
- ✓ Create and remove user's defined sets;
- ✓ Create a separate toolbar on the basis of any set of the main toolbar.

Bird's Eye View Window

The "Bird's eye view" window helps quick navigation around the drawing. It always displays the whole drawing image, regardless of the working window size currently set for the active drawing window.

Visibility of the "Bird's eye view" window can be controlled by the textual menu item "Customize|Tool Windows|Bird's Eye View" or via the context menu coming up on right mouse button click over any of the toolbars.

The "Bird's eye view" window can be docked along any of the application window borders or stay floating.

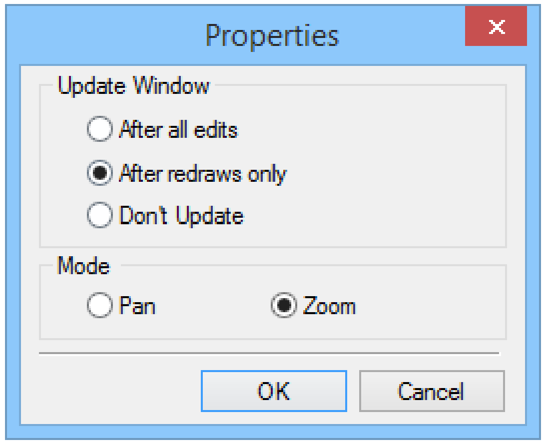


The modes of the "Bird's eye view" window can be controlled via the context menu coming up on right mouse button click within the window.

Pan. In this mode, a box follows the cursor within the "Bird's eye view" window indicating the area of the drawing to be displayed. The size of the box can be changed by switching to zoom mode. To select the area to be viewed on the drawing, press . Dragging the box across the "Bird's eye view" window panes the actual drawing dynamically according to the box movement.

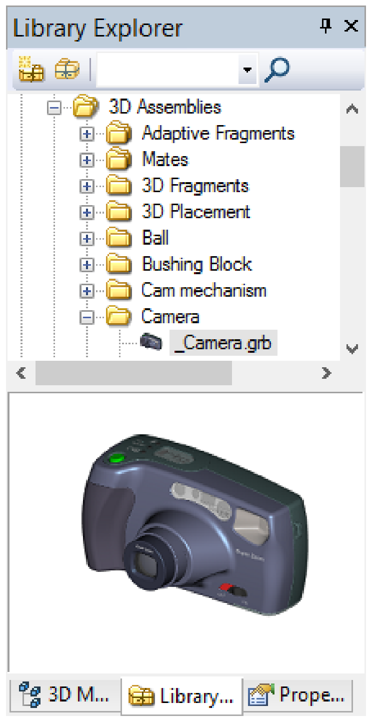
Zoom. In this mode, no box is displayed on entering the "Bird's eye view" window. Press in the "Bird's eye view" window to define one corner of the viewing box and drag the cursor, rubberbanding the box. Doing so, define the area of the drawing to be zoomed on, and then release .

Once defined, the viewing area will be highlighted in the "Bird's eye view" window, and the respective portion of the drawing will be zoomed on in the active drawing window.



Properties. Selecting this item brings up a dialog box for defining the window update parameters and the pan vs. zoom mode selection.

Using Library Explorer



The library explorer window can be used for opening documents for editing, along with the O: Open Model command. It comes up on starting the application and docks by the left border of the application window. It also can stay floating.

The user can control visibility of the Library Explorer window via the textual menu item Customize > Tool Windows > Library Explorer or in the context menu on right mouse button click over any toolbar.

The library explorer shows the content of the open libraries. It allows to select libraries, open documents for editing and insert documents into the current one as fragments or pictures.

The library explorer window may have a preview pane at its bottom or right-hand side. This pane will display the preview image or the properties of the selected document.

The library explorer window can have various different settings accessible by .

More details on working with the model window and library configurations follow in later chapters.

External Links

A T-FLEX CAD document can have links to other files: T-FLEX CAD documents (fragments), graphics files (pictures), database files, etc.

To ease working with composite documents, T-FLEX CAD provides a mechanism of links management. A link is a T-FLEX CAD object containing the path to an external file (the target of the link).

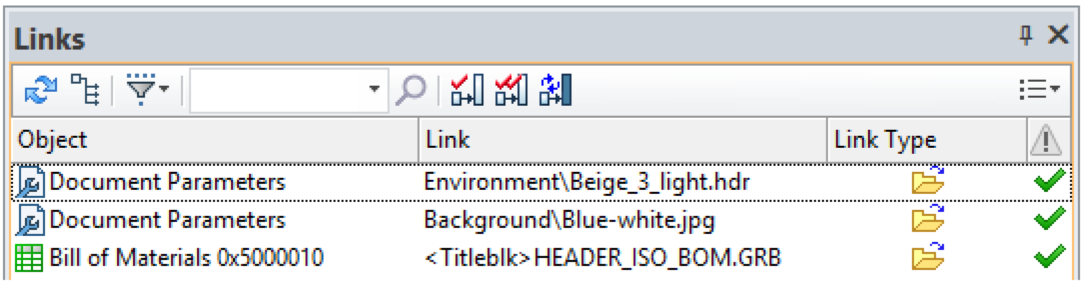
Links are used when creating fragments, pictures and other T-FLEX CAD elements for specifying the source of external data. The same link can be used by different elements; for example, several fragments based on one file will refer to one link.

The links management mechanism allows managing the way of storing the link targets. T-FLEX CAD allows storing the link targets either outside a T-FLEX CAD document as a conventional external file ("external link"), or directly within the file of the composite document ("internal link"). The internal storage of the link increases the size of the composite T-FLEX CAD document, but allows dealing with it as with one file.

The links management mechanism solves the problem of moving large assembly documents. By using it, you do not have to search for all fragments files that may be located in different folders, on different disks, in libraries, etc. All you need to do is to "zip" the assembly model into one file with a provision of future unzipping, and move it to another place in the file system or into a storage of a document management system.

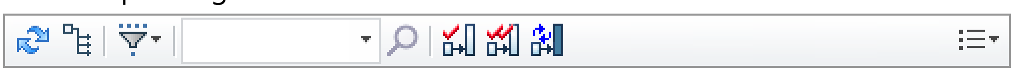
Links window serves for managing links in a composite document. To call the window use **Customize** > **Tool Windows** > **External Links** command.

Links window appears upon calling the command. It includes list of all links, existing in the document.

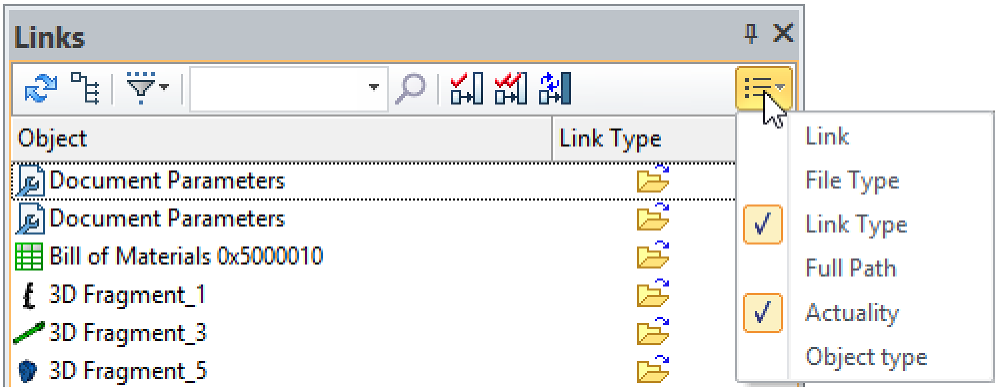


Toolbar

Some commands for operating with the window are located on a toolbar.



Columns are used to receive information about files included into the document. The columns may be selected from the drop-down list.



Columns:

Link. Displays link to the file according to the location of the composite document. If the composite document and the file are in the same folder, only the filename will be displayed in the column.

File Type. Displays the file format.

Link Type. Displays the link type:

> **Embedded** is a link to the document saved within the file of the current document.
>
> **External** is a link to the document stored in the standard filesystem.
>
> **Substitute** is a link to an external document whose location was automatically defined by the system at the time of running the **AM: Move Assembly** command with the **Substitute** setting.

More information about the command can be found in “Links. Managing Composite Documents” chapter.

Full Path. Displays a full path to the file.

Actuality. Displays file link actuality.

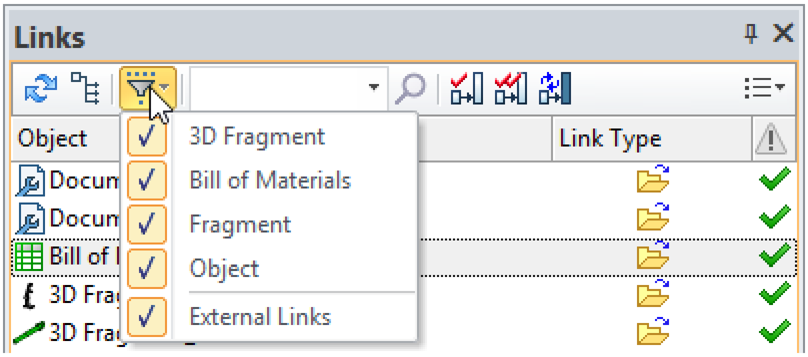
The icon indicates that the link is actual.

The icon indicates that the file on a given link is not found.

The icon indicates that the changes from the file have not been put in the assembly and the document needs to be updated.

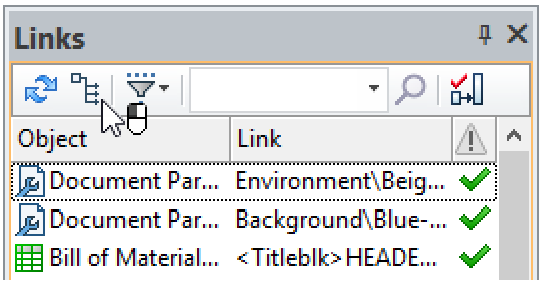
Object Type. Shows a type of the object.

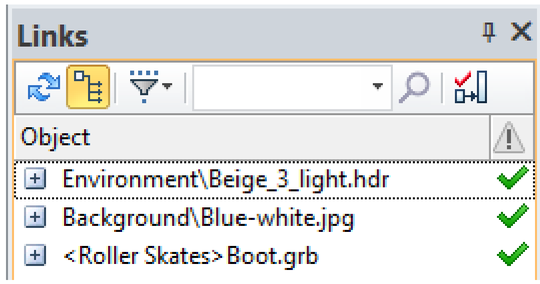
You can customize display of object types, external and internal links in the window using filter.



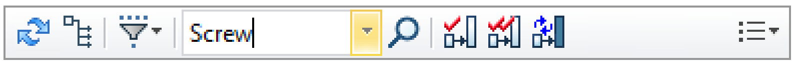
Links to new files are displayed in the window after activating option.

Files associated with one link unite when the button is active.





You can search for files using a search bar.



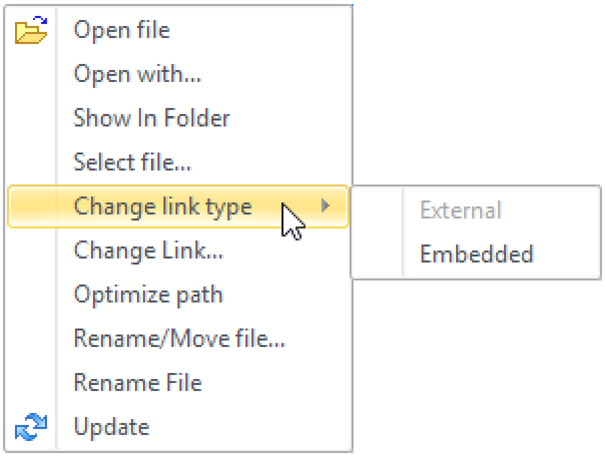
Update Links option allows to reload data from all external files contributing to the composite document.

Update Assembly option starts the converter that converts and saves all the fragment files that are included in the assembly

Refresh files is provided for updating the fragment document per the changes in the assembly when working by “Top – down” approach or in the assembly context.

Context menu

Commands for working with elements in the **Links** window are available in the context menu.



Open file. Allows to open one or several selected files in T-FLEX CAD.

Open with. Allows to open the selected file with the help of an external application. The application can be selected from a coming up window.

Show in Folder. Opens Windows folders with the selected files.

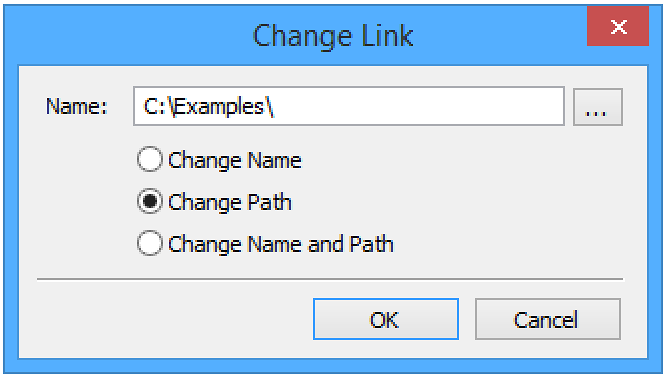
Select file. Option allows to select the file which will replace one or several selected files.

For example, if the file name was changed and the file dissappeared from the assembly, you can select the renamed file using the **Select file** command.

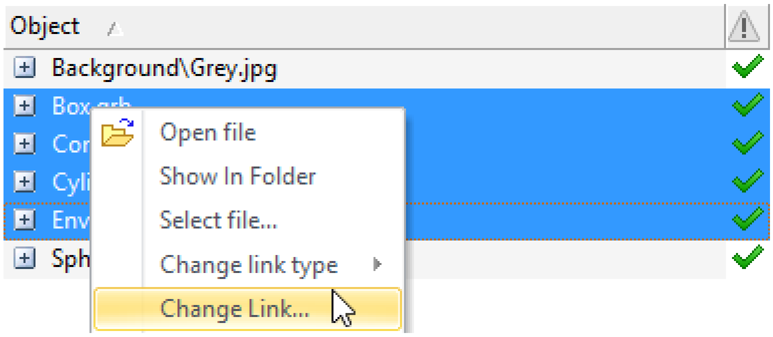
Change link type allows to change one or several links types. For example, setting the type "Embedded" for an "External" link will put the external file, which is the link target, into the current T-FLEX CAD document, all data included, so that the file ceases being external.

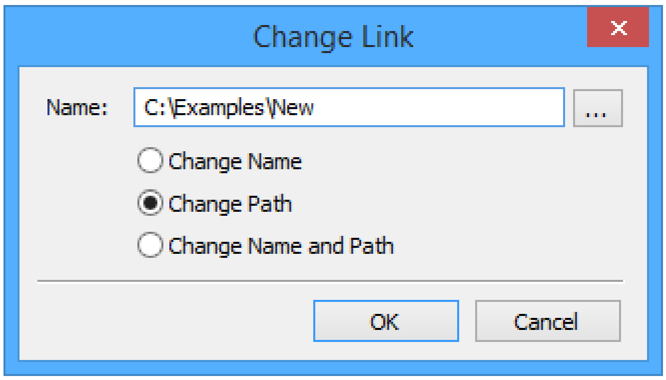
> An embedded link can be used for temporary storage of fragments data within an assembly document. For example, this is necessary when you need to move an assembly document to another computer. "External" link can be re-assigned afterwards to return back the external storage of fragments in the separate files.

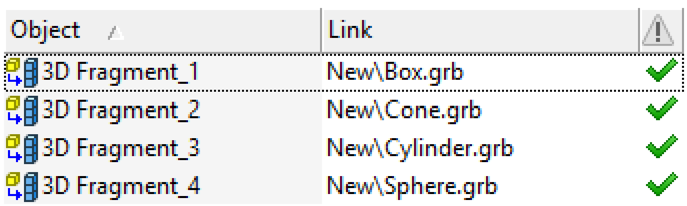
Change Link command calls **Change Link** dialog window. You can change **name**, **path** or **name and path** for the link here.



The command can be used for multiple links. For example, if you change paths to several fragments, they will refer to files with the same name located in another directory.







> If there are no files with desired name in the specified directory, a warning will be displayed in the diagnostics window.

Optimize path. All absolute paths will be replaced by relative path.

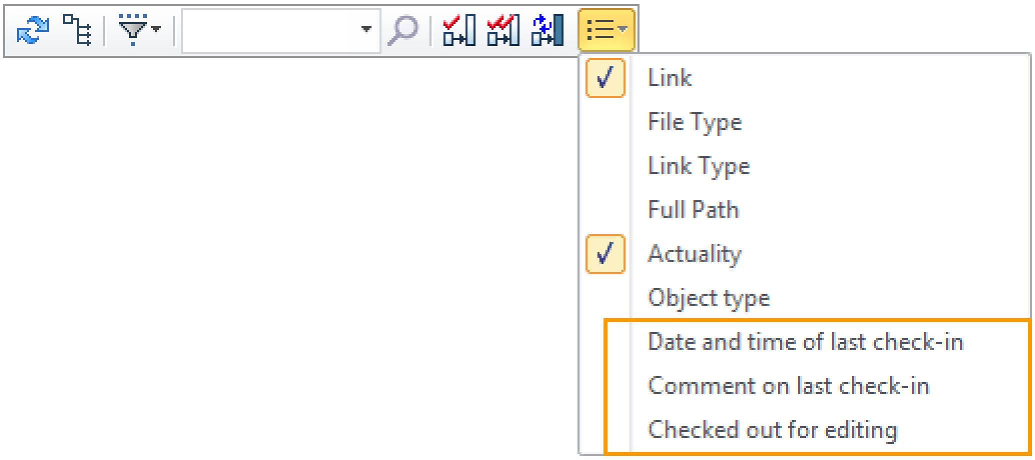
Rename/Move file. Allows to specify a new directory for fragment file storage and to rename the file. The file will be still associated with the current assembly.

Rename file. Allows to rename the fragment file associated with the link.

Update dublicates the  option from the toolbar.

A drop-down list with commands for working with the selected file is located in the bottom part of the context menu.

Additional columns appear in the window if integration with T-FLEX DOCs is enabled. These columns are used for team development.



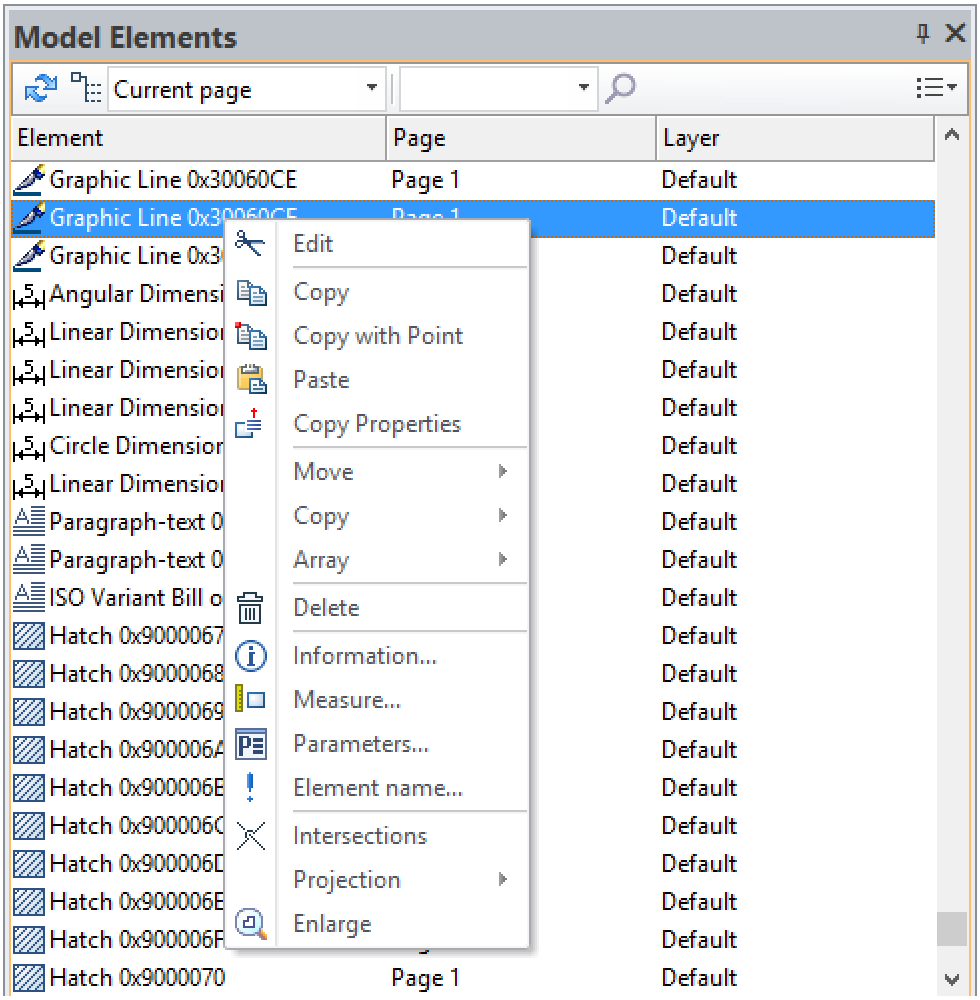
Sorting and grouping are available for records of the **Links** window.

> More information about sorting and grouping can be found in “Variables” chapter.

Model Elements

A **Model Elements** window is used to display all elements in the document.

Both 2D and 3D elements may be displayed in this window. Context menu with the corresponding set of commands is available for each element.



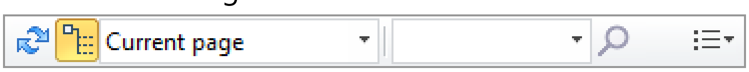
When you select elements from the list, they are highlighted on the drawing and in the 3D scene.

In the window, you can select elements to be used in the current operation, for example, select graphic lines for the **Copy** operation.

This window is particularly useful when working with drawings. It provides information on each element of the drawing on the current page or on all pages of the document.

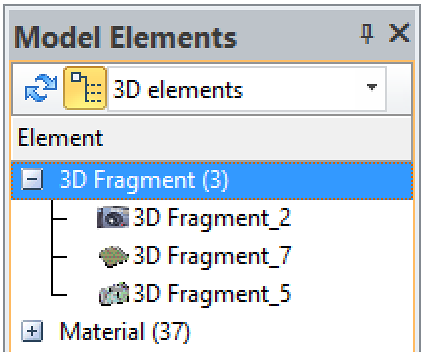
Toolbar

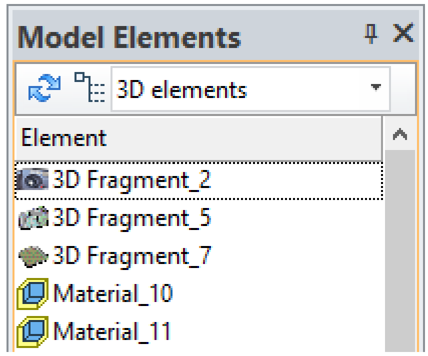
The toolbar is used for the window management.



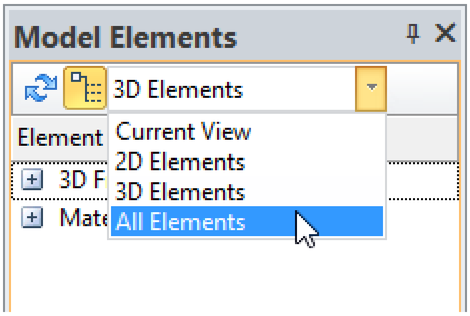
To update the content of the window option  is used.

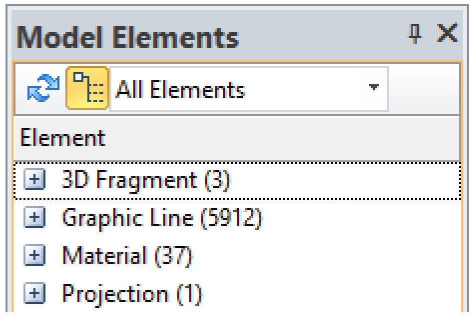
Option  helps to enable or disable the grouping of elements by type.





The drop-down list is used to configure the display of elements in the window.





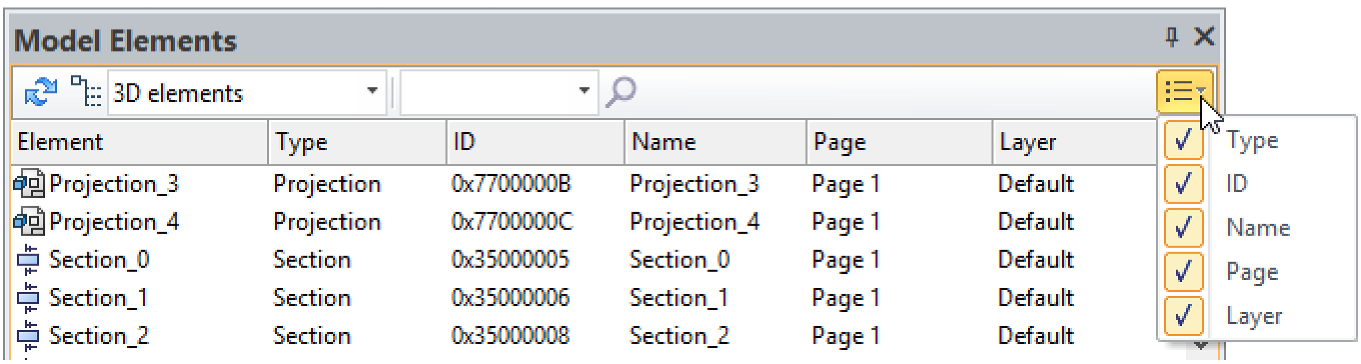
Current view – shows elements in the current active view.

2D Elements – show all 2D elements, existing in the document.

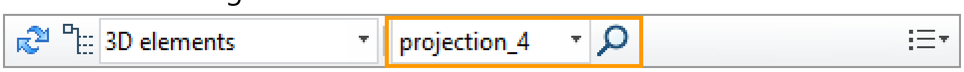
3D Elements - show al3D elements, existing in the document.

All Elements - show all elements, existing in the document.

In the right corner of the window, you can select a column, which will display information about elements: **type**, **ID**, **name**, **page** and **layer**.



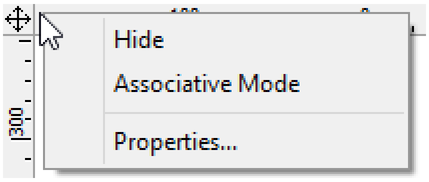
You can search for elements using the search bar.



Sorting and grouping are available for records of the **Model Elements** window.

More information about sorting and grouping can be found in "Variables" chapter.

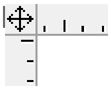
Rulers



The rulers display X and Y coordinates of the current drawing window. Ruler properties can be set with the help of the context menu.

> Ruler visibility can be set via the textual menu Customize > Tool Windows > Rulers or the context menu coming up on clicking over any toolbar.

Rulers can be used for navigating around the drawing. In the mode when there is the button in the corner between the vertical and horizontal rulers, the horizontal ruler can be dragged by pressing and holding and moving the cursor right and left. The drawing image moves together with the ruler and cursor.
Release to fix the image in the current location. Similarly, the drawing can be moved up and down by dragging the vertical ruler.



Pressing the button at the horizontal and vertical rulers crossing with switches to another button mark. In this mode, the rulers can be used for zooming the drawing in and out.

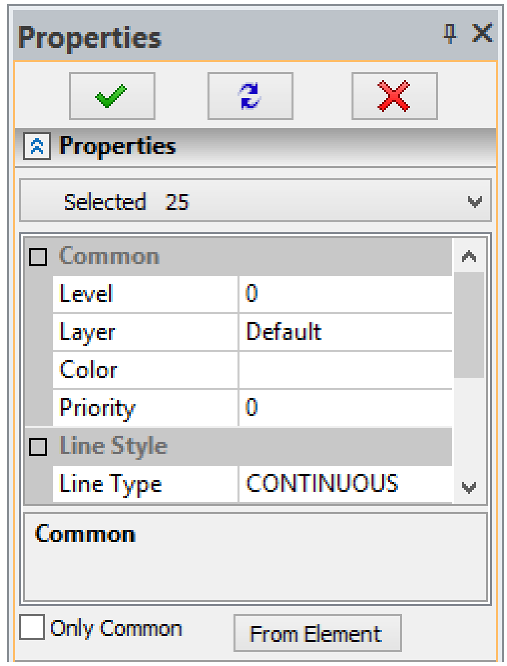
To zoom the drawing in, point the cursor at the horizontal or vertical ruler, press and hold , and drag right or up respectively. To zoom out, drag left or down, respectively. Releasing fixes the drawing image in the current zoom.

To switch the mode back to panning, press the button at the rulers intersection once again.

> Rulers can be displayed in 3D window. In this case, they allow you to better navigate in 3D space, estimating distances and sizes. Ruler shows 3D coordinates of the window on an imaginary plane parallel to the screen.
>
> With the help of rulers in 3D window, you can move and copy the workplane. More information can be found in the "Workplanes" chapter of the "3D Modeling" book.

Property Window

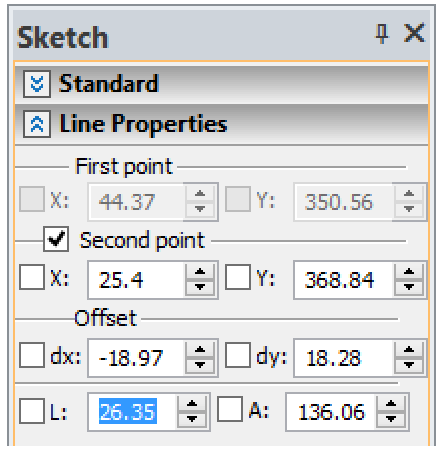
The property window is used for setting and modifying various parameters. One way to use it is in the command-waiting mode for quick editing of the selected element properties. The other way is using it within various 2D and 3D commands for setting various parameters of the elements being created or edited.



This window can be floating or docked along one of the application window borders. Its visibility is controlled by the icon (it can be found on the toolbar "Main" in the mode "Compatible"), as well as via the textual menu item Customize > Tool Windows > Properties Window or via the context menu coming up on click over any toolbar.

This window comes up automatically on entering the commands that use it. Upon leaving such a command, the property window will automatically disappear, unless was docked by the application window or open prior to entering the command.

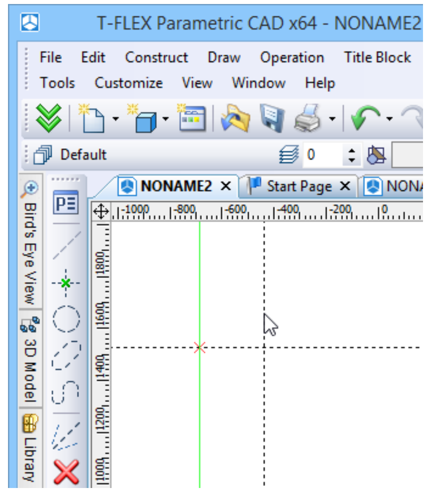
The title and the content of the window depend on the current command and option. The parameters displayed in the window can be input directly by typing on keyboard. The current input box in the dialog can be set by pointing cursor and clicking , or via the keyboard. The key sequence for jumping to a particular input box is shown in a pop-up help coming up while resting cursor over the entry.



The property window may be expandable. In many commands, portions of the property window dialog box may be collapsed by default. The special buttons - and are provided for expanding and collapsing such portions. Once a portion of the dialog was "expanded" while in some command, this setting will be remembered specifically for the given command.

A special provision is made for the property windows in the commands that allow variables and expressions as parameters. The current value of such a parameter is calculated and displayed at the right of the parameter input box.

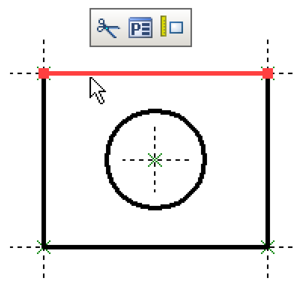
Automenu



Automenu is a special toolbar that contains icon buttons of the available command options. Automenu is context-sensitive. This means, its content changes depending on the current command and a state of the command.

Two outcomes are possible when selecting an action-starting icon in the automenu. First – the result comes right after selecting the icon. For instance, selecting the parameters setting icon , instantly brings up the parameter dialog box on screen. Second – the cursor changes the shape according to the selected option. To obtain the result, the user needs to move cursor to an appropriate location and press . For instance, selecting the construction line - - adds a line mark to the cursor. The cursor then should be moved to a line to be selected, and the button pressed. Only then, the construction line will be selected.

Dynamic Toolbar

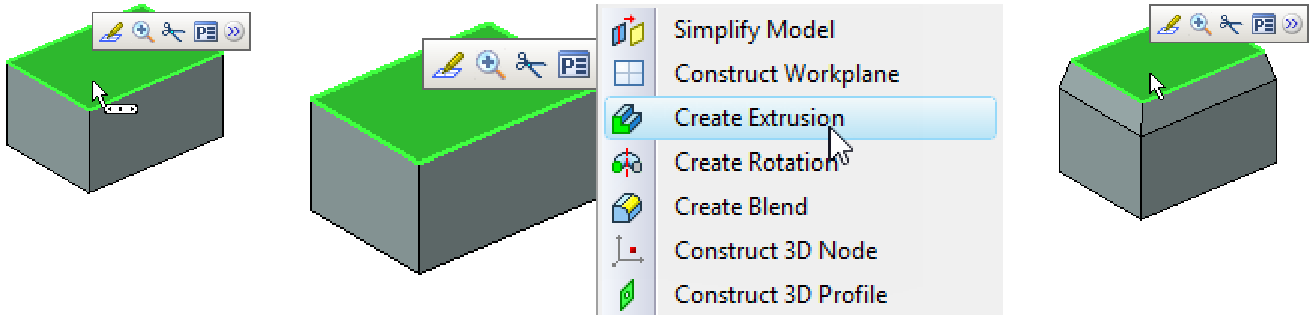


In the command anticipation mode, upon selecting elements with the help of , a special dynamic toolbar will appear on the screen. It contains the icons of frequently used commands for the elements of the given type. The dynamic toolbar disappears automatically after some time has passed or after moving the cursor to a certain distance from the toolbar.

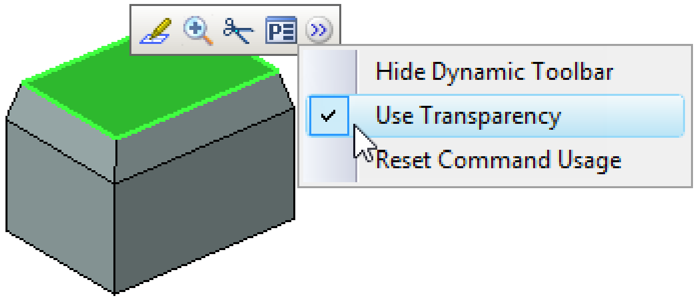
The presence of the dynamic toolbar, while choosing 2D and 3D elements, depends on the settings of the command **SO: Set System Options**. For 3D elements the dynamic toolbar will be shown if the flag **Use Dynamic Toolbar** is turned on in the given command dialog box on the tab **Dynamic Toolbar.**

In addition to that, while working with 2D elements, the parameter "Transparent Element Editing" has to be turned off. By default, the dynamic toolbar appears on the screen only for 3D elements.

In addition to the icons of frequently used commands, the button for calling the list of additional commands will be shown in the dynamic toolbar. Upon calling a command from the additional list, the selected command is automatically transferred to the main set of buttons of the dynamic toolbar (for the elements of the given type). Modifications in the dynamic toolbar are retained in the current Environment of the system.



To cancel changes in the set of buttons of the dynamic toolbar, it is possible to use command “Reset Command Usage” in the context menu of the dynamic toolbar.

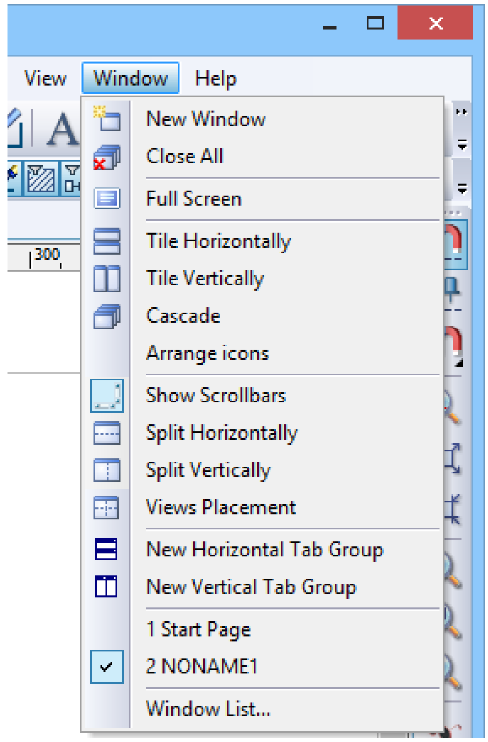


In the context menu of the dynamic toolbar, the flag Use Transparency is also available. When this flag is set on (for default settings), the toolbar looks semitransparent when it appears on the screen. The semitransparency diminishes as the cursor moves closer to the toolbar. When the flag Use Transparency is turned off, the toolbar is always displayed as nontransparent.

Active Drawing Window

The T-FLEX CAD allows the user to work with several documents simultaneously. There is a separate window for each open document. This allows working simultaneously with several drawings or 3D models, switching from one document window to another as required.

The commands designed for working with document windows are grouped in the submenu "Window" found in the textual menu.



Document tabs

For controlling windows the document tabs can be used. Visibility of the tabs is controlled by the flag Customize > Tool Windows > Document Tabs.

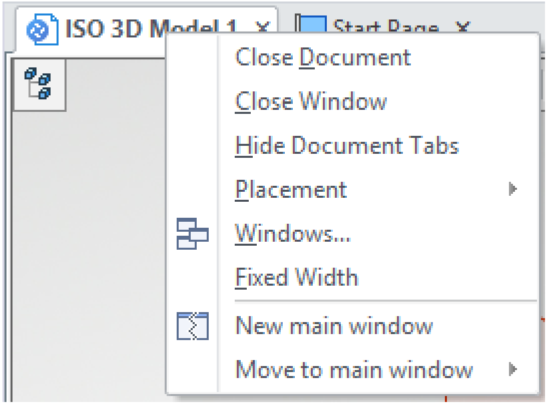
By using these tabs it is possible to switch from one open document window of the T-FLEX CAD into another – for this purpose it is sufficient to point the cursor at the desired tab and click .

Also, with the help of tabs it is possible to change the arrangement of document windows. To do that, the cursor of the mouse should point at the tab of the document, which needs to be moved. Then should be clicked, and without releasing the pressed mouse, the tab of the document should be moved to the required location in the row of tabs.

With tabs, you can also close the document window. On the right side of each tab there is button . Clicking this button will close the current window. Moreover, you can close tab by clicking mouse wheel on it.

> By default the document tabs (if they are visible in the T-FLEX CAD window) are placed above the upper border of the document window. Their location can be changed if desired. For that, it is

necessary to point at the tab of any document with the cursor and with the help of call the context menu. Here you need to select the Placement item.



All unsaved documents are marked with “*” symbol on their tabs.

Close document option allows to close all tabs that belong to similar document.

Close Window option allows to close current document window.

Fixed width option allows to set similar width to all opened tabs.

New main window option allows to create a new main window and copy the selected tab in it.

Move to main window option allows to move tabs between main windows.

More information about main windows can be found in “Quick start” chapter.

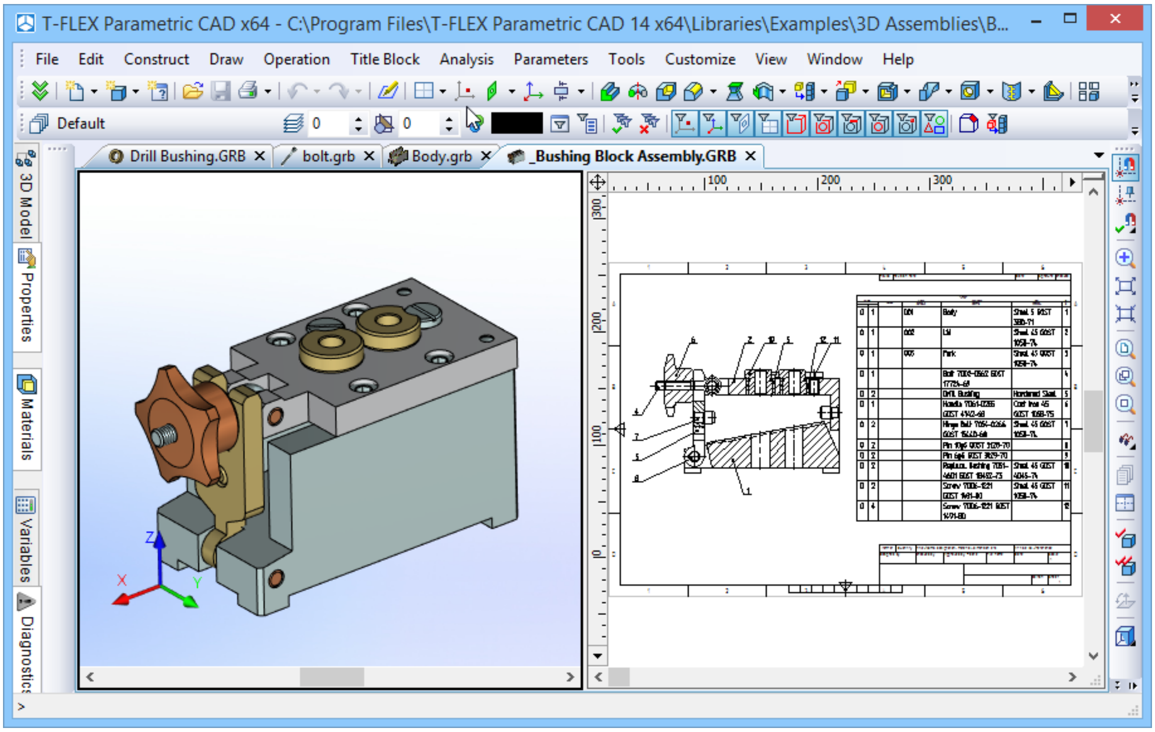
Windows... command opens dialog box for managing opened tabs.

More information about the Windows... command can be found in “Selection of active window” section.

Document Window View with Turned on/off Document Tabs

The view of the open documents with turned on tabs is different from that with turned off tabs.

When window tabs are on, the windows of open documents occupy the whole space of the T-FLEX CAD working window. The active window, (i.e. the window of the current document) is on the top of other documents.

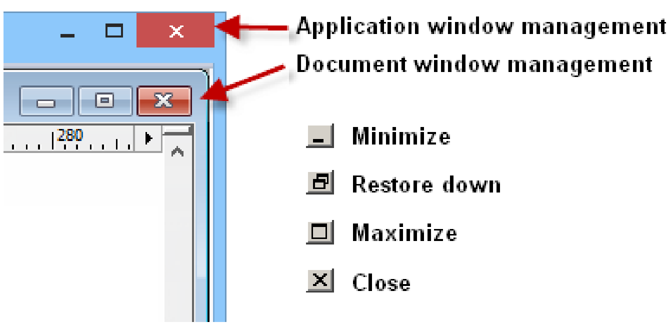


When the tabs are turned on, it is not possible to minimize the window or change its size.

When the tabs are turned off, the windows of open documents can be in one of the three states:

1. *Maximized*. The window occupies the whole working area of the T-FLEX CAD, and the document title is not shown. The name of the active document appears in the title of the T-FLEX CAD window.
2. *Minimized*. The window represents itself a status bar with the name of the document and system buttons for controlling the window.
3. *Arbitrary*. The window has a smaller size than that of the working window. It has a title area, in which the name of the document is shown.

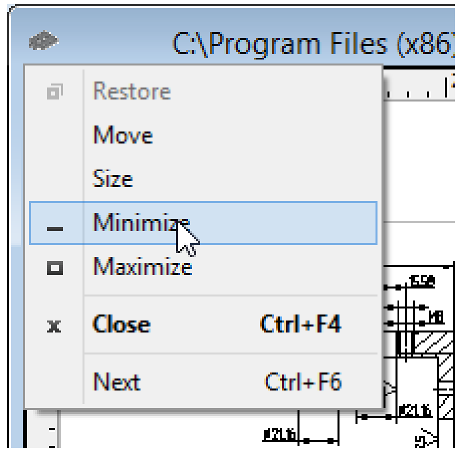
Regardless of the window size, with the tabs turned off, the document window has three buttons enabling to control its state and close the current window.



When the window is maximized, these buttons are located under the respective ones of the application window.

> Pay attention as to what button to use when closing a window.

An alternative way of maximizing a window is double-clicking 🖰 on the window title.



If a window has a title bar, that is, it is not maximized, then the window management context menu can be called by pointing at the icon at the left end of the title bar and clicking 🖰. For a maximized window, this icon is located in the very left of the application textual menu bar.

The terms "collapse" and "expand" will further be used along with "minimize" and "maximize" respectively.

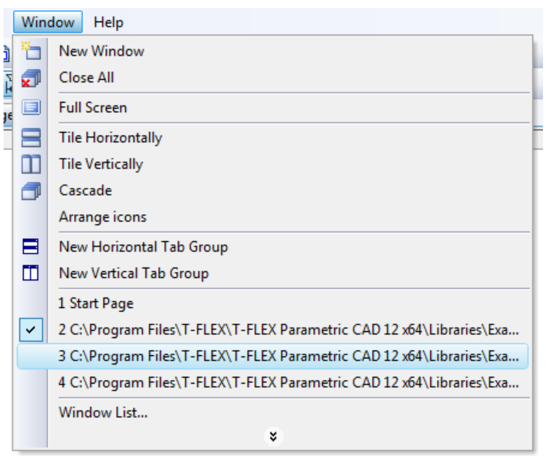
When a window is restored down, its size can be modified. Simply move the cursor over any window edge until it changes to a double-headed arrow, and then drag the edge of the window to the desired size.

Selection of active window

The active window (i.e. the window of the current document) can be selected in a number of different ways. When the document tabs are turned on, the corresponding tab should be simply pointed at with the 🖰.

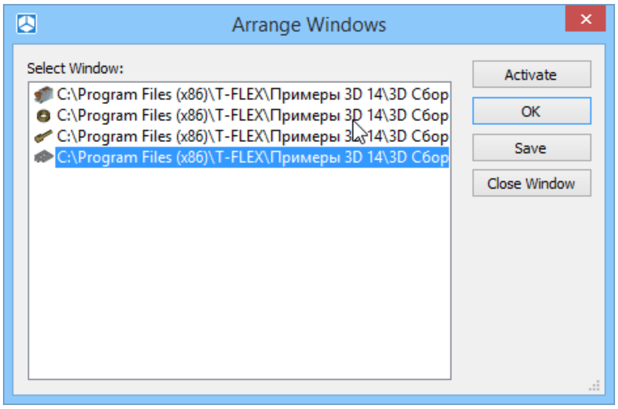
> If the document tabs are turned off, use key combinations <Ctrl><F6> or <Ctrl><Tab> for consecutive switching from one window to another.

Also, the list of open documents found in the textual menu "Window" can be used. The window that is currently active will be marked by tick in this list. To make another window active it is sufficient to point at it with the cursor and click 🖰.

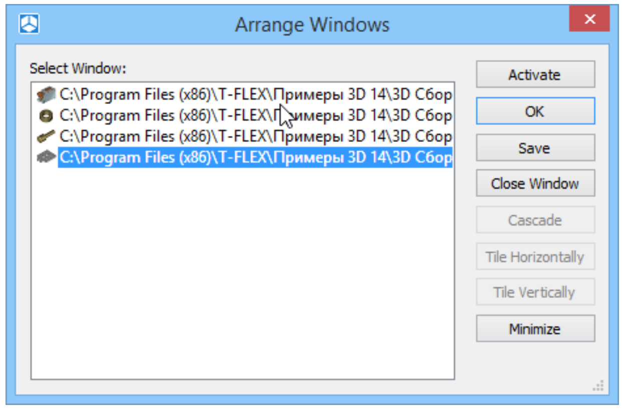


The number of windows, shown in the list, cannot exceed ten. If there are too many documents open at the moment, the command Window > Window List... can be used for selection of the specific window.

After calling this command, the dialog box “Arrange Windows” pops up. The desired window can be chosen in this dialog box from the full list of open windows. Note that the set of available buttons in this dialog box depends on the presence of document tabs.



With document tabs



Without document tabs

When the tabs are turned on, there is one more way for switching between the windows. In the upper right corner of the current window (with the document tabs arranged in a standard way – along the upper border of the window) there is a button ▾. When this button is pressed, the list of all open documents pops up, in which the desired window can be selected.

Drawing Window Scrollbars

The system supports drawings of any format. However, the screen size is fixed. For working convenience, drawings are displayed by selected portions zoomed on screen. The scrollbars help quick navigation around the drawing page.

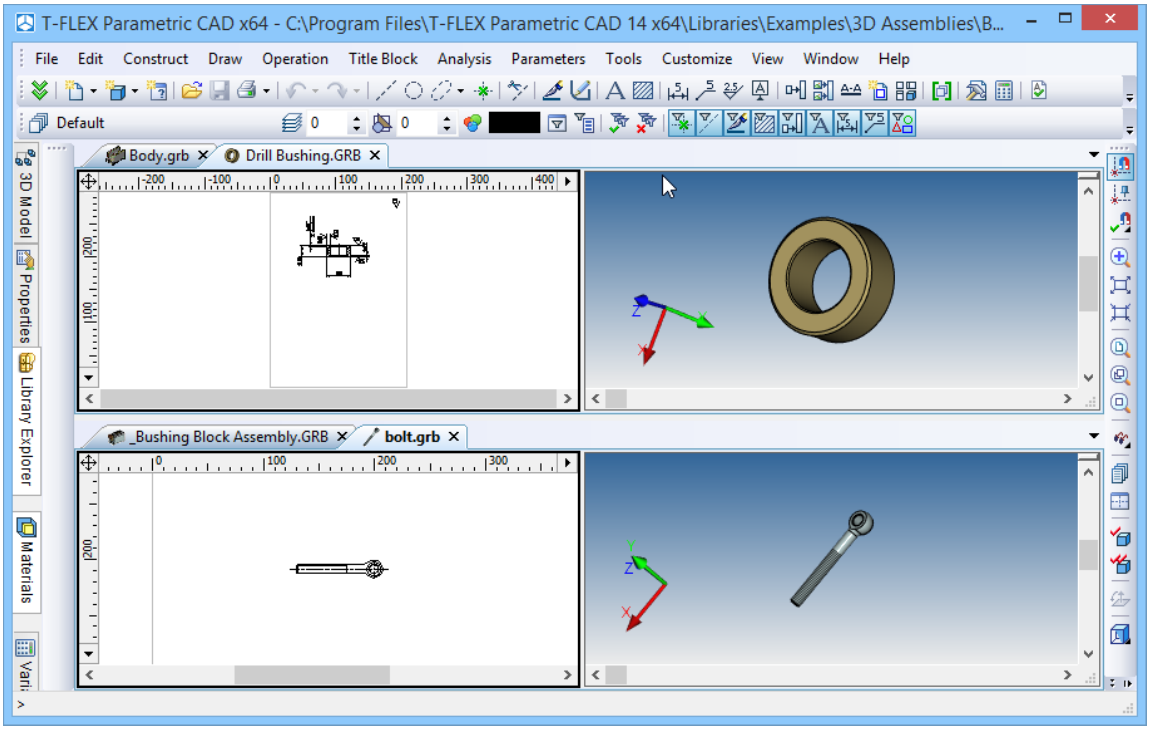
Calling the command **WSS: Show/Hide Window Scrollbars** toggles on/off visibility of the active window scrollbars:

Icon	Ribbon
	View → Window → Show Scrollbars
Keyboard	Textual Menu
<WSS>	Window > Show Scrollbars

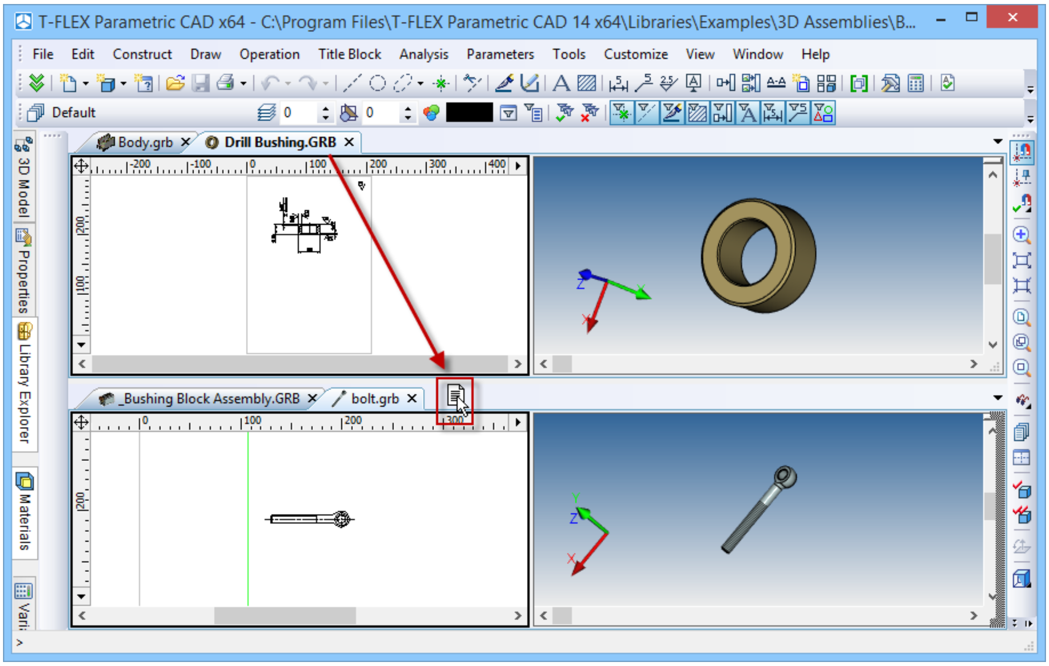
Hiding scrollbars increases the window working area. The function of the scrollbars is not available via keyboard input. However, there is a number of tools offering similar function, and in large variations. See, for instance, the rulers functionality described above, or the use of the mouse wheel.

Arranging document windows with turned on document tabs

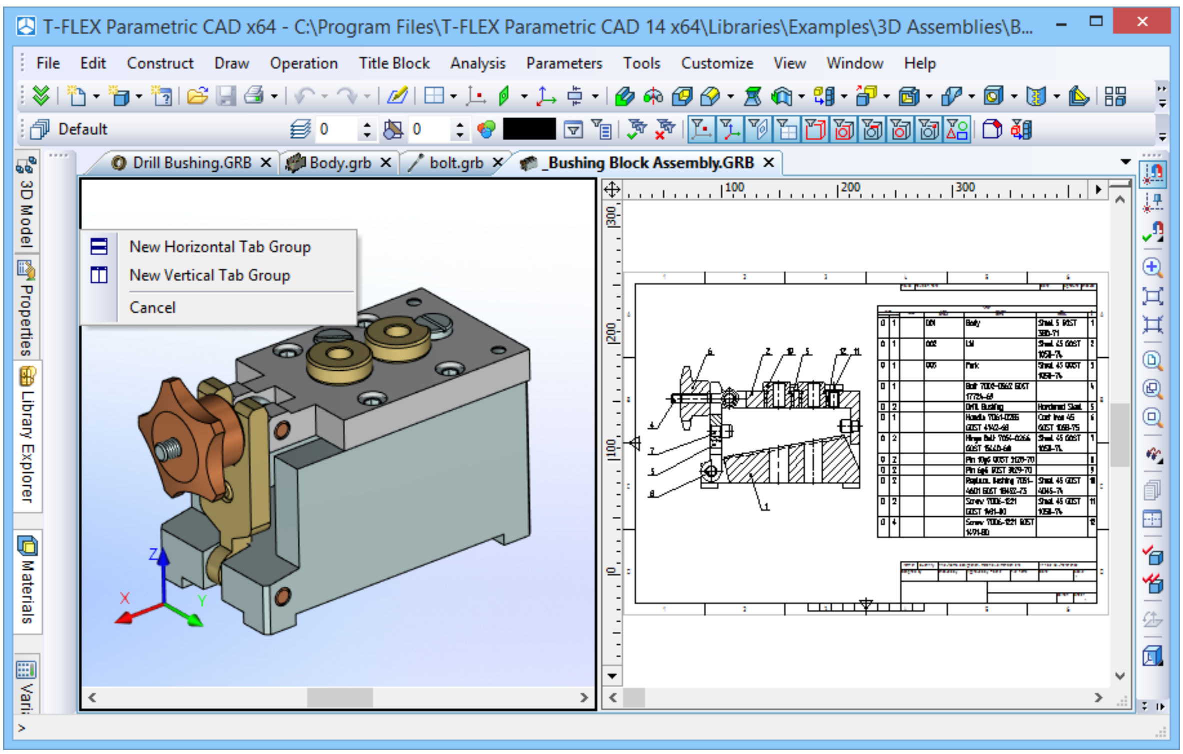
When the window tabs are turned on, the document windows can be grouped into horizontal or vertical groups. Any number of document groups can be created simultaneously, however all groups must be either horizontal or vertical.



To create a new group it is sufficient to move the tab of one of the windows to the lower or upper border of the working window of the system. To move tabs, one has to point at the document tab with a cursor, click 🖱, and without releasing the mouse, move the cursor to the required place.



When the tab is moved to the right border of the working window, the new vertical group is created, to the lower border – the new horizontal group. When the tab is moved to the upper or right border region of the working window the command menu pops up, which duplicates commands for creating groups from the textual menu “Window”.



For moving windows from one group into another it is sufficient simply to place a document tab to the desired group of tabs. For removing a group it is enough to move all windows of this group into another group.

Besides the tools described above, to create or change the groups the commands of the textual menu can also be used. The commands Window > New Horizontal Tab Group and Window > New Vertical Tab Group make new horizontal/vertical group, respectively. The document window that is active at the moment of calling the command is placed into that group. The commands Window > Move to Next Tab Group, Window > Move to Previous Tab Group allow moving the window of the current document into another group.

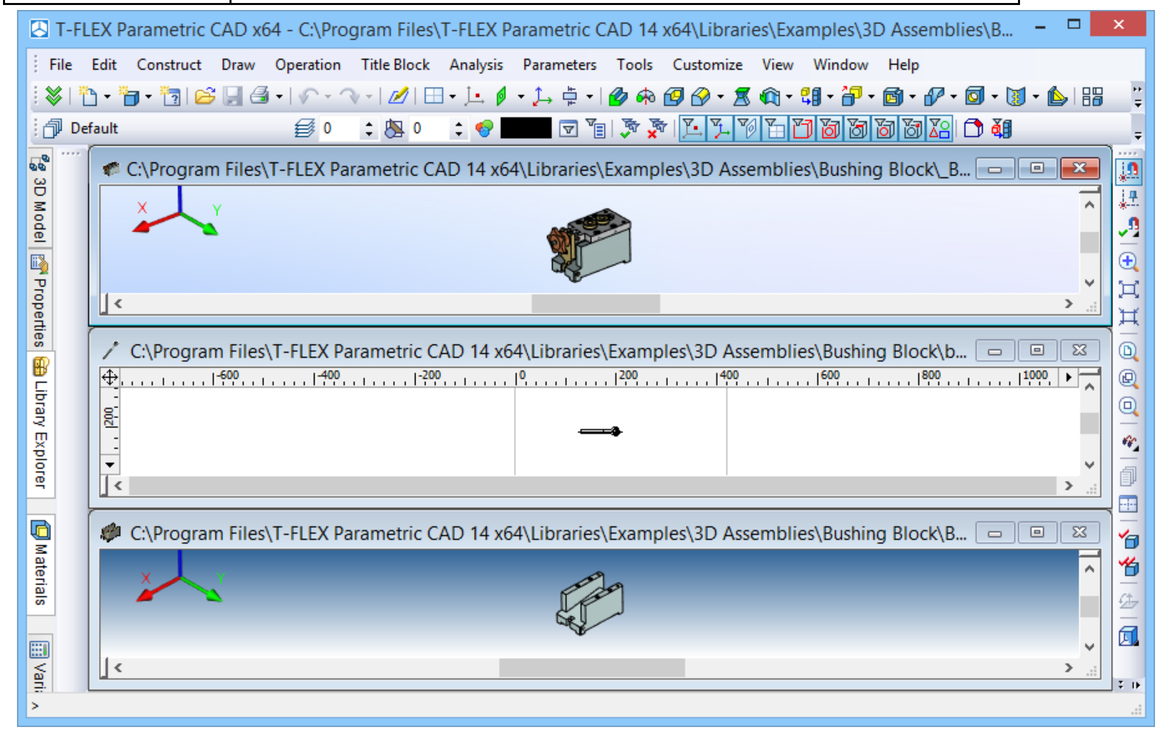
Arranging Document Windows with Turned off Tabs

If the tabs of the documents are turned off, the document windows can be expanded to the entire region of the T-FLEX CAD working window, can be diminished to the arbitrary size, can be minimized.

The document windows in this mode can be arranged in any of the traditional ways:

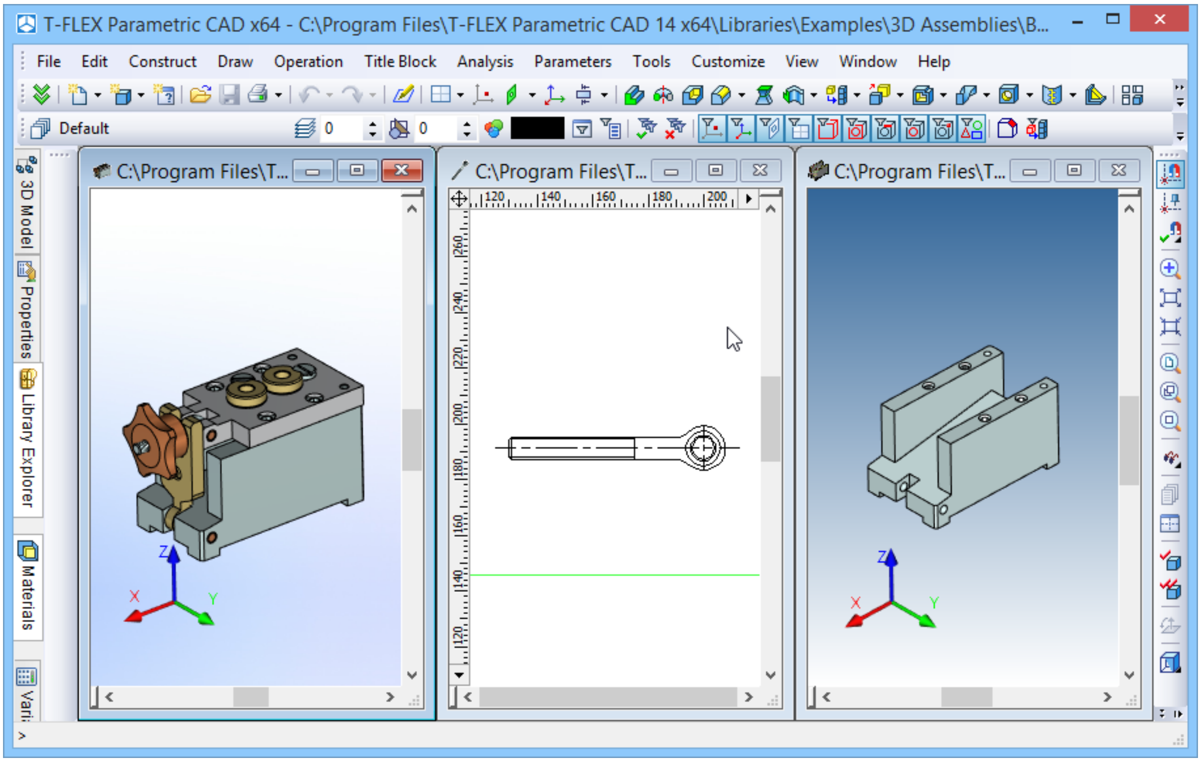
1. Tile Horizontally. Do this by the command WHT: Tile Windows Horizontally. Call the command using:

Icon	Ribbon
	View → Window → Tile Horizontaly
Keyboard	Textual Menu
<WHT>	Window > Tile Horizontally



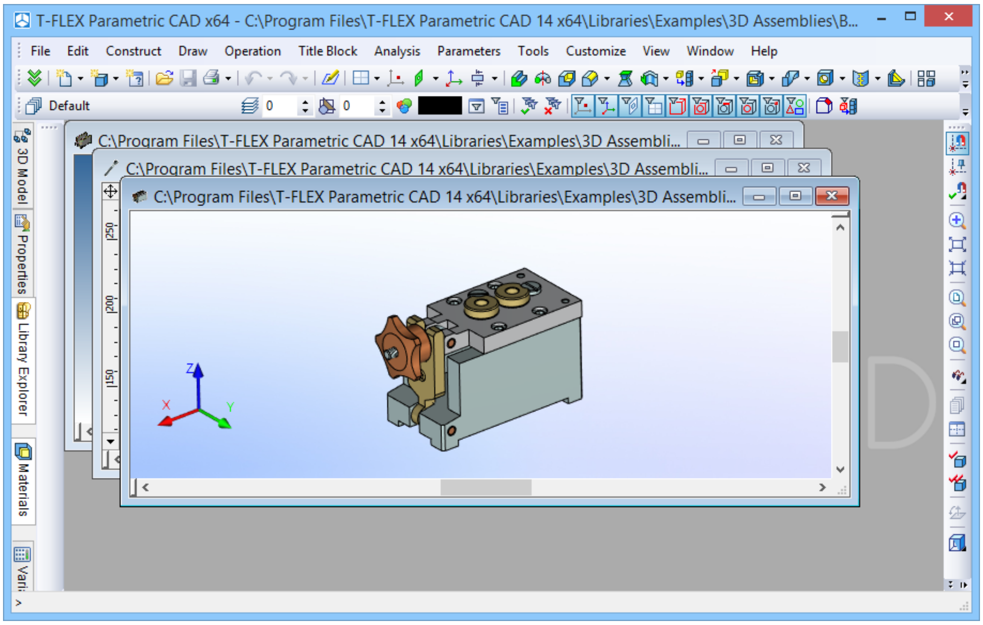
2. Tile Vertically. Do this by the command WVT: Tile Windows Vertically:

Icon	Ribbon
	View → Window → Tile Vertically
Keyboard	Textual Menu
<WVT>	Window > Tile Vertically



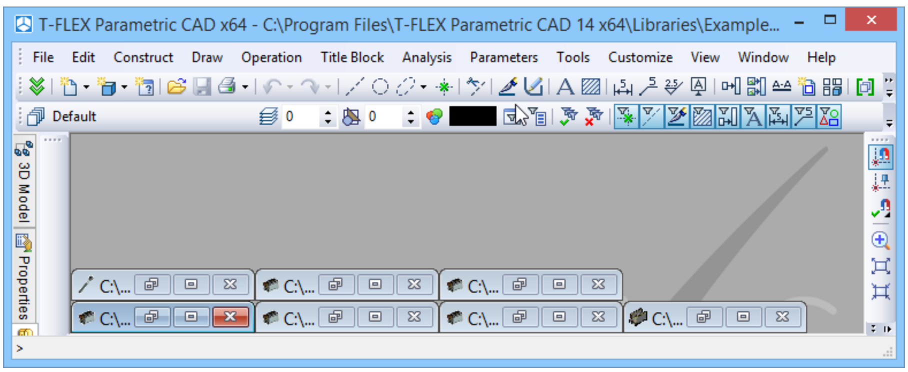
3. **Cascade**. Do this by the command **WCA: Cascade Windows**. Call the command using:

Icon	Ribbon
	View → Window → Cascade
Keyboard	Textual Menu
<WCA>	Window > Cascade



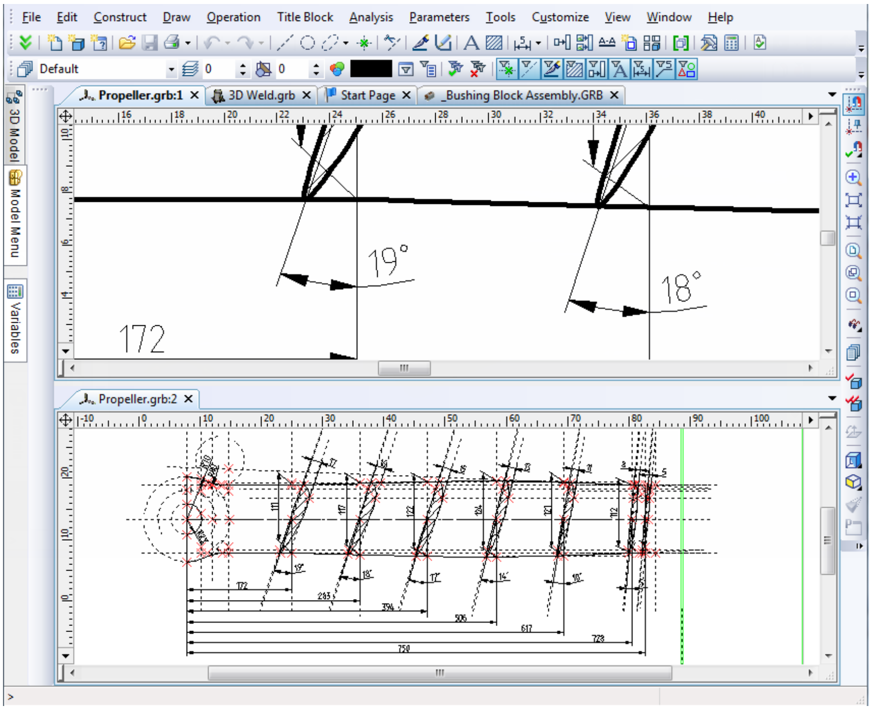
The commands WHT: Tile Horizontally, WVT: Tile Vertically, WCA: Cascade are also available when the tabs of the documents are turned on. Using these commands in this case entails a forced transition into the mode of the turned off tabs.

When the windows of all documents are minimized, they can be placed along the lower border of the working zone with the help of the command Window > Arrange icons.



Additional window of document

T-FLEX CAD enables to create additional windows for already opened documents. The name of document and the serial number of the given window appear in the title bar of such windows, for example: *"3D Drawing 3:1"*.



All operations with the drawing/model, performed in one window of the given document, will be transferred to other windows, opened for the document. For example, if an element for editing has been selected in one window of any document, the same element will be selected in another window. Additional windows can be conveniently used when the drawing contains small elements, separated from each other at significant distances, but upon constructing a drawing both types of elements are used simultaneously. It is possible to adjust the first window with the required magnification to the first group of elements, the second – to another group. And upon creating new elements it is possible to make a simple transition from one window to another and select necessary elements.

> To achieve the same purpose, splitting the document window into several panes can be applied. When this is done, inside the same document window two or four 2D or 3D windows are created, in which the drawing or 3D model of the given document will be displayed. It will be described below how to use this option.

A new window can be opened with the command WO: Open New Window. Call the command using:

Icon	Ribbon
	View → Window → New Window
Keyboard	Textual Menu
<WO>	Window > New Window

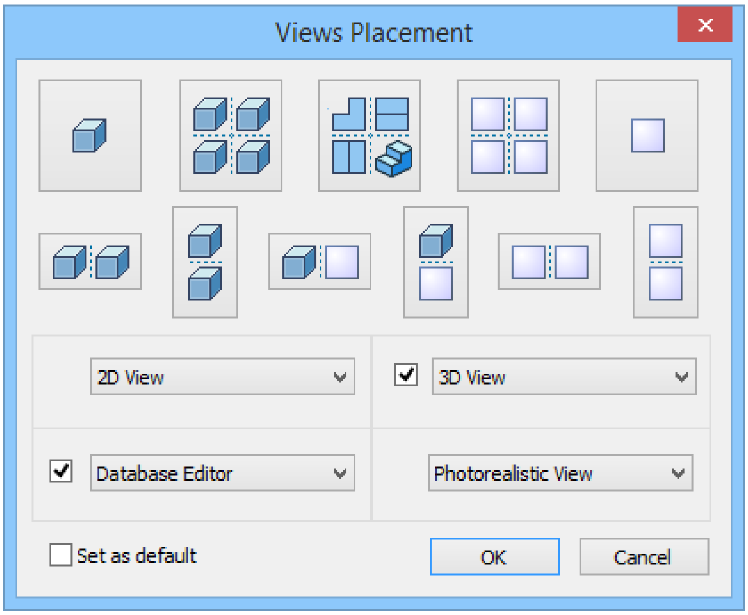
After calling this command:

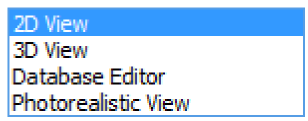
- If option Set as default is active, it automatically creates windows with preset parameters without appearance of the dialog box.
- If option Set as default is inactive, it works the same way as command Views Placement.

"Views Placement":

Icon	Ribbon
	View → Window → Views Placement
Keyboard	Textual Menu
<WSV>	Window > Views Placement

You can choose the arrangement of 2D and 3D views from the dialog box Views Placement with the help of corresponding buttons or by selection from four different types of view using pull-down lists.





Splitting Drawing Window

An active window can be split horizontally into two views by calling the command WSH: Split Window Horizontally:

Icon	Ribbon
	View → Window → Split Horizontally
Keyboard	Textual Menu
<WSH>	Window > Split Horizontally

To remove the horizontal split, toggle the icon off.

To split the current window vertically into two views, call the command **WSR: Split Window Vertically**:

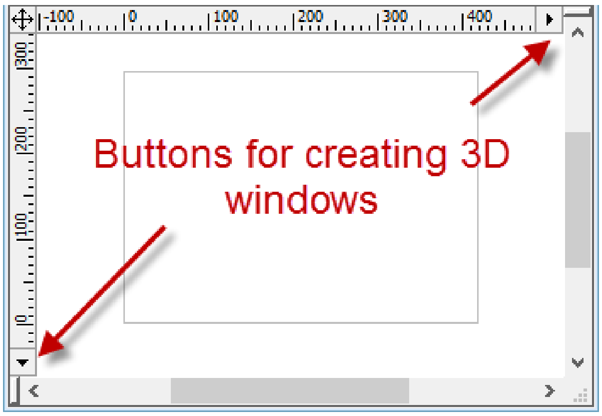
Icon	Ribbon
	View → Window → Split Vertically
Keyboard	Textual Menu
<WSR>	Window > Split Vertically

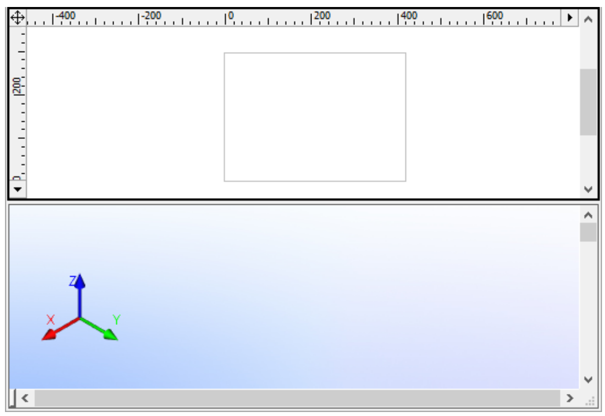
To remove the vertical split, toggle the icon off.

Consequent calling the two commands splits the active window into four panes.

Current window	
1	2
3	4

To split a window into two panes, one can also use the split boxes on the scrollbars. The split box at the left of the horizontal scrollbar divides the window into two panes horizontally.





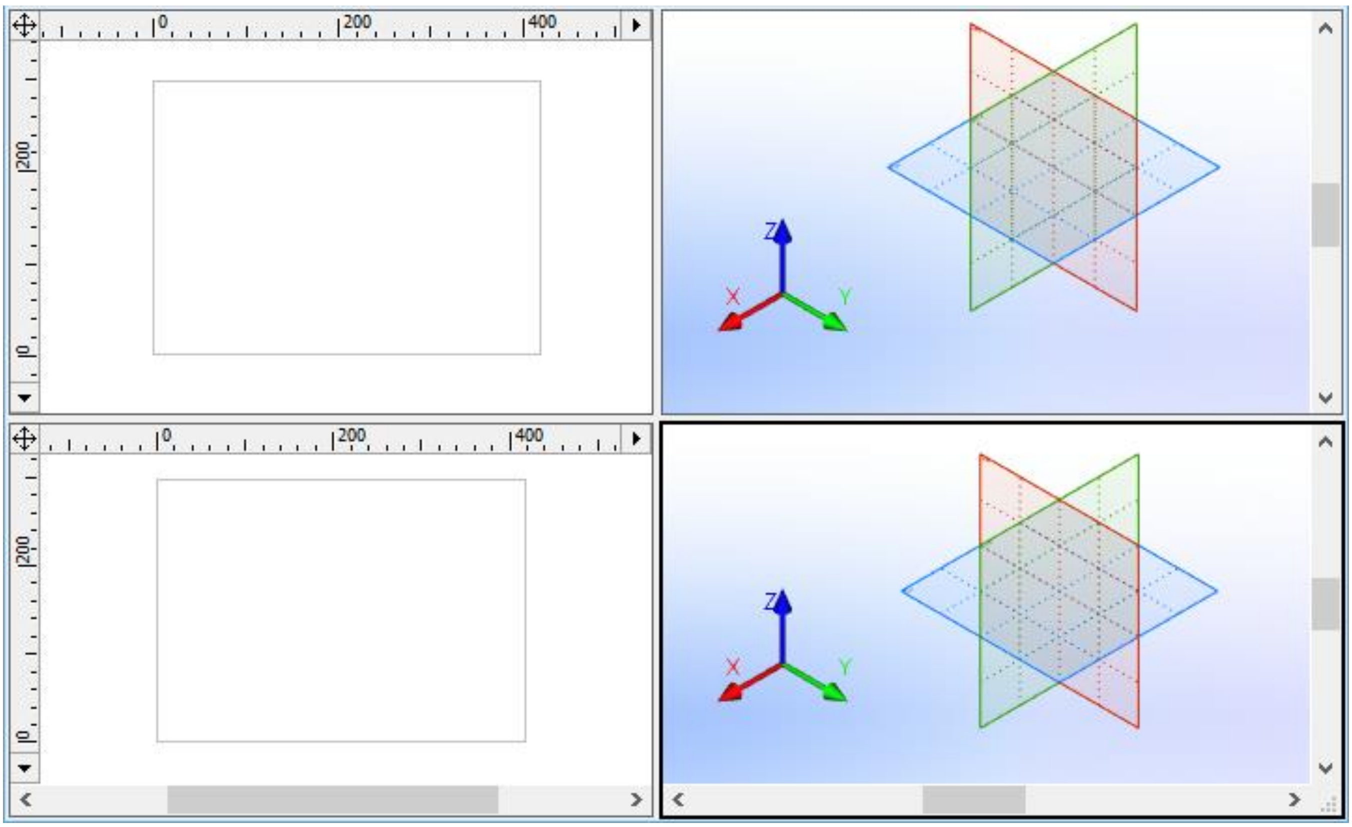
Once the window is divided into two panes horizontally, the split box disappears. The size of the panes can be controlled using the vertical split bar. Place the cursor over the split bar. As it changes to “adjust split” arrows, press and hold , and drag to the desired location of the split. To close a pane, drag the cursor beyond the respective border of the drawing window.

A window can be divided vertically into two panes in the same way.

2D windows have additional buttons for dividing a window into two or closing one of the two windows. These buttons are located on the rulers, one on the horizontal ruler at its right end, the other on the vertical one at the bottom.

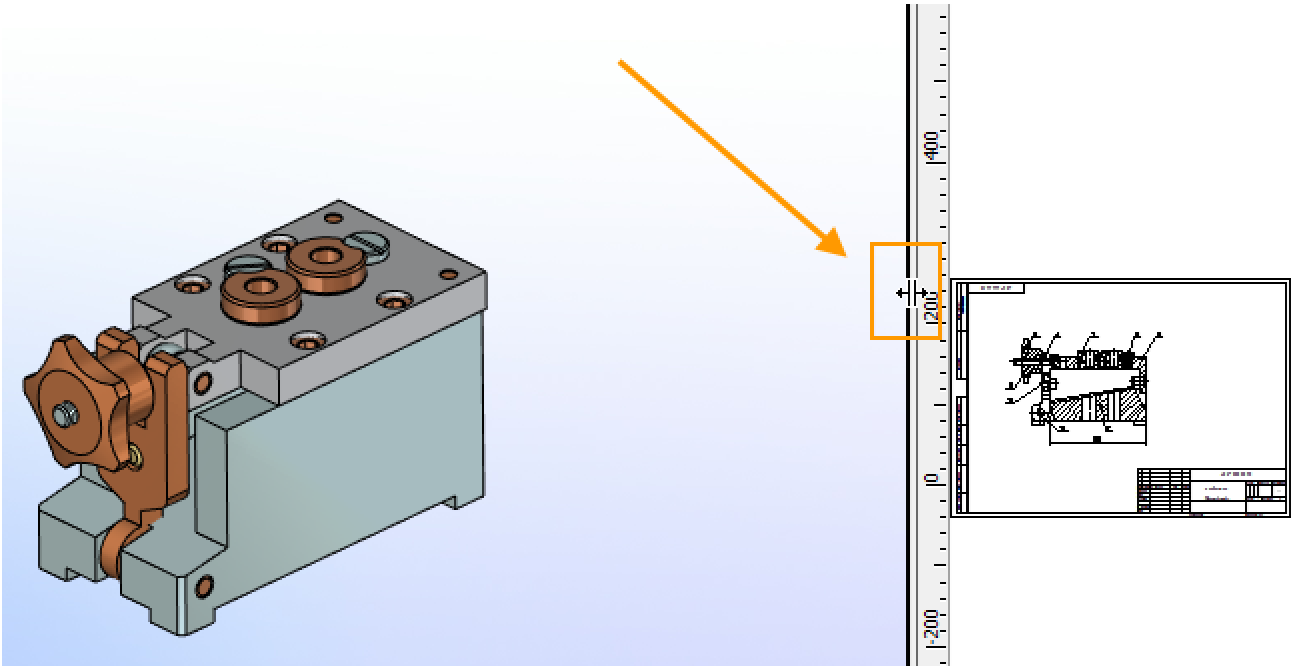
The button on the horizontal ruler works as follows. If there is currently a single window, then pressing the button splits the window vertically into two equal panes. The user is not prompted for the window type, instead, a 3D window pane is created on the right-hand side.

If the window is split horizontally into two panes, then two more 3D window panes will be instantly created by pressing the button. If the window is already split vertically into two panes, pressing the button closes the second window. If the window is split both vertically and horizontally into four panes, pressing the button will close the right pair of the panes at once. The button on the vertical ruler works accordingly. Such buttons exist in 2D windows only.



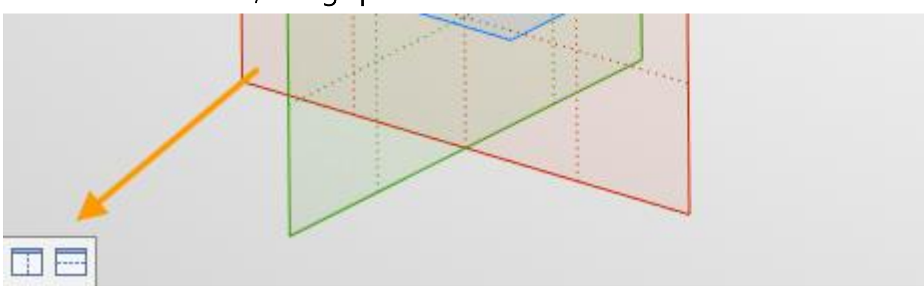
If a window is split into panes, for example vertically, then the vertical ruler is used for both parts. Actions performed with this ruler are reflected in the pane which is currently active. In the case of vertical division each pane has its own horizontal ruler. If the window is split horizontally, then two vertical and one common horizontal rulers are used. If the window is split into four panes, then four rulers work, their actions are reflected in the pane which is currently active. To make a certain window pane active point at it with a cursor.

To get rid of window splitting it is enough to reduce the size of one of the panes up to zero. By reducing the size in this way the pane will be removed.



Splitting of the document window can be retained for further sessions with the document. To do so, enable the **Fixed set of windows** flag in the dialog box of **ST: Set Document Parameters** (the **3D** tab) and save the document. When you reopen the document, the window will be divided as it was at the time you saved the document after setting the above parameter. You will not be able to change the document splitting by changing windows size to zero (moving the window's separator), or by adding a new split bar.

You can split the current 3D window, using special buttons in the left bottom corner.



Closing document window

To close the window of a document the button × in the right upper corner of the window is used (with standard arrangement of document tabs). Upon pressing this button the current window will be closed. If for the given document several windows were open, then the rest of the windows remain open.

To close all windows of the current document at once the command FCL: Close Model can be used:

Icon	Ribbon
	File → Close
Keyboard	Textual Menu
<FCL>	File > Close

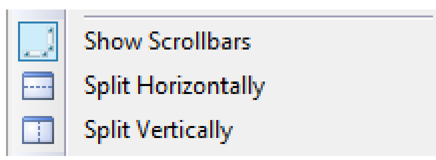
To close all windows of all open documents the command WCS: Close all Windows can be used:

Icon	Ribbon
	View → Window → Close All
Keyboard	Textual Menu
<WCS>	Window > Close All

> After calling these commands the windows in which the drawings were not modified will be automatically closed. System will offer you to save changes for unsaved modified documents before their closing.

Flagged Commands

In the textual menu, the icons on the left of command names indicate the command on/off status. Thus, for instance, the following diagram represents a situation when the active window is split vertically, its scrollbars being hidden.



Managing Multi-Page Documents

A T-FLEX CAD document may contain multiple 2D pages. The 2D window may be displaying all or only selected document pages, depending on the drawing settings. When working with a multi-page document, the user can manage the visibility of pages by removing from display those not being currently worked on.

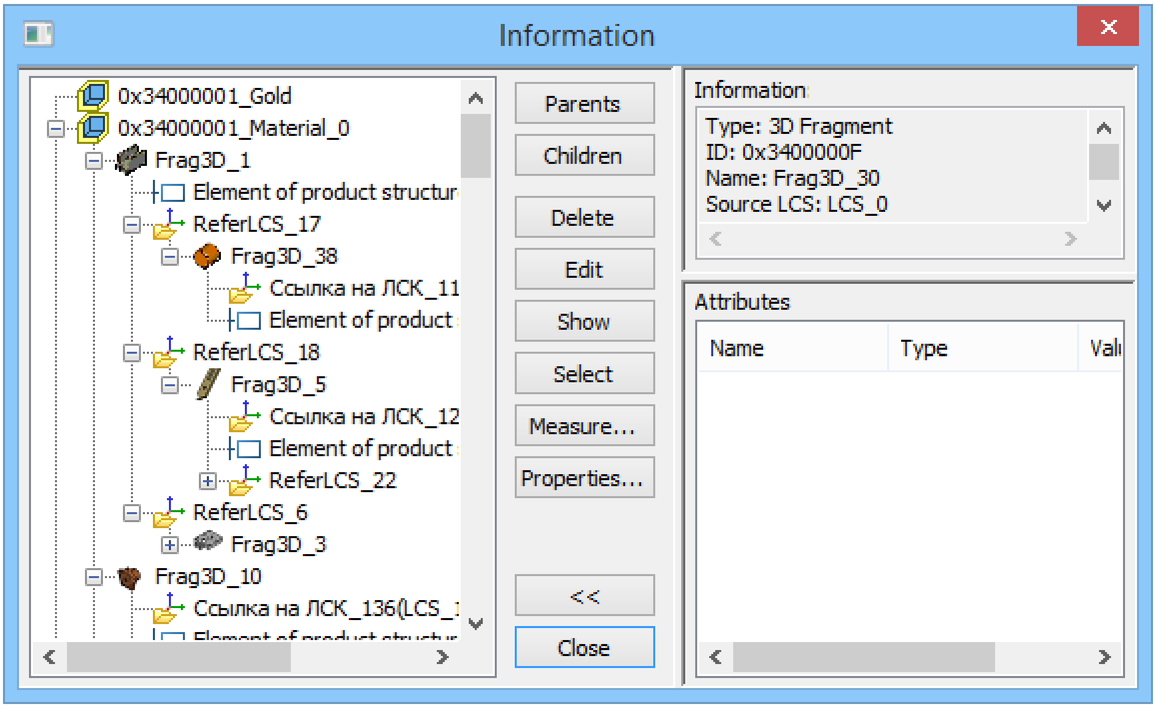
If a T-FLEX CAD document contains several pages, then the tabs with the names of the visible pages may be shown in the lower part of the drawing's window (with the default settings; the arrangement of the page tabs can be modified). One can switch from page to page using those tabs, by clicking them with , or by using the keys <Page Up>, <Page Down>. The tabs can be hidden/shown using the command Customize > Tool Windows > Page tabs.

> See details on working with multi-page documents in the chapter “Pages”.

Information Window

Keyboard	Textual Menu
<Alt><F1>	Help > Information

Calling this command brings up a dialog box that provides access to all current document elements for editing and information querying. Unlike the “3D Model” window, this dialog box displays all 2D and 3D elements.



The current document elements are displayed in a large pane on the left-hand side. The hierarchic structure is represented by the tree, with the base elements of the drawing or the model at the root. (The base elements are those created in absolute coordinates and not referencing any parents.) To select an element, click it with . The selected element will be highlighted on the drawing or in the 3D window.

The following buttons become accessible for selected elements:

[Parents] reformats the model tree, leaving only the selected element and those elements referenced by this one.

[Children] reformats the model tree, leaving only the selected element and the elements that reference this one.

[Delete] closes the dialog box and calls the deletion command on the selected element.

[Edit] closes the dialog box and calls the editing command on the selected element.

[Show] closes the dialog box and zooms the active drawing (model) window on the selected element.

[Select] closes the dialog box, leaving the element selected for further manipulations.

[Measure…] calls the "Measure Element" dialog box for reading geometrical data of the selected element. The Model Tree dialog stays on screen for further actions.

[Properties…] calls the parameters dialog of the selected element. The Model Tree dialog stays on screen for further actions.

[Close] closes this dialog box.

[<<] [>>] hides/shows the following additional panes in the "Information" dialog box:

Information. This pane displays brief information about the selected element.

Attributes. This pane displays information about the attributes of the selected element.

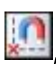
Creating and Editing Drawing Elements

The system provides a specific command for creating and editing each type of model elements. This section describes main concepts of using these commands, as well as general principles of creating and editing a 2D drawing.

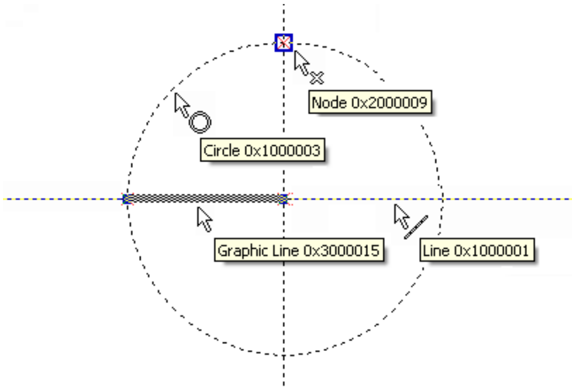
Snapping Mode. Snap Types

T-FLEX CAD system supports two distinct modeling modes:

- ✓ One is free mode in which the elements are selected within commands using the automenu and the keyboard.
- ✓ The other, object snapping mode, provides pre-highlighting of the elements available as references in element creation and editing commands. The latter mode is enabled by default on starting the application.

The pictogram , located on the toolbar "View", controls the snapping modes. Use this icon to enable or disable the object snapping mode.

An element is pre-highlighted in the object snapping mode as the cursor approaches the element. Meanwhile, the cursor itself gains a mark corresponding to the pre-selected element, and a popping up help message displays the name and Id of the element. On the screen this looks like the following diagram:



The pre-highlighted element can be selected using the mouse. This relieves the user from using the automenu or the keyboard in most cases.

Various construction and graphic elements are pre-highlighted in creation and editing commands only when it makes sense. Thus, for instance, in spline creation, only nodes will be pre-highlighted, as the spline is created based on a set of nodes. No other elements will be pre-highlighted on cursor approaching, as this does not make sense for spline creation.

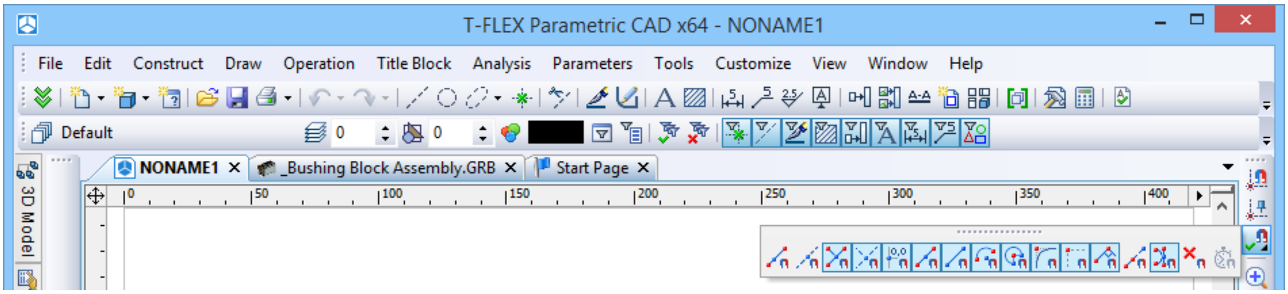
> Please note that the current documentation refers to the element selection mode with disabled object snapping when describing commands (implying only the use of automenu options).

To temporarily disable object snapping within a command, hold the <Ctrl> key down. Snapping is suspended as long as the key is held.

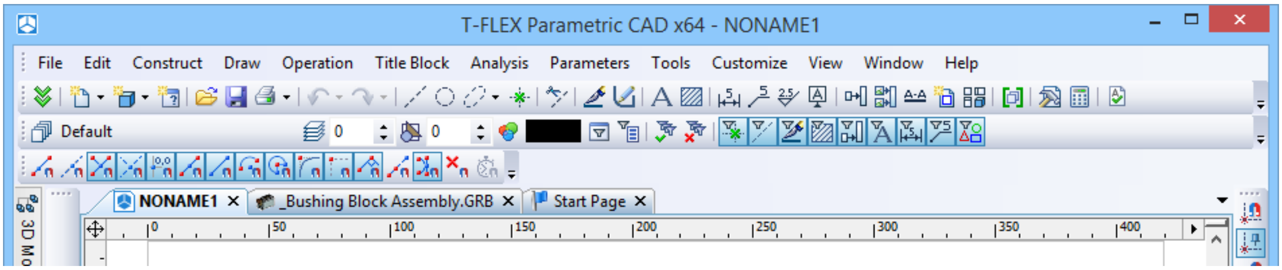
When defining positions of various 2D elements in their creation/editing commands, with the object snapping enabled, not only can you use the existing elements (construction lines, graphic lines, nodes etc.), but also select characteristic points defined by *object snaps*. Nodes can be automatically created in the selected points. Those could be nodes at intersection of construction lines, nodes from fragments, nodes on dimensions, leader notes, tolerances and text entities, nodes aligned vertically/horizontally with another 2D node, nodes at the center of a graphic circle line or circular arc, etc.

> The most number of object snaps is used in the sketch-creating command SK: Create Sketch. Some of the object snaps may be unavailable in other 2D commands. Besides that, the use of snaps is affected by the settings made in the command **SO: Set System Options**. You specify what snap types can be used when working with a 2D drawing on the Snaps tab of this command. There you can also set the priority for each snap. Snap priorities determine, in what order the system will offer them to the user (in the cases when several snap choices are found).
>
> A detailed description of setting up snaps in the command **SO: Set System Options** is given in the chapter “System setup”.

Most of the object snaps can also be managed using the specialized "Snaps" toolbar. By default this toolbar is "hidden" inside the toolbar "View". To get an access to this toolbar, press the button .

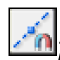
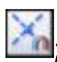
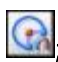


For displaying this toolbar in an "independent" mode, move the cursor to the title area of the toolbar, press and, without releasing the mouse, drag the toolbar into the desired location. In the future this toolbar can be left in the floating mode or snapped at any place of the T-FLEX CAD window.

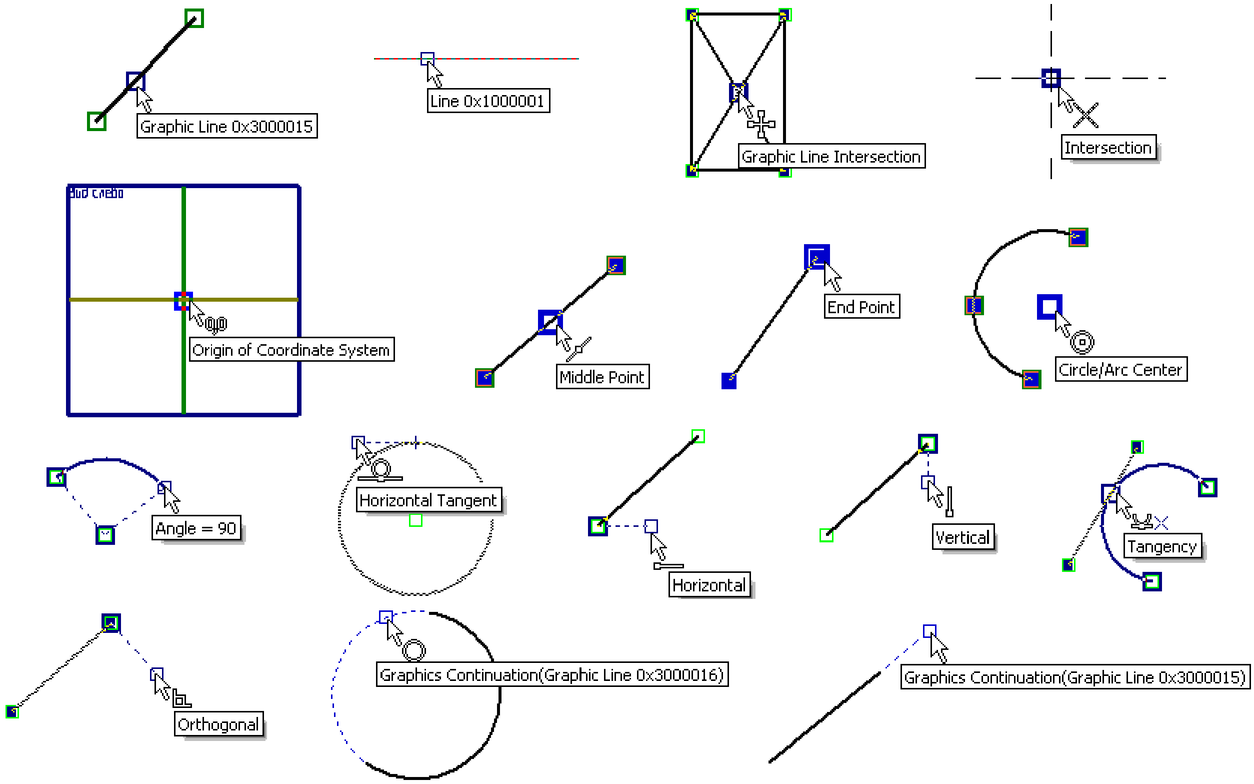


Using this toolbar, one can set and unset the snapping modes by clicking the desired icons with . All snappings can be simultaneously turned on or off by the button  - "Clear all sketch Snaps". Also, all snapping modes except the required one can unset by clicking appropriate icon with <Ctrl> button pressed.

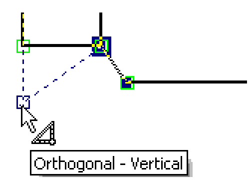
Listed next are the main types of object snaps used in T-FLEX CAD:

- Snapping to a point on a graphic line or construction line – , ;
- Snapping to graphic line intersection – ;
- Snapping to construction line intersection – ;
- Snapping to the coordinate system origin ((0,0) point) – ;
- Snapping to the midpoint of the graphic line – ;
- Snapping to graphic line end points – ;
- Snapping to the center of an arc or circle – ;
- Snapping to arc angles 90°, 180°, 270° – ;
- Vertical/horizontal tangency to circle – ;

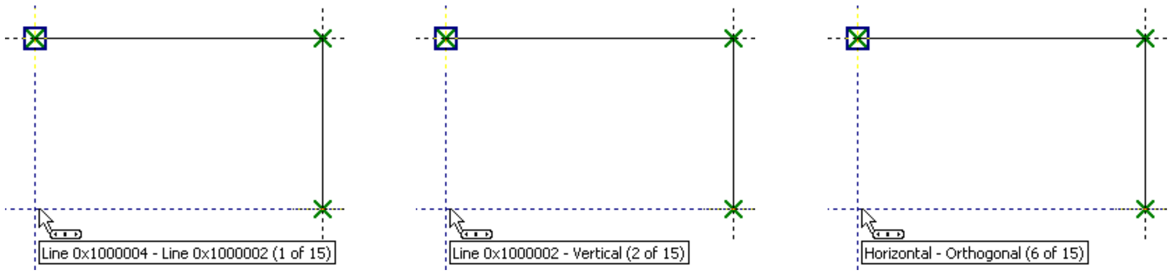
- Cursor becoming aligned horizontally or vertically to another element point or 2D node – ;
- Automatic definition of a line normal – ;
- Cursor becoming aligned to the extension of a graphic line – ;
- Automatic definition of a tangency to an arc or circle – .



In the creation/editing process, the system automatically finds the allowed snaps and offers them to the user (by flashing a snap type next to the cursor). Besides that, the system monitors for a coincidence of two object snaps, for example, vertical – horizontal, perpendicular – horizontal, etc.

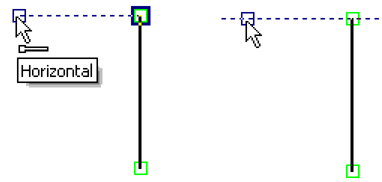


If several object snap choices are found at a given point, the system lets the user select the desired snap (or a combination of two snaps). To do this, you need to place the cursor at the desired location and rest it for a while. Then the cursor changes its appearance: the mark appears next to it together with a tooltip showing the total number of object snaps found by the system. Use the mouse wheel to scroll through those snaps. Clicking determines the snap that will be used in the creation or editing of the current 2D element.



A system-offered object snap can be locked by the <Spacebar> function key.

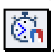
For example, let's fix horizontal snapping to one of the segment nodes. To do this, get horizontal snapping with this node and press the key <Shift> or <Spacebar>. A temporary dotted line will be constructed through this node, the cursor sliding along as a free node.



Snapping that are turned on on the sketch snapping toolbar stay active continuously throughout the sketch command session. If snappings are adjusted often, one can use temporary object snappings -the "one-action" snappings.

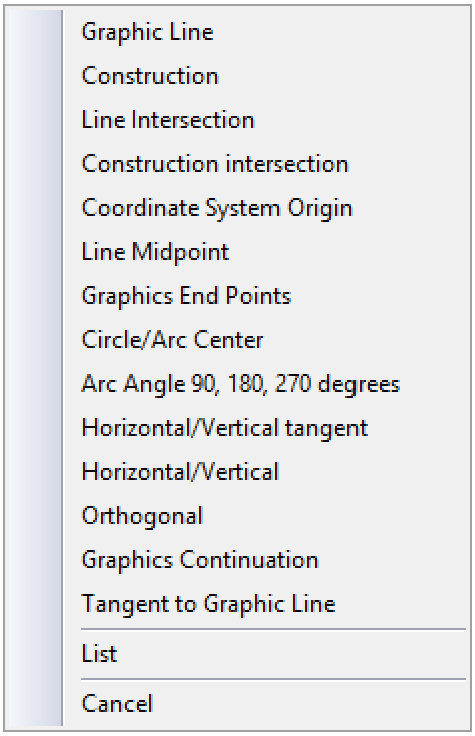
Such a snapping can be turned on by several means:

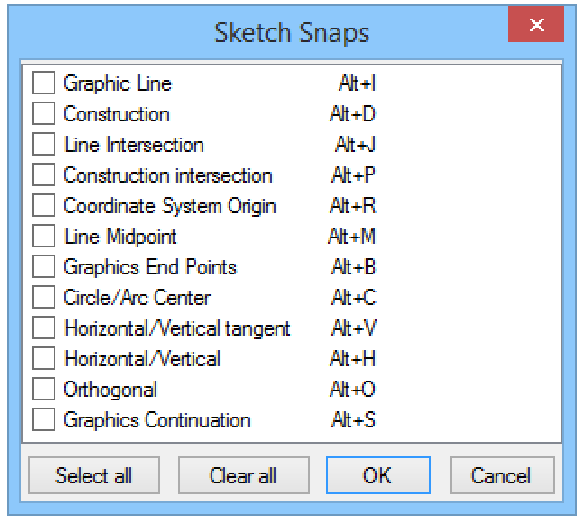
By the button  on the snappings toolbar. This brings up a context menu for specifying a temporary snapping (just one); it also lists key combinations that can be used for invoking a temporary snapping without calling the menu. To define several temporary snappings, use the item **[List]**. Upon picking the item, the context menu is replaced by a dialog box that allows turning on several temporary snappings simultaneously.

By pressing and releasing the middle mouse button or the wheel button while keeping the mouse pointer still in the working window area. As a result, the same menu will appear on the screen as when using .

By pressing key combinations assigned to each snapping.

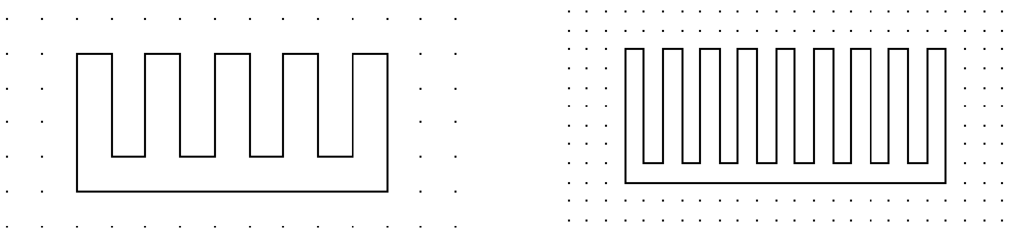
When temporary object snapping is turned on, all permanent snappings are ignored. The described temporary snappings act until the first click .





Using Grid

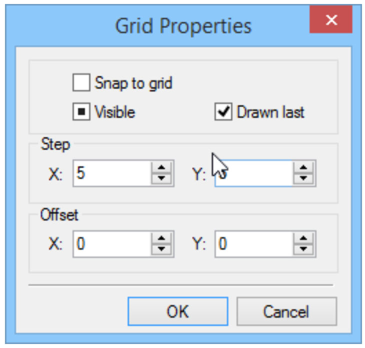
When creating a drawing, it is sometimes helpful to use a grid of dots. In this way, snapping will occur to the grid dots while creating various drawing elements. The precision with which you create drawing elements can be controlled by specifying the appropriate grid step.



The grid can be turned on for the active page by the command QG: Change Grid Settings:

Icon	Ribbon
	Edit → Document → Grid
Keyboard	Textual Menu
<QG>, <ALT><F6>	Customize > Grid

The following required parameters are defined in the Grid Properties dialog box:



Visible. Sets the display mode of the grid. The grid color is defined in the system options (the **SO: Set System Options** command).

Snap to grid. Sets the element snapping to grid mode.

Drawn last. Defines the order of drawing the grid on screen.

Step X. Defines the grid step along the X-axis of the drawing.

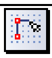
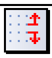
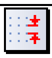
Step Y. Defines the grid step along the Y-axis of the drawing.

Offset X. Defines the grid shift along the X-axis of the drawing with respect to the origin (0,0).

Offset Y. Defines the grid shift along the Y-axis of the drawing with respect to the origin (0,0).

The grid options are saved with the drawing.

The grid management commands are accessible via the textual menu Customize > Snap:

	<Ctrl><G>	Grid Snap On
	-	Enlarge Grid Step (doubles)
	-	Reduce Grid Step (halves)

If the grid snap is turned on then the grid knots serve as the snapping nodes for the drawing elements.

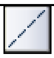
General Concepts of Element Creation

Placement of any element on the drawing can be defined in the following ways.

Independent of other elements. This kind of placement is defined by the absolute coordinates of the element on the drawing, independently of other element locations. Placement of such elements is usually set by clicking  or by assigning exact values of snapping coordinates in the command's properties window.

Dependent on reference elements. The element location will depend on the location of the reference element this one is related to. When the location of the reference elements is modified, the current element will relocate accordingly.

To select the reference elements to snap to, the options are provided for selecting a line, a circle, a node, etc. in most 2D element creation commands. The variety of available options depends on the element being created. The most commonly used snapping options are presented below:

	<L>	Select Line

	<C>	Select Circle
	<N>	Select Node
	<E>	Select Ellipse
	<S>	Select Spline

When the object snapping is on, use of these options is not essential. However, using the options in this case helps narrow down the range of elements available for snapping. Thus, for instance, with the option active, only circles will be pre-highlighting when moving the cursor around the drawing.

When creating and editing elements, an earlier created relation of this element with another element can be abolished by the following option:

	<K>	Break (kill) relations

Object snaps can be used in both ways of defining the 2D element position. The set of the available snaps depends on the current command. By using snaps, a 2D element being created can be tied to:

- ✓ a free 2D node automatically created at the specified location (that is, not tied to objects used for snapping);
- ✓ a tied (constrained) 2D node automatically created at the specified location (the tie of the node with the source elements is maintained);
- ✓ in free coordinates (snaps define only the absolute coordinates of the element being created).

Tied nodes are always created when having snaps to a construction line intersection, circle center, end points of graphic lines, characteristic points of drawing annotation elements (dimensions, leader notes, roughness symbols, tolerances), as well as 2D fragments.

When using all other snap types, the status of the auto-parameterization mode is regarded (the icon on the "View" toolbar). If the auto-parameterization mode is enabled, then a tied node is created. Upon disabling the auto-parameterization mode, either a free node is created, or a point is picked with appropriate coordinates (when creating a leader note, roughness symbol, tolerance, section view and 2D fragments).

Most creation commands allow setting parameters of all newly created elements. To do that, parameters need to be set right after the input of the command, before the start of element snapping and assigning its location. Assigning parameters can be done in either command's properties window, or in a special parameters' dialog box, called by the following option:

	<P>	Set parameters

The parameters of a particular element being created can be defined in the command's properties window during its creation. One can also use the option , but only if calling it during an element creation process after defining its position and snapping.

Commands for creating some of the 2D elements (dimension, roughness, leader note) provide option of assigning parameters from already existing element of the same type:

	<Alt+P>	Copy Properties from Existing Element

Values of the copied parameters can be set as default parameters (parameters that will be assigned to the newly created elements of this type).

Any creation or construction command allows calling the editing command from within, using the option:

	<F4>	Execute Edit Element command

You will return into the original element creation or construction command after completing editing in the editing command.

In the number of 2D commands for completion of element creation you must use option:

	<Ctrl+Enter>	Finish input

Canceling an element selection performed within a creation or editing command is done by the option:

	<Esc>	Cancel selection

This option does not cancel the command itself.

To quit a command, use the option:

	<Esc>	Exit command

General Concepts of Editing Elements

In editing commands, element selection is done by the cursor. To select, move the cursor to the element and click or press <Enter>. Different elements are highlighted in different ways. Some are painted with colors, others surrounded by a frame. To relocate a selected element, move the cursor to the desired position and click . The element will relocate (if the method of its snapping allows that).

If a wrong element was selected, cancel the selection with the option:

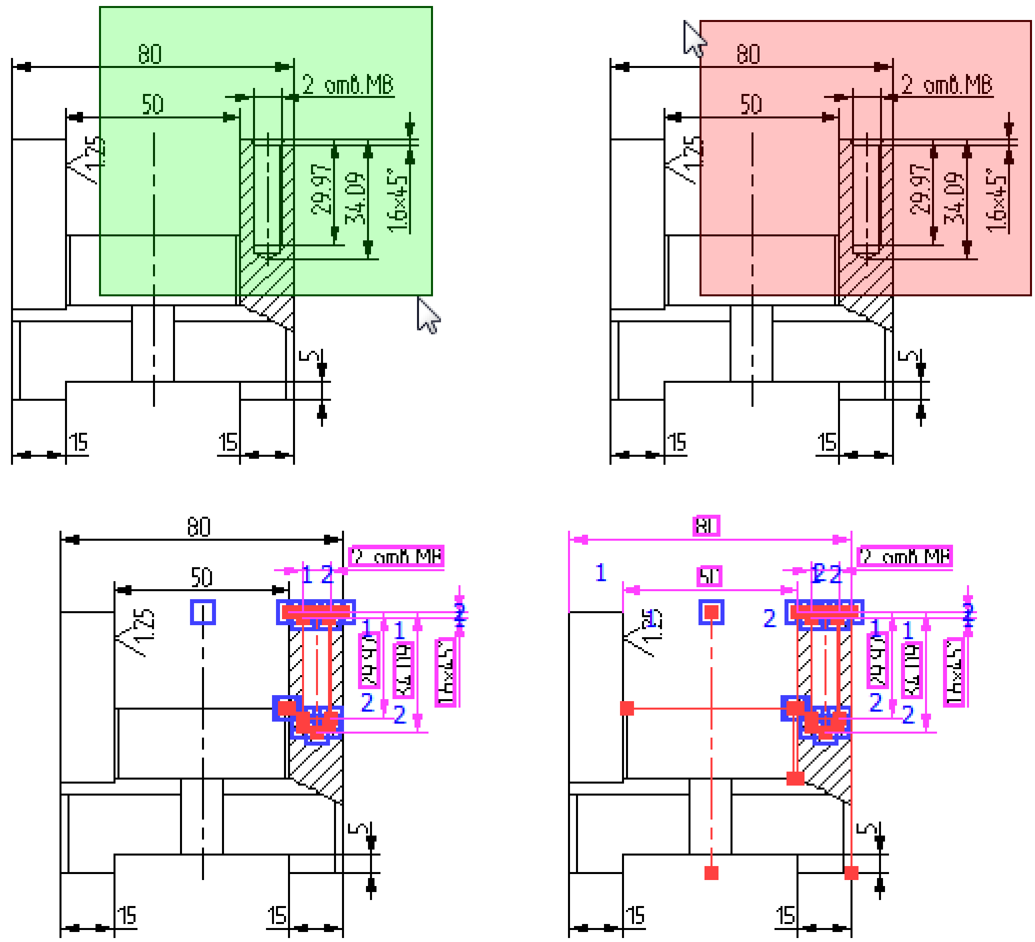
	<Esc>	Cancel selection

or select the next nearest one using the option:

	<I>	Select Other Element

The subsequent elements of the given type can be selected by using this option repeatedly.

In editing commands, the user can select multiple elements using box selection. To do so, move the cursor to the intended location of one corner of the box, press and hold , and drag the cursor to the location of opposite corner of the box, then release. If the cursor was moved left-to-right when marking the box, all elements that are fully within the specified region are selected. In this case the selection box is painted with a green color. If the cursor was moved right to left, the objects are selected by the crossing frame. That means that not only the objects that are entirely within the selection box, are selected, but also the objects intersected by the box. In this case the selection box is painted with a pink color.



A group of elements can also be selected by subsequent picks with the <Shift>+ combination. An element can be excluded from selected by picking it with the <Ctrl> + combination.

All existing elements of the given type can be selected at once using the option:

	<*>	Select All Elements

Selecting an element from a list is done using the option:

	<R>	Select element from list

The list can be composed differently for elements of different types. For instance, when editing fragments, the list will contain all model fragments, while when working with nodes, the list will contain only the named nodes.

All editing commands allow deletion of a single or multiple selected elements, using the option:

	<Del>	Delete selected Element(s)

The following option is available within common 2D element editing commands:

	<O>	Create Name for selected Element

This option allows assigning a name to the selected element. The name is a unique attribute of an element and can be used, for instance, for searching elements using the command FD: Find Element, for selecting elements in a list, and for creating nodes from fragments within the EN: Edit Node command. When the entered name is the same as a one already assigned to another element, the system will output the message "Incorrect Element Name or Name already exists".

The 2D node editing command allows assigning names to multiple selected nodes simultaneously. In this case, the names are made by appending subsequent numbers to the entered name, for instance, *"name1"*, *"name2"*, etc.

When 3D elements are constructed or created, the system assigns them "default" names. If necessary, the user can change a name in the element parameters window.

Editing commands allow the user to change selected element parameters. This can be done directly in the command properties window (just like at the time of creating this 2D element), if only one element was selected for editing.

If several elements are selected, use the option:

	<P>	Set parameters

After calling the option, a dialog box comes on screen first, offering to select the parameters to be modified. Next, the parameters dialog appears. Any changes to parameters not selected for editing in the previous dialog, will be ignored. Some parameters of the selected elements can be modified using the system toolbar.

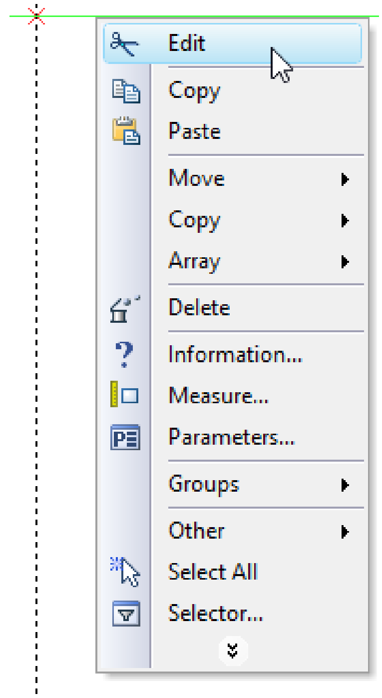
When editing dimensions, roughness symbols, leader notes, just like at the time of their creation, you can copy parameter values for the edited element from another element of the same type, using the option .

Selecting elements outside any command

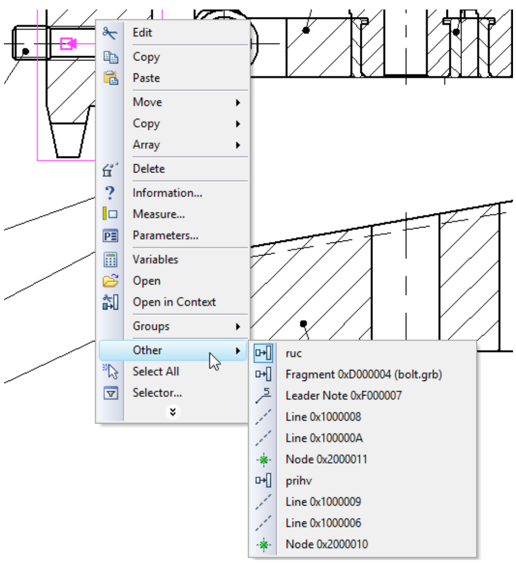
Elements can be selected for editing even outside any command, when the system is in the command-waiting mode.

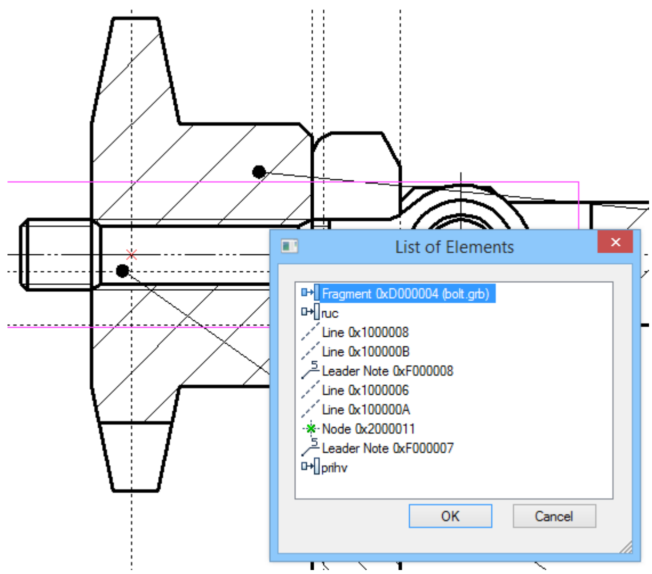
Selecting an element with automatically starts the given element editing command. Double-clicking will start the editing command and bring up the element parameters dialog box.

The context menu of an element can be accessed by right-clicking on the element. The menu contains items for editing, deleting, moving and copying the element, as well as changing its properties by calling the parameters dialog box. One can also view the information about the selected element, measure it, and change the selector settings.



When working with complex drawings, several elements might be near the cursor. To select the desired element in this situation, use the "Other..." item in the context menu for selecting the element from list. The list contains the elements nearest to the cursor. Only the elements allowed by the selector settings are included in the list. The number of the nearby elements in the list can be set in the selector settings dialog box. This dialog also provides the options for the list representation. The latter can appear as a context menu or as a resizable dialog box floating on screen, providing the user better view of the drawing elements.





A group of elements can be selected in the command-waiting mode as well. Just like in the case of the editing commands, various methods can be used for the group selection: selecting by box left

> to right (selected are all elements that are fully within the specified region); selecting by box right to left (selected are all elements that at least partially enter the specified region); a serial selection of elements using <Shift>+, <Ctrl>+. The context menu will contain the commands for moving/copying, deleting and modifying properties of the selected elements.

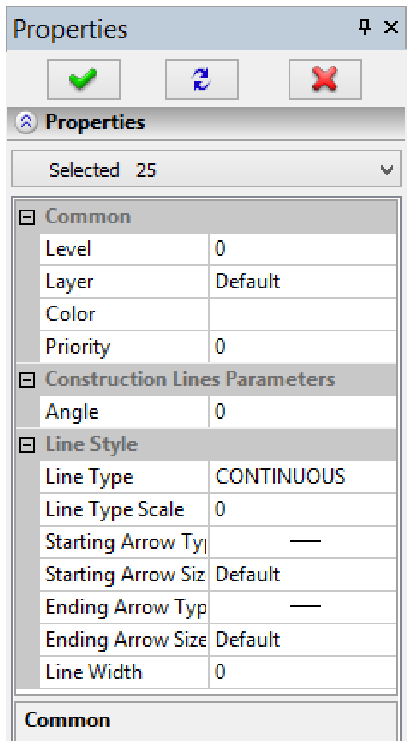
Changing various type element parameters outside any command

Use the properties window to simultaneously change parameters of multiple elements while in the command-waiting mode. In this way, unlike using the specific element editing commands, one can change parameters of various type elements simultaneously.

While in the command-waiting mode, the property window contains a dialog box for changing parameters of the selected element. The dialog is inactive by default. To activate the dialog, enter the properties window and expand the group Parameters. After that, upon selecting any element, the element parameters will be displayed in their property window. To open the dialog box automatically, select elements and call the Parameters command in the context menu.

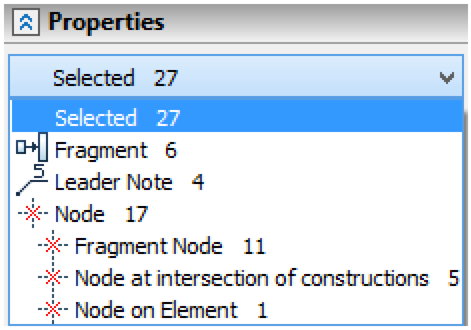
To turn off the active mode of the dialog, close the “Parameters” group.

> Note: upon single element selection, the Parameters command call from the context menu will open the parameters dialog box for the given element.



The properties dialog box for the selected elements consists of two parts. The main part is “Properties”, and the auxiliary one is “Property Sets”.

The main part contains the property table for the elements being edited. By default, all selected elements are subject to editing. The box “Selected” in the upper part of the dialog box displays the number of the selected elements. The list of elements to be edited can be limited to elements of one type by selecting the type in the pull-down list off the mentioned box. In this case, the table will contain only the properties of the selected type elements. The entered changes will also affect the elements of this type only rather than the whole selected group.



By default, the table displays all properties of the elements being edited. Checking the **Only Common** box limits the table contents to the common properties only.

To change properties of the elements, select the desired properties in the table, enter the required values in the cells on the right-hand side, and press the or button in the upper part of the dialog box.

Upon picking the ("End edit") button, the entered changes are applied to the selected elements. The element processing ends and the elements get de-selected at this point.

The ("Apply Changes") button applies the entered changes to the elements as well. However, element processing continues in this case. This button is handy in the cases when various parameters are to be assigned to different element groups within the selected set.

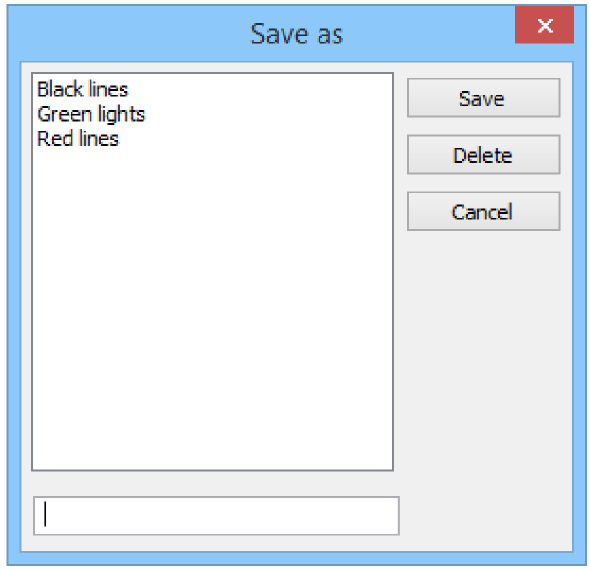
The ("Cancel edit") button can be used to abandon the entered changes and finish processing the selected set of elements. Abandoning changes and finishing the selected element set processing can also be done by simply clicking within the drawing area.

An additional button **[From Element]** allows selecting an element on screen whose properties will be used as current properties of the edited elements. To use this option, first select the properties in the table whose values are to be taken from the element. Then press the button and select with the desired element on the screen. The parameter values will assume those of the selected element.

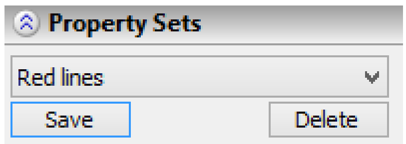
An auxiliary part of the dialog box, the “Property Sets”, allows to save the current set of properties under a specific name for their later reuse.

To save the composed combination of parameters as a set of properties, press the **[Save]** button. A “Save as” dialog box will come up on screen for specifying the name of the new set. All existing named property sets are listed in a box in the upper part of the dialog box. A set can be deleted from the list by selecting with and pressing the **[Delete]** button.

The name of the set to be saved is entered in a box in the bottom part of the dialog box. Upon entering the name, press the **[Save]** button. The “Save as” dialog box will close, and the saved set name will appear in the pull-down list. The **[Cancel]** button closes the window without saving the new set.

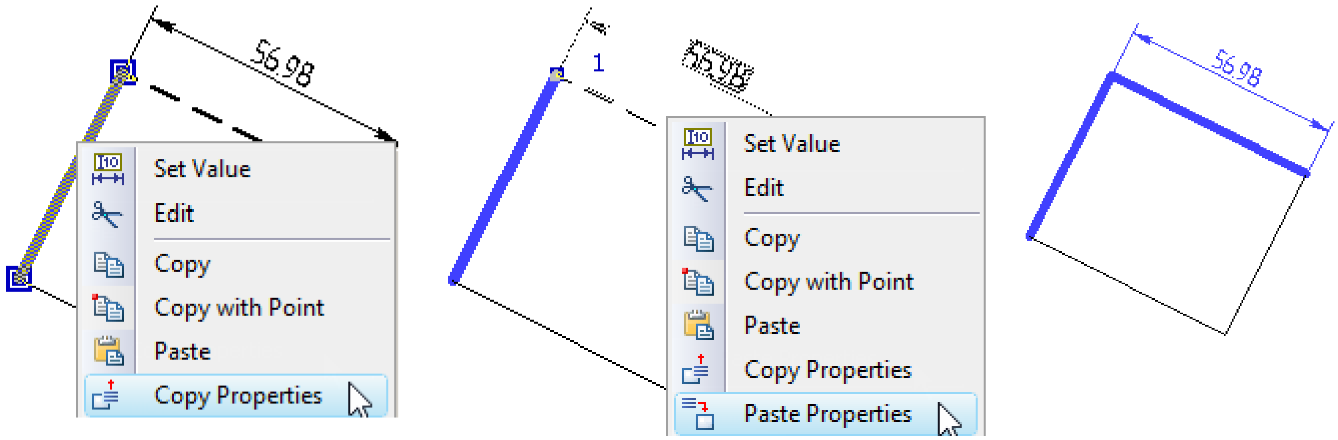


To load a saved set, simply select it in the pull-down list of sets.



Copying element properties through clipboard

In the context menu for any 2D element the command Copy Properties is available. When this command is called, parameters of the selected element are copied into the internal clipboard. After that, upon selecting any other 2D elements the command Paste Properties will be available in the context menu. When this command is called the parameters copied into the clipboard will be applied to selected elements.



Limiting Element Selection. Using Selector and Filter

When working with a dense drawing, it is often difficult to select the desired element on the screen. In this case, it may be necessary to limit the list of the elements available for selection. This can be done in several ways. Some of them, such as using the level and layer mechanisms, were already mentioned in the "Brief Introductory Course" volume. However, these mechanisms either modify the drawing, or allow to temporarily hide construction elements only.

The most general and convenient way that does not require drawing modifications is using the selector and the filter. These tools perform similar functions of limiting selection, however, the selector does this based on element types, while the filter – on the element parameters. Besides, changing selector settings is only available in command-waiting mode, while the filter works in transparent mode. The latter means, the filter settings can be modified at any time, without quitting the current command. The selector and filter settings work independently, adding to each other's function. The elements, whose selection is disallowed by either the selector or the filter, can't be selected on the drawing neither by , nor via the creation and editing command options described above.

Selector

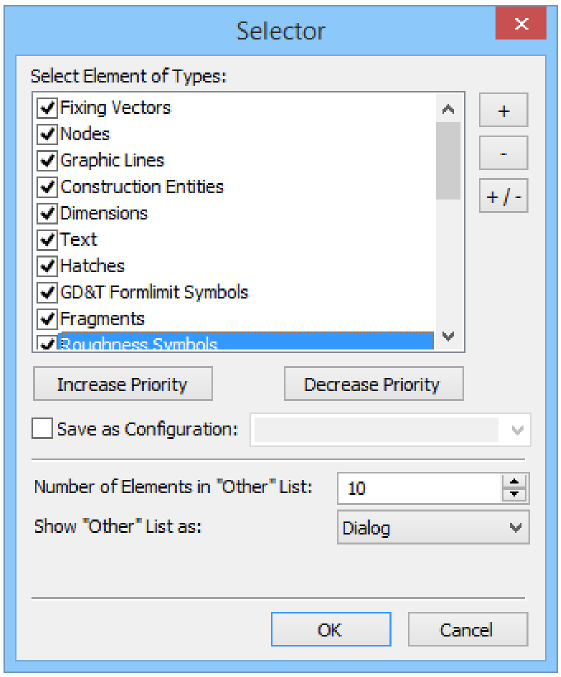
The selector settings are managed by the command FT: Set Selector Configuration. This command can be called only in the command-waiting mode from the toolbar or the textual menu as follows:

Icon	Ribbon
	
Keyboard	Textual Menu
<FT>	Edit > Selector

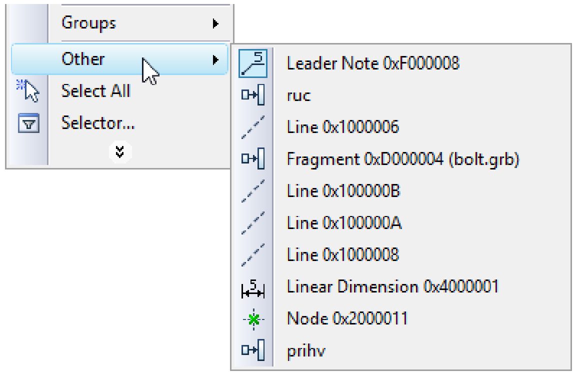
Upon calling the command, the selector configuration dialog box comes up on screen. The main field of this dialog, Select Elements of Types, contains the list of all system element types. The elements allowed for selection are checkmarked at the left of their type names. By default, all elements are allowed for selection. To disallow selection, un-check the respective type with the click.

The buttons +, - and +/- help quickly set, clear and invert checkmarking of the element types.

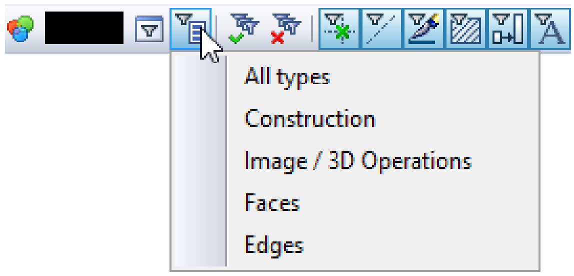
A specified combination of settings can be saved as a named selector configuration. To do so, check the Save as Configuration item and enter the name for the new configuration in the box on the right-hand side.



Additional items in the selector configuration dialog box, such as Number of Elements in 'Other' List and Show 'Other' List as, allow setting different modes of the list display. The list comes up for a selected element upon calling the Other command in the context menu. The effect of these settings was described above, in the "Selecting elements outside any command" topic.



Pressing the **[OK]** button saves the defined settings and closes the command. The **[Cancel]** button closes the dialog box without saving changes.



The selector can later be quickly set up based on a saved configuration. This is done using the button on the system toolbar. Pressing this button brings up a pull-down list containing all available selector configurations. Selecting a configuration in the list automatically sets up the selector per the configuration parameters.

There are several additional buttons on the system toolbar for controlling and quick adjustment of the selector settings.

The and buttons are used to quickly allow/disallow selection of all types of elements.

The buttons with various element type symbols, such as the , , , , , , , and buttons in the 2D window, other in the 3D window, define the current set of the elements allowed for selection. The "pushed" icons correspond to the element types allowed for selection. Besides, one can quickly allow/disallow selection of the respective element types by pressing these buttons. Pressing any of these buttons toggles its setting to opposite. This allows or disallows selection of the respective element type in the selector settings. Pressing any of these buttons while holding the <Ctrl> key down, turns on selection of exclusively the given element type. Selection of other element types simultaneously turns off. The same result can be achieved by double clicking the required button.

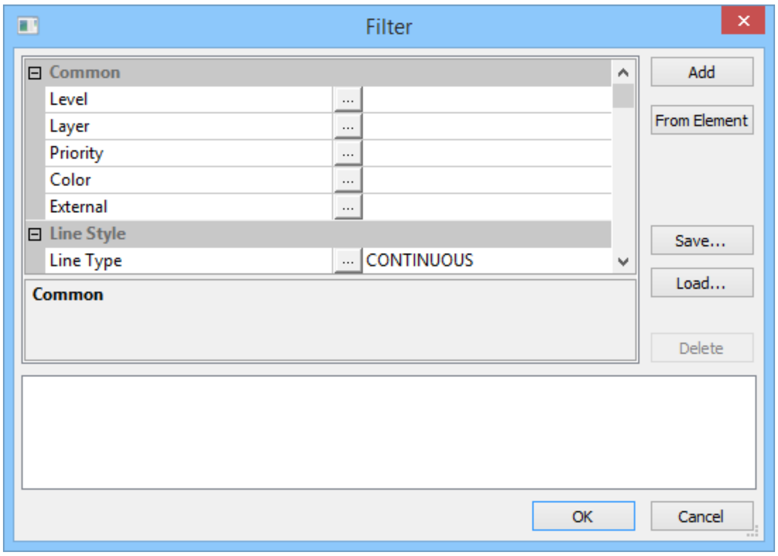
Filter

Filter parameters can be set or modified either in the command-waiting mode or in the transparent mode within any command. Call the command using:

Icon	Ribbon
	
Keyboard	Textual Menu
<FL>	Edit > Filter

Managing the filter involves setting one or more conditions on the parameters of the objects to be selected. The elements are disallowed for selection whose parameters do not satisfy any of the filter conditions. This is so even for elements allowed for selection by the selector.

Calling the command brings up the filter parameters dialog box.

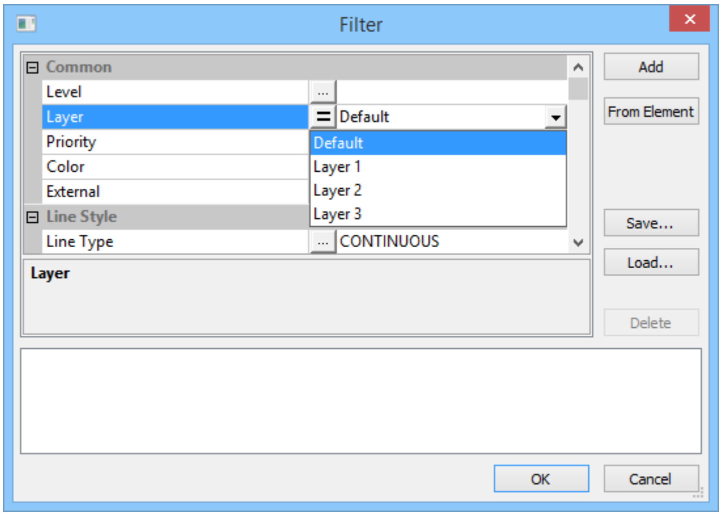


The current filter parameters, which are the currently active condition set, are displayed in the lower part of the dialog. This set consists of one or several conditions joined by Boolean “OR” operator. Thus, an element is allowed for selection if at least one of the conditions is satisfied among the current set.

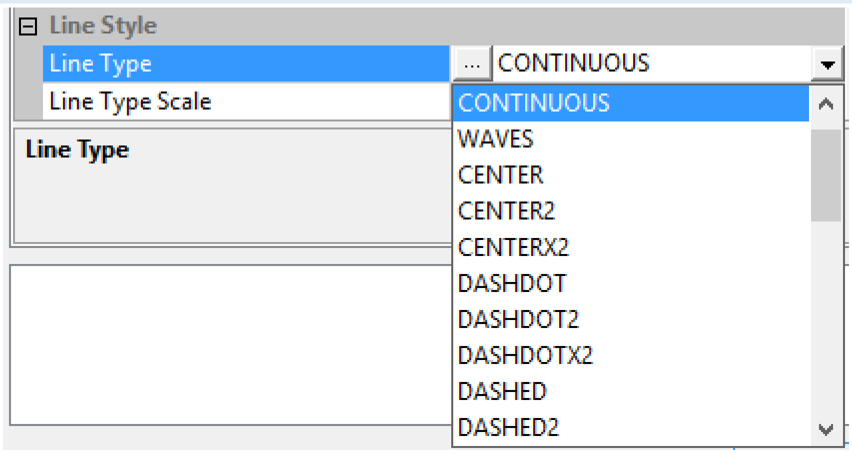
Each condition in a set is written out on a separate line. It consists of limitations on the element parameter values. The limitations are joined in a condition by Boolean “AND” operator. To satisfy a condition, the element must comply with all and any of the limitations thereof.

To create a condition, use the main pane of the filter dialog box. This is a table of properties of all elements in the current document.

To define a limitation on the value of some parameter you need to select the parameter in the table and to press ... button in the middle column. The drop-down list will appear. Here you can select limitation type of the parameter value: Equal, Not Equal, Greater, Less.



The parameter value for the selected limitation is specified in the right column. Numeric and text values are specified manually. If in the system there are values lists for a parameter, you can select a value from the drop-down list. The drop-down list appears automatically when you press on the field.



Once all limitations are defined, press the **[Add]** button. The just created condition will appear in the lower pane of the dialog box. If there was already a set of conditions at the time of the new condition creation, the latter becomes part of this set.

When creating a condition, the parameter values can be read from a specific element. To do so, checkmark the necessary properties, and then press the **[From Element]** button. The dialog box will temporarily disappear from screen, making possible selection of the desired element in the drawing window using . Once an element is selected, the filter parameters dialog box comes back on screen. The checked parameter values will be the same as those of the selected element.

To delete the current condition set or a part thereof, use the **[Delete]** button. To do so, first highlight with one or several conditions. Then press **[Delete]**, and those will be deleted.

A current condition set can be saved under a specified name for further reuse. To do so, use the **[Save…]** button. Upon pressing the button, a "Save as" dialog box will come on screen for saving the condition set.

Specify the name of the set to be saved in the lower pane of the dialog box. After entering the name, press the **[Save]** button. The name can be selected from the list of the existing set names in the upper pane, using . Besides, this dialog allows deleting a previously saved set. To do so, select one in the list and press the **[Delete]** button. The **[Cancel]** button allows to disregard the deletion and quit the dialog.

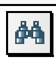
To use an earlier saved condition set, press the **[Load…]** button. After pressing the button, the "Load" dialog box will come up on screen for loading the named condition set. Working with this dialog is similar to the set saving dialog. The upper pane of the dialog contains the list of available sets. Use to select the desired set from the list. The name of the selected set is displayed in the lower pane of the dialog.

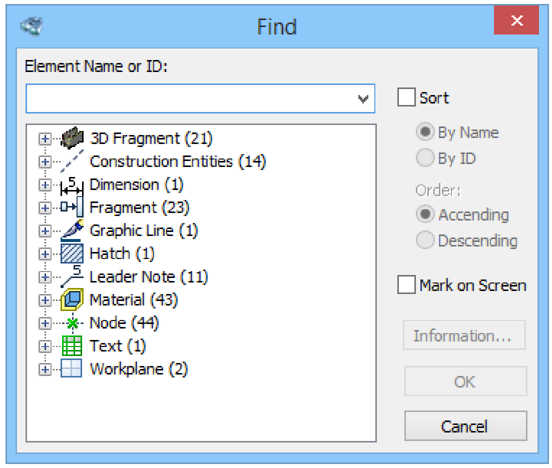
Once selection is done and the **[Load]** button pressed, the dialog closes and the contents of the selected set are added to the list of the existing sets of conditions. This dialog also allows deleting any of the existing named sets using the **[Delete]** button.

The specified set takes effect after closing the filter dialog. Only the elements satisfying the current filter settings will be available for selection in any mode of T-FLEX CAD system.

Element Search

Sometimes, the system might fail to calculate location of some element during regeneration. In such a situation, the system will display an appropriate message with the Id of this element. To find this element on the drawing, one can use the command FD: Find Element:

Icon	Ribbon
	Edit → Additional → Find
Keyboard	Textual Menu
<FD>	Edit > Find



Upon calling the command a dialog box comes up on screen for searching a 2D or a 3D element. An element can be searched by either of the two ways as follows.

One way is to use the input box in the upper part of the dialog. Enter the Id or the name of the searched element. If such element is found, the buttons in the right part of the dialog box will become accessible. Meanwhile, the element may be marked on the screen, depending on the Mark on screen attribute. Pressing the [OK] button closes the dialog window, while highlighting (selecting) the found element on the screen. Pressing the [Information] button opens the element information window. If the element is not found, the buttons remain inaccessible.

A pull-down list of the input box in the upper part contains the previous queries. An Id or name can be selected from this list if desired.

Another way of searching for an element is using the tree in the main pane of the dialog box that contains all model elements. When an element is selected in the tree, the upper input box displays its Id or name. The buttons in the right part of the dialog become accessible as well.

An additional Sort flag serves to sort elements in the tree by the name or by the ID in the desired order (ascending or descending).

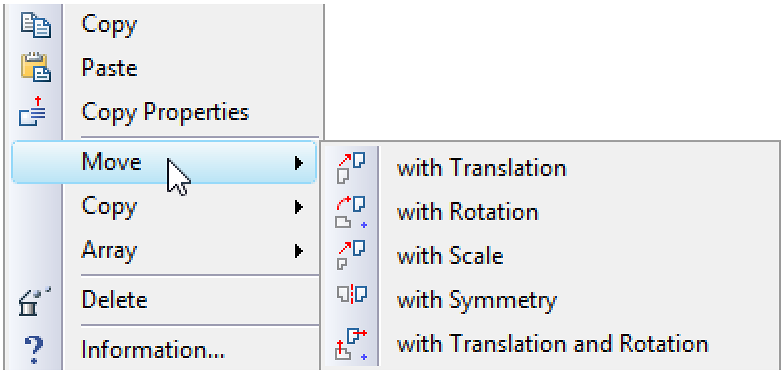
> The search command can be called in transparent mode from within any other command. In this case, the total list will only contain the elements that are allowed for selection in the current command.

Moving, Copying, Transforming Elements. Working with Clipboard

New drawing elements can be created using already existing ones.

> For this purpose, use the general move/copy command. It was described in the chapter “Moving and Copying Drawing Elements. Arrays. Use of clipboard”. This command can be called either from the textual menu and keyboard, or from the context menu for the elements to be transformed.

To call the command from the context menu, select the necessary drawing elements and right-click. The context menu will be containing commands by groups for calling various modes of the move/copy command, specifically, Move, Copy, Array.



The modes under the "Move" group allow changing location and size of the selected elements. Besides, an option is provided for moving all related elements. For example, moving some construction element will be affecting the placement of all related elements to this one, either the construction or the graphic ones.

Meanwhile, all parametric dependencies between elements will stay intact.

The "Copy" group provides the modes for creating a copy of the selected elements (as well as all related ones) at any location of the current document. The created copies can be made associatively related to the original objects, or become independent elements.

The linear and circular array creation modes are provided under the "Array" group. Similar to simple copying, the created result can be either an array with associative relation to the original objects, or a set of independent elements.

T-FLEX CAD also works with the clipboard. Clipboard commands can also be called either from the textual menu, or using the context menu for the selected elements (Copy, Copy with Point, Paste, Paste Special…). Thus, selected elements can be copied into another T-FLEX CAD document or into an external application. One can also insert a picture or text from an external application into a T-FLEX CAD drawing.

Undoing User Actions

Errors unavoidably occur when working with any system, especially while learning. Correcting errors takes time. T-FLEX CAD system helps simplify this process. A certain number of latest user actions are remembered by the system. The length of the undo and redo buffers is set in the command **SO: Set System Options**, on the **Performance** tab, in the **Undo/Redo buffers** box.

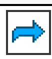
The user actions remembered by the system can be undone by a certain number of steps back. This can be done by repeatedly calling the command UN: Undo Changes, that brings the system back by one step. The UN: Undo Changes command can be called from any other command using <Alt><BackSpace> or <Ctrl><Z> combination.

If the command UN: Undo Changes was called in error, there is the RED: Redo Changes command in the system, which restores the undone action. The RED: Redo Changes command can be called from any other command by <Ctrl><BackSpace> or <Ctrl><Y> combination. Repeatedly calling the command RED: Redo Changes brings the system into the state when undoing began.

The UN: Undo Changes command can be called as follows:

Icon	Ribbon
	
Keyboard	Textual Menu
<UN>, <Alt><BackSpace>, <Ctrl><Z>	Edit > Undo

The **RED: Redo Changes** command is called via:

Icon	Ribbon
	
Keyboard	Textual Menu
<RED>, <Ctrl><BackSpace>, <Ctrl><Y>	Edit > Redo

To cancel or repeat several actions at once, press the button to the right of the icon of the corresponding command. After pressing the button the dropping list of actions which can be canceled or repeated will pop up. Then it is enough just to select the desired group of actions with the help of .

General Principles of Assigning Parameters. Assigning Variables to Parameters

General principles of assigning parameters

Various ways of assigning parameter values are used in element creation and editing commands. These include using parameter dialog box and property window, as follows:

A parameter can be assigned a constant value. For example, the parameter "Rotation angle" of a text can be assigned 0.

A parameter value can be substituted by the string "Default". This means, the parameter value will be set from the respective parameter of the command **ST: Set Document Parameters**. For example, the parameters on the **Font** tab in the parameter dialog box for dimensions, roughnesses and notes will be substituted from the **Font** tab of the command **ST: Set Document Parameters** when the respective elements are displayed.

Using default parameters helps quickly modify elements of the whole drawing. For example, using default parameters for dimensions allows to instantly change dimension display and, therefore, the whole drawing. This can be done by modifying parameters on the **Dimensions** tab of the **ST: Set Document Parameters** command.

The values of most of various element parameters defined by number can be set using string variables and expressions. In this case, the parameter value will be driven by the value of the variable or expression. In this way, the value of the parameter can be changed by varying the respective variable value. This mechanism allows changing any parameters of the following T-FLEX CAD elements: the size of text boxes, the slanting angle, the size of arrows of the dimension leaders and graphic lines, etc. You can use variables to define drawing parameters that are defined in the **ST: Set Document Parameters** command, such as scale, paper size, font size, etc. Variables can also be used for defining the system visibility levels of the elements set in the command **SH: Set Levels**.

Assigning variables to parameters

When assigning a variable to a numeric parameter, enter the variable name or expression without any special symbols. Examples: `A` or `A+B`

When assigning a variable to a string parameter, enter the variable name or expression in braces. Examples: `{$NAME}` or `{A+B}`

When assigning string parameters in braces one can enter either the real variables or textual variables.

If a new variable name was entered when assigning a parameter, the value of this variable must be set after leaving the menu.

When a variable is introduced, the format of its value representation can be specified along. Use the following syntax for typing variable values:

`{<variable name>}` or `{<format>,<variable name>}`

The following example demonstrates use of formatted variable representation.

```
Today {”%lg”,DAY}, {”%s”,$MONTH}, {YEAR}
```

Note that the textual variable $MONTH begins with the ‘$’ character, as this is the prefix for all textual variables.

> The format structure, used for the T-FLEX variables, follows the syntax of the input/output formats in “C” programming language.

Using formats will help you control the appearance of the variable value on screen. For example, formats can control the number of displayed decimal digits or justification of the displayed value.

Context menu for dialog input boxes

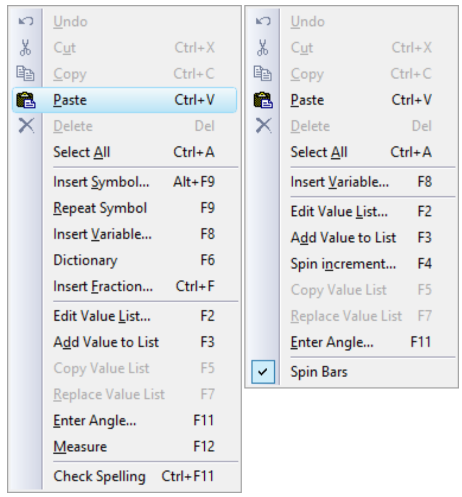
When working with dialog boxes, an additional set of commands is available in context menus. A context menu can be called by placing cursor within an input box of the dialog and right-clicking :

Undo. Undoes the last change.

Cut <Ctrl+X>. Cuts selected text to clipboard.

Copy <Ctrl+C>. Copies selected text to clipboard.

Paste <Ctrl+V>. Pastes text from clipboard.



Delete <Del>. Deletes selected text.

Select All <Ctrl+A>. Selects all text in the current input box.

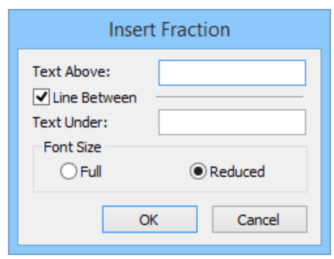
Insert Symbol... <Alt+F9>. Inserts a symbol from a special symbol table. The symbol code is actually entered in the input box instead of the symbol itself, for example, %%066 for the diameter symbol.

This may be used for entering symbols in some textual input boxes. The data from these boxes will be inserted in the drawing. See, for example, the “Text before dimension” input box.

Repeat Symbol <F9>. Inserts last symbol again.

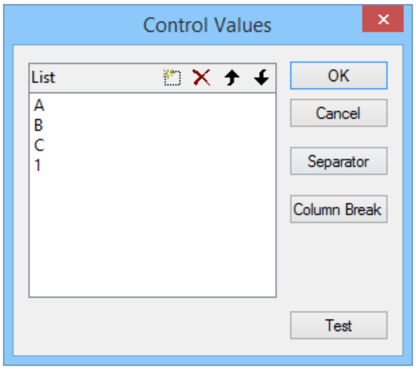
Insert Variable... <F8>. Inserts an existing variable from list. The variable name is inserted in the input box in braces. The drawing will display the actual value of the variable. The variable values can be changed in the variable editor or, in some cases, directly on the drawing (see the section Paragraph text of the Text chapter).

Dictionary <F6>. Inserts text from dictionary. For detailed information, see the topic Working with dictionary of the Text chapter.



Insert Fraction... <Ctrl+F>. Inserting the fraction into the dialog box. Can be used, for instance, for assigning the content of text fields in dimensions, leader notes, text, etc.

Upon calling the command the window of an auxiliary dialog is displayed for setting the parameters of the fraction.



Edit Value List... <F2>. Value lists can be created for the dialog input boxes. The lists are preset for some boxes, for example, the input boxes “Datum” and “Value” in the GD&T Symbol Parameters dialog box. The command brings up a window for editing the values list.

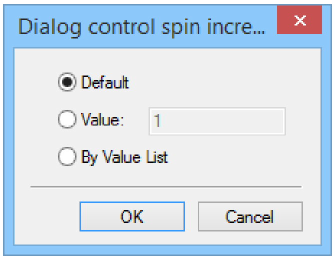
The list can be divided into columns. Entries in a column can be grouped between horizontal dividers.

Insert Value to List <F3>. This command adds the current value from the dialog input box into the list. If the list did not exist, it will be created.

Copy Value List <F5>. This command copies the list of values of the given dialog field into the clipboard.

Replace Value List <F6>. This command replaces the list of values assigned to the given dialog field by the list of values from the clipboard. The list must be copied to the clipboard in advance using the command Copy Value List.

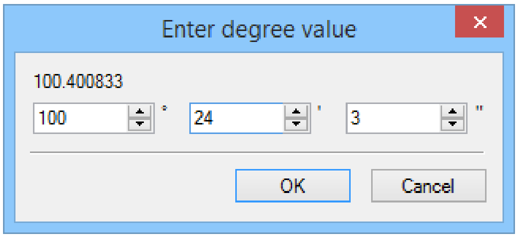
Spin Bars. This command enables the stepper – the way to modify the parameter in the respective field using the mouse wheel or the button .



Spin increment... <F4>. You can define the parameter value increment of the stepper. One of the three settings can be chosen in the spin increment control dialog box: “Default”, “Value”, “By Value List”.

Value. Set a numeric value of the increment.

By Value List. Setting this option will allow to scroll through the list of values in the case the list was created for this input box of the dialog.



Enter Angle... <F11>. This command allows converting an angle value to the decimal format. The command brings up a dialog box. The respective input boxes of the dialog allow entering an angle value in degrees, minutes and seconds. This value will be converted into the decimal format.

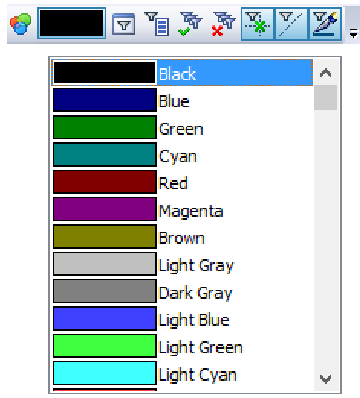
Measure. <F12>. This command allows reading geometric data from existing drawing elements and using it for creating new elements. Parametric dependencies can also be introduced between the elements. For more information, see the chapter “Measuring Elements and Relations between Them”.

Check Spelling. <Ctrl+F11>. Checking the spelling of the content of the dialog field, for which the context menu has been called.

SETTING COMMON PARAMETERS OF SYSTEM ELEMENTS

Each T-FLEX CAD system element, whether a construction or a graphic one, has its own set of parameters that the user can define and modify. In particular, the color, level and layer parameters are present in each set of parameters. Defining and using these parameters will be described here so not to repeat the description for each element.

Color



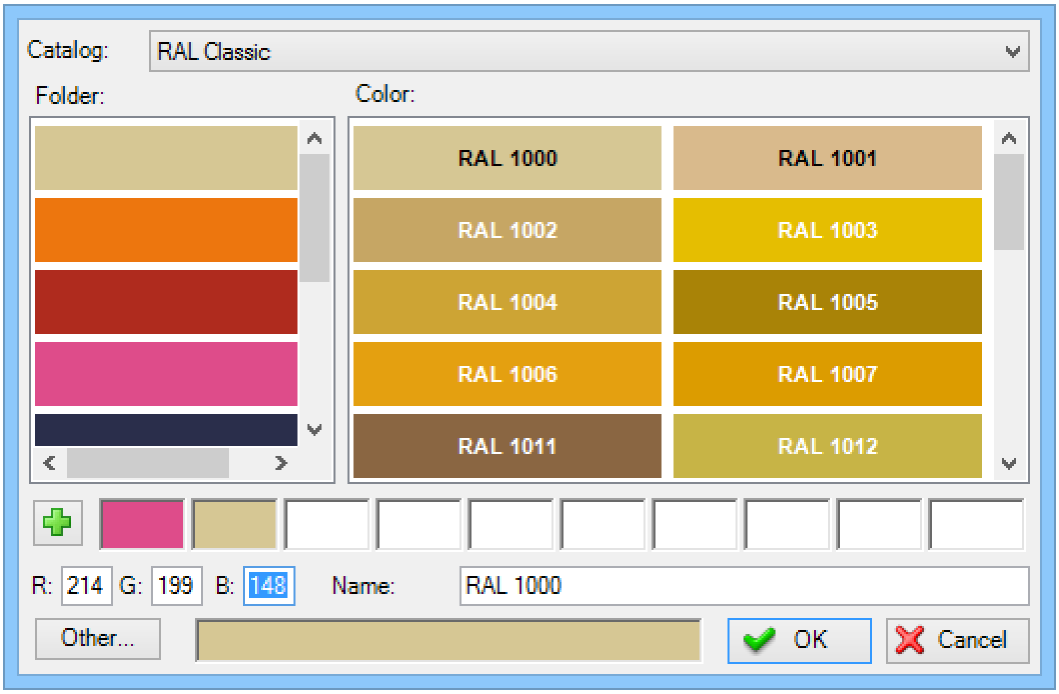
Each element has a color. The parameter dialog includes the input box “Color:”.
This box shows the color used for displaying the given element of the model. The color can be changed by selecting from the list.

An element color can also be set using the system toolbar.

Setting colors via the system toolbar is available in creation and editing commands.

You can select color from the catalog of colors. Use special button  next to the drop-down list of standard colors in the properties dialog to open appropriate color selection dialog.

Also you may use appropriate button  in the system toolbar to open convenient color selection dialog.



Color catalog contains a set of pages (the list on the left) containing colors. Color is defined as RGB components and name.

Besides the list of colors, the dialog contains 10 selected colors, which user may add using them to have for faster and direct access.

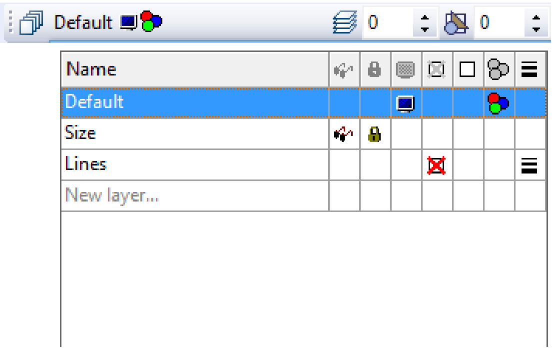
Each color catalog is stored in a separate file with the .acb extension. To edit a color catalog you can use any text editor, or one of the editors of available for working with .acb file format. By default, available catalogs of colors are located in system folder Program/ColorBooks. You can change the path for searching catalog of colors at "Folders" tab of "Customize | Options" command dialog.

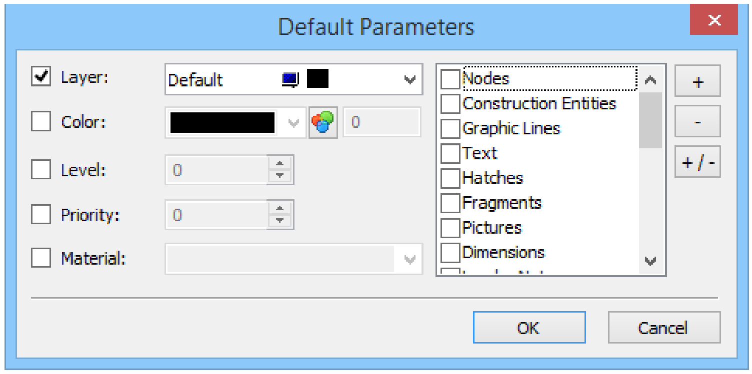
Layer

A layer is a parameter of any drawing element. It defines the element association with a particular group of the model elements.

The user can define the layer name for each system element to belong to. A layer name is a string of up to 20 text characters.

An element layer can also be set on the system toolbar.





Drop-down part of the control is resizable. The list can be sorted according to various parameters of layers. Clicking an icon can modify layer parameter without leaving the current command. The list has item "New Layer ...» which allows you to create a new layer without exiting the command and to make it active.

Layer parameters can be created, deleted and modified using the command **QL: Configure Layers**:

Icon	Ribbon
	3D Model → Style → Layers
Keyboard	Textual Menu
<QL>	Customize > Layers

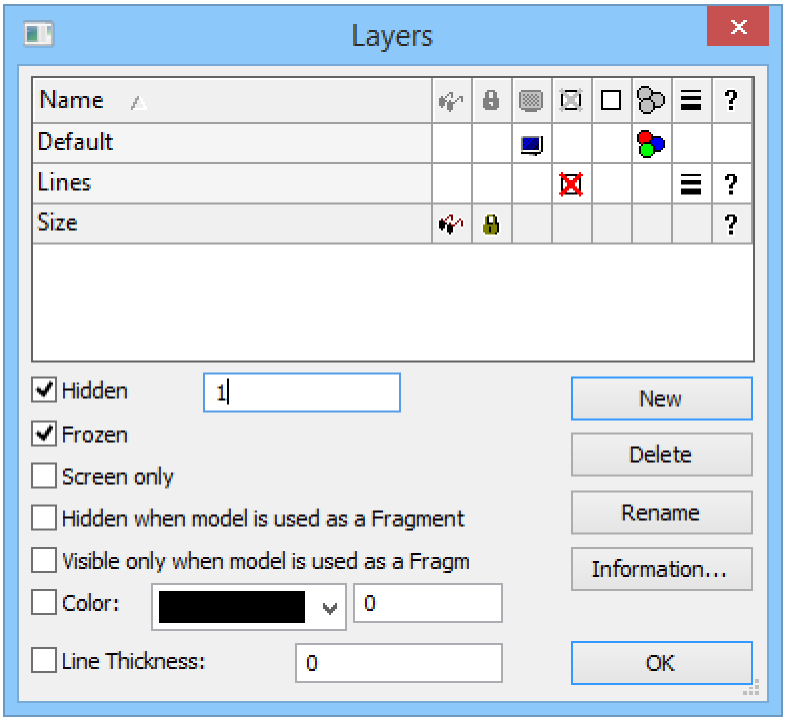
After calling this command the dialog window Layers appears. In the window of the given dialog box the list of the layers existing in the given document and their parameters are shown. Under this list there are fields for assigning parameters of the layer and buttons for performing different actions with the fields.

The button **[New]** creates the new layer in the document. After pressing this button the system asks to give a name to the created layer.

The button **[Delete]** removes unused layer (it becomes available only upon selecting from the list the layer marked with the sign **?**).

The button **[Rename]** allows assigning the new name for the layer selected from the layers list.

The [Information] button allows to display the list of objects located on the selected layer.



For changing parameters of any layer it is necessary to select it from the list of the layers and set on/off the required flags under the list. You can also set parameters by clicking on the corresponding icon field.

By entering layer parameters you define the properties of the elements belonging to this layer. You can select several layers simultaneously and set parameters for them all.

The following parameters can be defined for each layer:

Hidden. A layer can also be assigned invisible property by using a variable. The variable can have two values: 0 – the layer is visible, and 1 – the layer is invisible.

> The variable values different from 0 and 1, are processed by the system as follows: the fractional part is dropped, and the resulting number is matched with 0. If matching, the layer will be visible, otherwise – invisible.

Frozen. When set, no element on this layer will be allowed for selection during element creation and editing.

Screen only. When set, all elements on this layer will be displayed on the screen only, but will not be printed, plotted or exported.

Hidden when model is used as a Fragment**.** When set, the elements on this layer will not be displayed when the drawing is used as a fragment.

Visible only when model is used as a Fragment. When set, the elements on this layer will only be displayed when the drawing is used as a fragment of an assembly.

Color. When set, all elements on this layer will be displayed in the specified color after the redraw. The color is selected from the color menu.

Line thickness. Upon enabling this flag, the same thickness will be set for all graphic lines in the given layer.

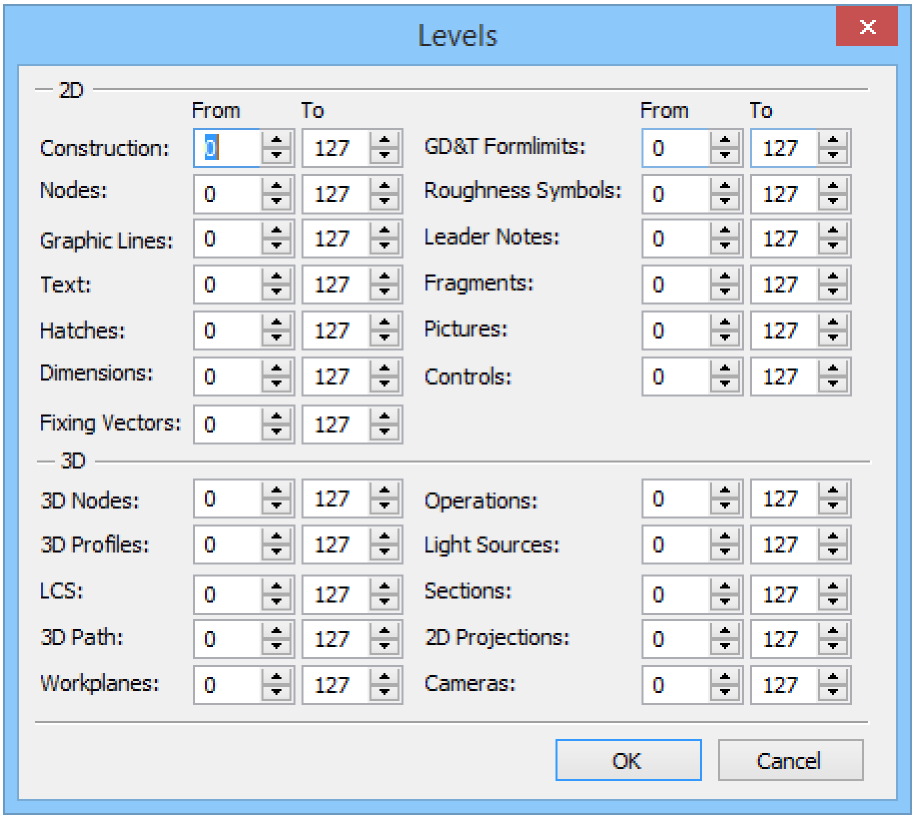
Level

Each model element is assigned a level. The level of an element is an integer. It defines whether the element will be displayed on screen after the redraw. In other words, it defines the element visibility.

The level value can be within the range from -126 to 127. Each element level is connected with the system element visibility range that is set in the command SH: Set Levels:

Icon	Ribbon
	3D → Style → Levels
Keyboard	Textual Menu
<SH>	Customize > Levels

After calling the command, a dialog box comes up for specifying the ranges of element levels.



The level visibility range is defined by two numbers in the range from -126 to 127 for each element type. An element visibility upon redraw is defined in the following way:

If the element level value is within the range defined for this type of elements, then the element will be displayed upon redraw.

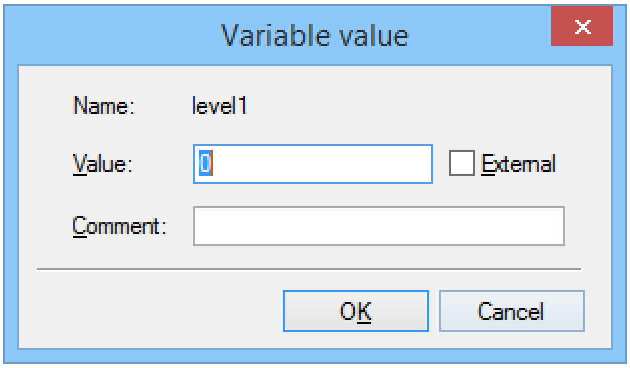
If the element level value is outside the range defined for this type of elements, then the element will not be displayed upon redraw.

An element level can be defined by a constant, variable or expression.

> Advanced usage of element levels in a drawing requires knowledge of working with variables and the command V: Edit Variables. Therefore, continue studying level setting after gaining the required knowledge.

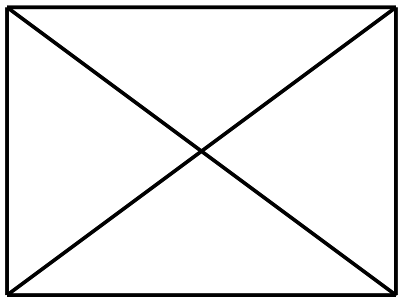
When using a variable for defining a level, enter the variable without braces, for example, *LEVEL1*.

After exiting the parameters dialog box of the given element, another dialog box will come up on screen for setting the value of the variable *LEVEL1*.

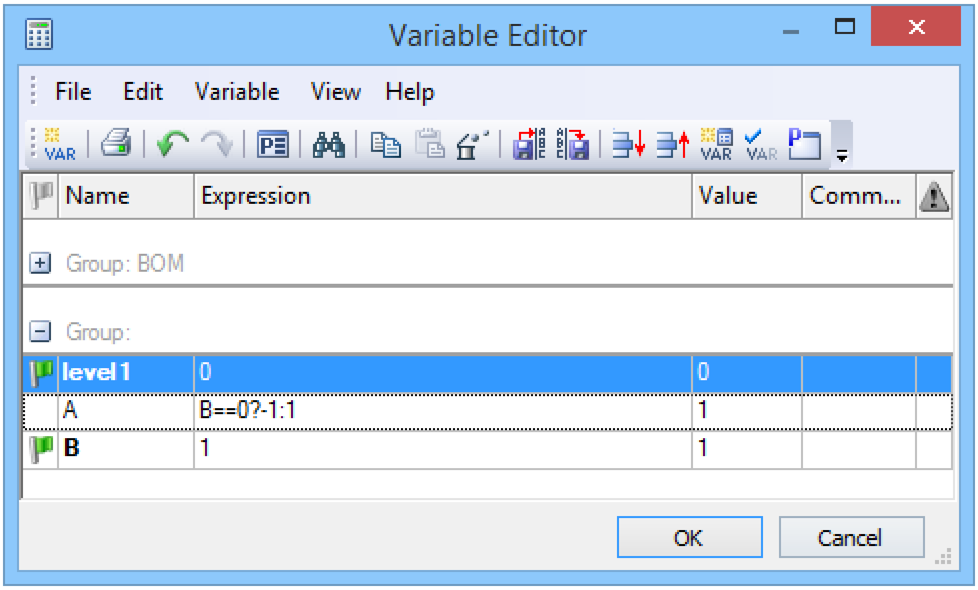


Using a variable as an element level allows modifying the way in which the drawing is displayed depending on specific conditions.

As an example, create a drawing shown on the following diagram.

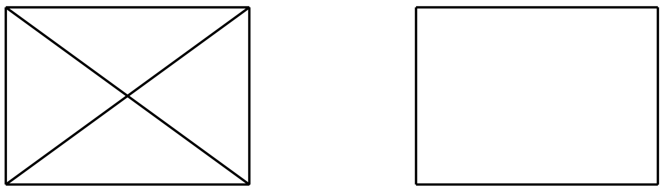


Set the level of the rectangle diagonals using the variable *"A"*. Set the value of the *"A" variable* equal to *"1"*. In the command SH: Set Levels set the visibility range for the graphic lines from *0* to *127*. In the variable editor create a variable *"B"* with the initial value *"1"*. Enter the following expression in the variable editor for *"A"*: *"B == 0?-1:1"*.



Thereafter, set the value of *"B"* first equal to *"1"*, and then *"0"*.

With the first value, the created line will be present on screen, while absent with the second value.



Thus, using variables as levels of various elements, you can create different variations of the same drawing.

Priority

When creating assemblies, especially, in engineering industry, it is often necessary that one element be drawn on top of others. This behavior is easy to realize using parametric fragments, hidden line removal, and an additional special parameter of graphic elements – the priority.

The fact is, the model elements are drawn on the screen or other graphical devices in a certain order. This order normally corresponds with the element types and the order of element creation. However, this order can be changed using priorities.

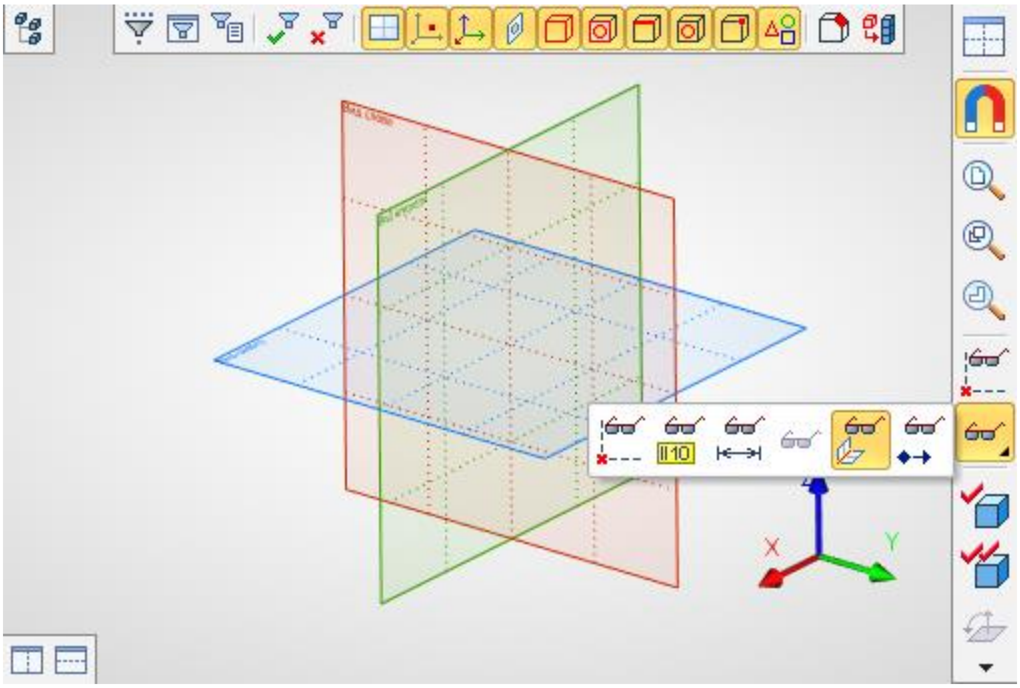
A priority, just like a visibility level of an element, is an integer from -126 to 127, which can be specified by a variable value or an expression. The order of drawing elements follows the rule: elements with lower priority are drawn before elements with higher priority. Therefore, an element with a high priority "obstructs" the elements drawn earlier. For fully benefiting from the hidden line removal functionality, the system provides a special attribute of the hatch contour: "Use for hidden line removal". When this attribute is turned on, the area of the hatch is filled with the background color. Therefore, using priorities and special hatches allows creating assemblies using overlays.

An example of using hidden line removal could be any assembly of co-axial parts, created from fragments. In this case, the fragment parts are created without hidden line removal required in the assembly. Simply set their correct priorities when assembling.

Using this method helps significantly speed up the process of creating assembly models and minimizes the necessity for editing elements when modifying the assembly model parameters.

Controlling Element Visibility

Additional tools for controlling element visibility on the drawing are provided by the commands SI: Hide Construction, Show Relations, SN: Hide 3D Annotations и ESO: Hide/Show Elements. These commands are available at the instrument toolbar “View” and in the menu “View”.



Command SI: Hide Construction:

Icon	Ribbon
	
Keyboard	**Textual Menu**
<SI>, <Ctrl><Shift><C>	View > Hide construction

The command hides all construction elements in the current window (the 2D view or the 3D view). A second call to the command restores the construction element display on the screen.

Command Show Relations:

Keyboard	Textual Menu	Icon
<->	-	

This command enables to hide temporarily all relations (see the chapter "Relations"), created in the current 2D window. The repeated call of the command restores the relations.

Command SN: Hide 3D Annotations:

Icon	Ribbon
Keyboard	Textual Menu
<SN>	View > Hide 3D Annotations

This command is available only for 3D version of the system. It enables to hide all 3D annotations (3D dimensions, notes etc.) in the current 3D window.

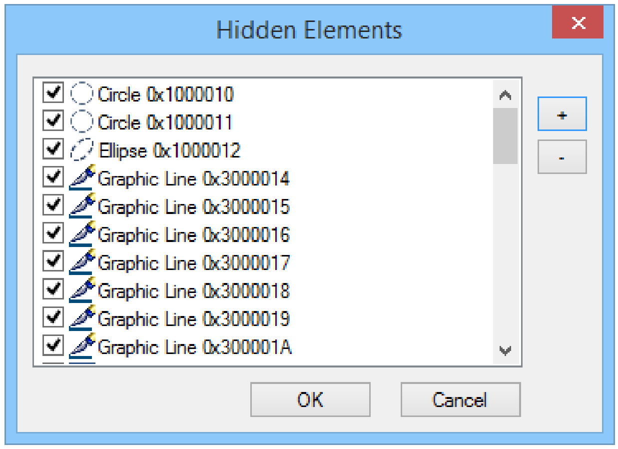
Command ESO: Hide/Show Elements:

Keyboard	Textual Menu	Icon
<ESO>	-	

This command controls visibility of particular drawing elements. The command automenu contains the following icons:

	<S>	Show Element types possible to select
	<L>	Show hidden Element list
	<*>	Show all hidden Elements
	<Esc>	Exit command

The option calls the selector dialog box defining the list of elements allowed for selection within the current command. The selector settings made within a command do not affect the settings made via the FT: Set Selector Configuration command. Upon entering a command, the selector default settings allow selection of all elements.



To hide an element, simply click it with . This hides the element on screen, making it a hidden element of the drawing. Hidden elements are assigned the "Hidden" attribute by the system. These are not displayed on screen but can be selected in 2D element creation and editing commands.

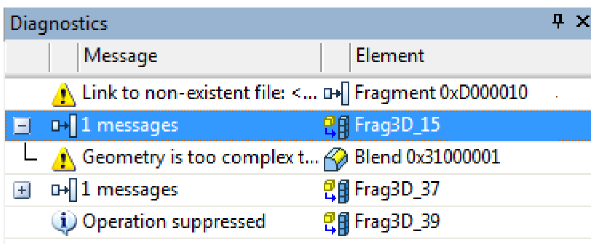
The option brings up a window with the list of all hidden elements.

To restore visibility of an element, uncheck the box before the element name. The graphic buttons "+", "-" clear/set checkmarks for all elements in the list.

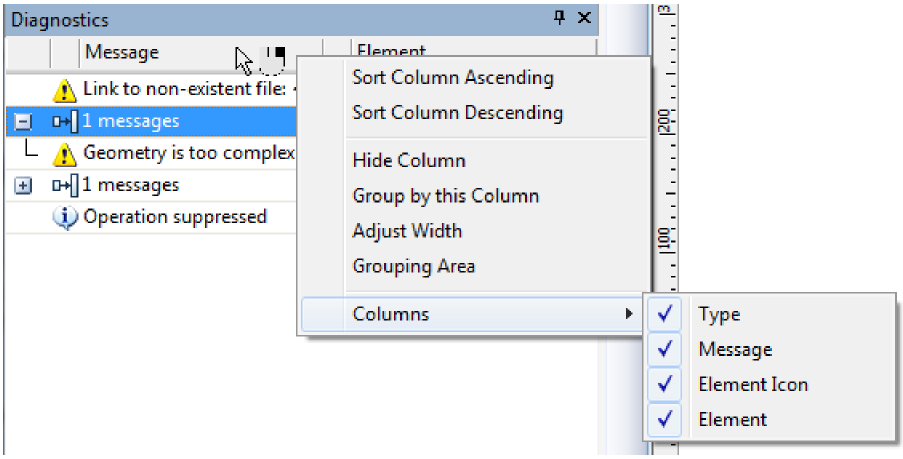
Calling the option makes all hidden elements visible.

DIAGNOSTICS WINDOW

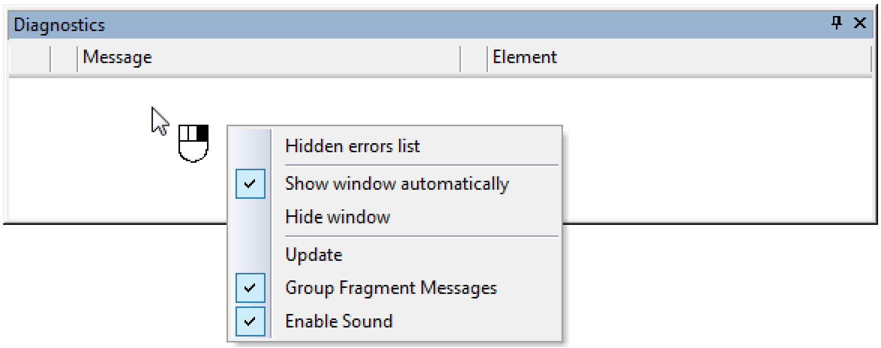
Messages about errors and various warnings of the system are shown in the special service window – diagnostics window. In addition to the cause of occurrence of the error, the system also shows information about the "faulty" element. The diagnostics window is created independently for each document opened in T-FLEX CAD application.



The appearance of the table of messages in the diagnostics window is determined by the user. It is possible to select the columns which will be displayed in the table, modify the rules for sorting and grouping the messages. All these actions are carried out with the help of the context menu of the bar of headers of the diagnostic window.



The context menu that is invoked *inside* the table of messages will have another view. If the diagnostics window is empty (no errors), only the service commands will be accessible in the context menu:



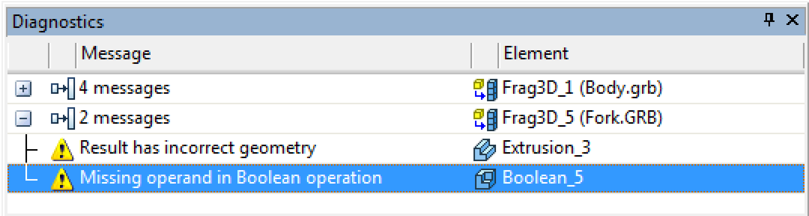
Hidden Errors List. This command invokes the dialog window with a list of hidden messages. To cancel the locking of messages of a certain type, it is sufficient to clear the flag to the right of the message.

Show Window Automatically. If this flag is enabled, the diagnostics window will automatically open when the new messages arise. This mode can be useful when combining several service windows (including diagnostics windows) into the one common window with tabs.

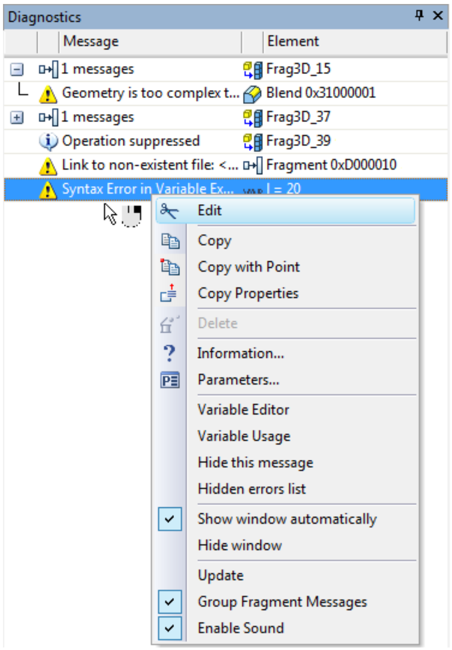
Hide Window. This command closes the diagnostics window. The user can again open the window with the Customize > Tool Windows > Diagnostics Window command.

Enable Sound. This command turns on/off the sound signal when the error arises.

Group Fragment Messages. This flag of the context menu controls the grouping of messages that come from one fragment. When the flag is enabled, the messages from different fragments are grouped by folders.



The context menu that is invoked upon pressing on the error message also contains the set of standard commands of editing a faulty element of a 3D model or a 2D drawing. For example, it is possible directly from the diagnostics window to launch the Change command for a faulty element.



In the context menu that appears upon selecting one error message, the following command is also accessible:

Hide this message. This command allows us to hide the message in the diagnostics window. When invoking this command, the dialog window with additional query appears: *"Hide Message only For Selected Element? Yes/No/Cancel"*. Depending on the answer of the user the following variants of the command's action are possible:

Yes. All messages of the selected type for the same object (2D or 3D element) will be suppressed (2D or 3D element);

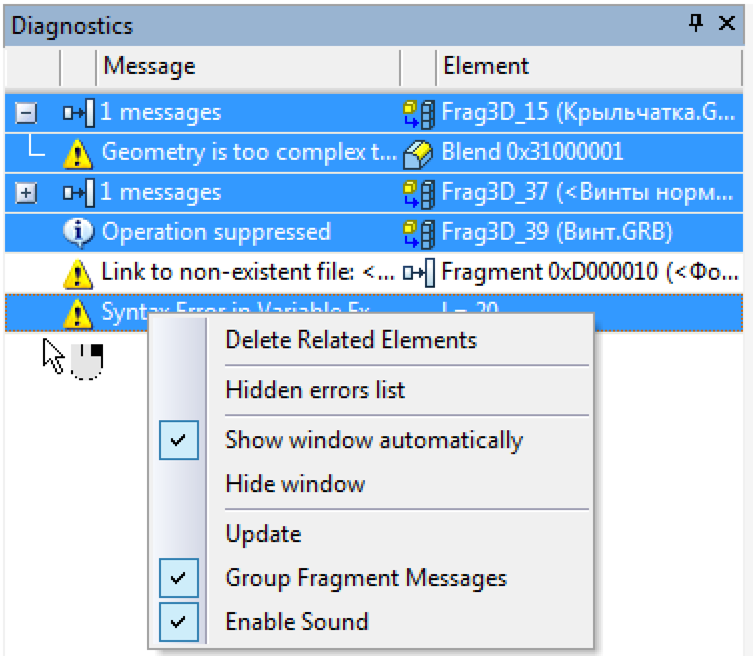
No. All messages of the selected type for all objects will be suppressed (in the framework of the current T-FLEX CAD document).

Cancel. The execution of the command is canceled.

Hidden messages are put into the list of hidden messages. They will not be displayed in the diagnostics window until the mode of locking is canceled. The action of this command is also extended to the subsequent sessions of work with the given file/system. It is possible to view the list of hidden messages of the current document and, if necessary, edit it with the help of the Hidden Errors List command.

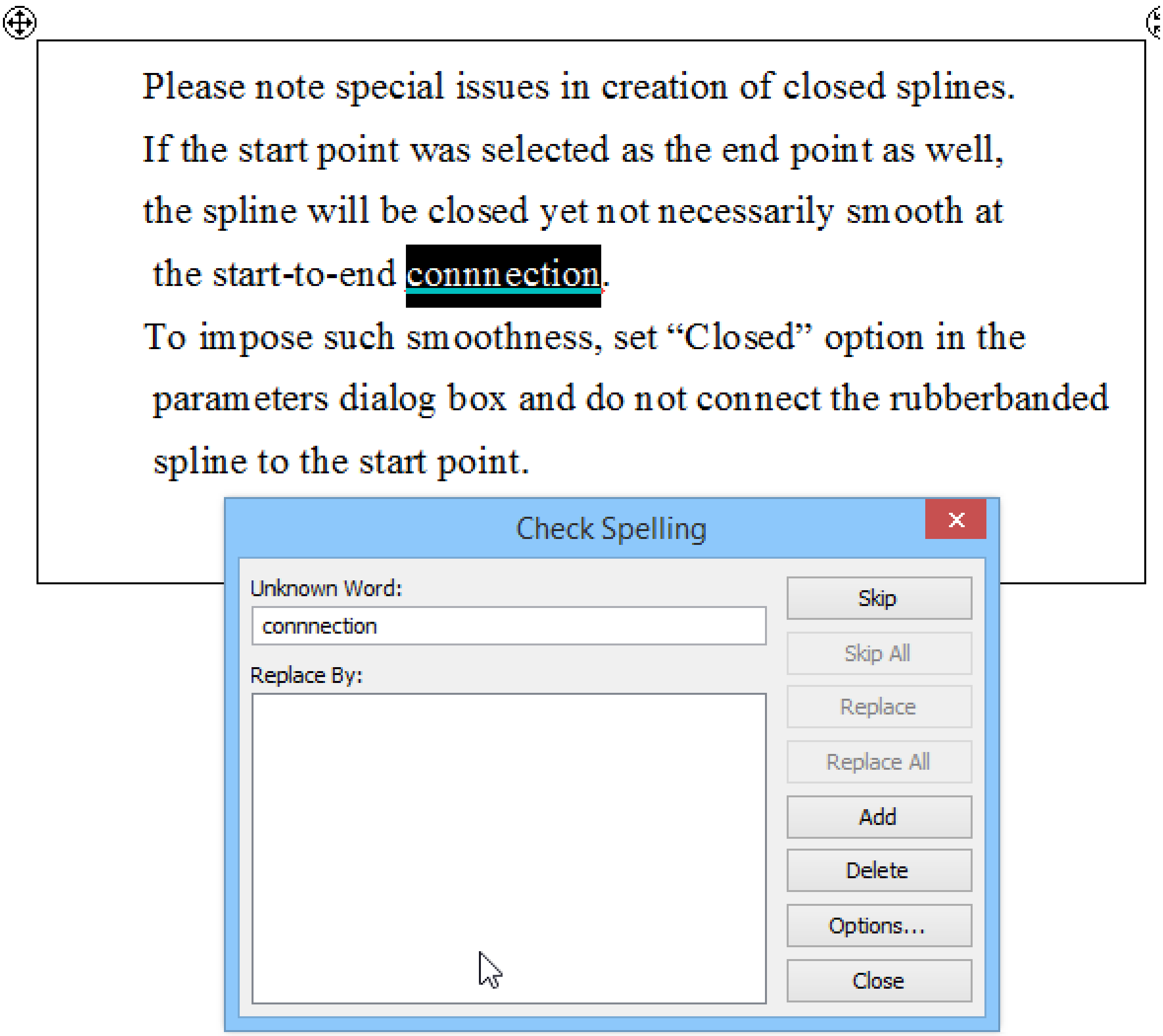
In the diagnostics window it is possible to select several messages simultaneously by using <Ctrl>+. When selecting several messages the following command will appear in the context menu:

Delete linked elements. This command allows us to delete all 2D and 3D elements linked with the selected messages.



CHECKING SPELLING FOR DRAWING

T-FLEX CAD allows checking the spelling of any texts in the drawing. Checking is carried out by tools of Microsoft Word.



To check the spelling of texts in the drawing the following command should be called:

Icon	Ribbon
	Tools → Tools →Check Spelling
Keyboard	Textual Menu
<Ctrl><F11>	Tools > Check Spelling

After calling this command it is necessary to indicate the text, which needs to be checked, with the . The command enables to select and check the spelling of several texts simultaneously on the current page of the drawing. With the help of the following option all texts in the drawing can be selected:

	<*>	Select all Elements

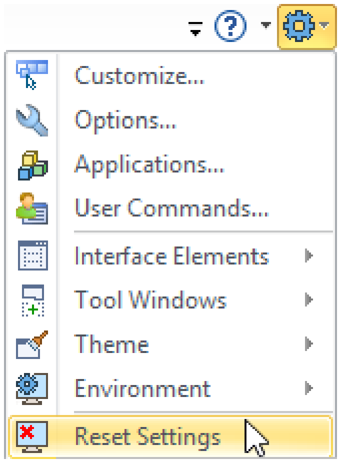
While the spelling is being checked it is possible to move from one checked word to another with the help of or buttons in the window of command's properties.

The command for checking the spelling of texts can also be called from the context menu.

CUSTOMIZING SYSTEM

T-FLEX CAD provides a wide range of system customization capabilities. You can set color preferences, customize dialog boxes appearance, define function key combinations for quick command access, set tool windows location on the screen. To define this kind of parameters, a customization group of commands is provided.

The information about the application settings is stored in the system registry. Different application settings for different users are supported on the same computer. To restore default system settings, use the item **Reset Settings**. The drop-down list is in the upper right corner.



SETTING OPTIONS. DIALOG OF "SET SYSTEM OPTIONS" COMMAND

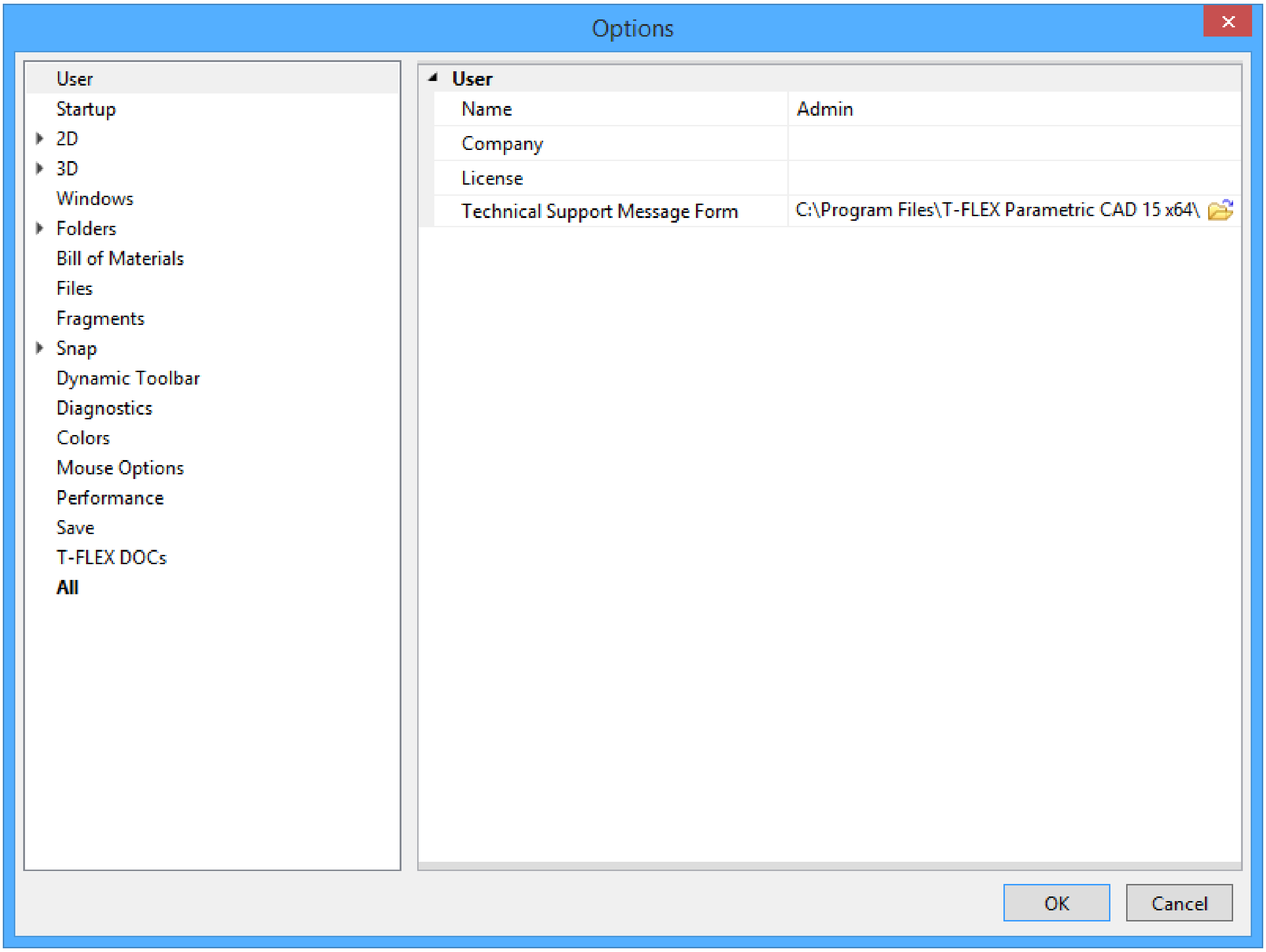
The command for customizing the system is **SO: Set System Options:**

Icon	Ribbon
	→ Options
Keyboard	Textual Menu
<SO>	Customize > Options

The command brings up a dialog box with various groups of parameters available on the respective tabs.

When you select any parameter its tip appears in the bottom part of the screen. Such tips are available for the most of parameters.

"User" Tab



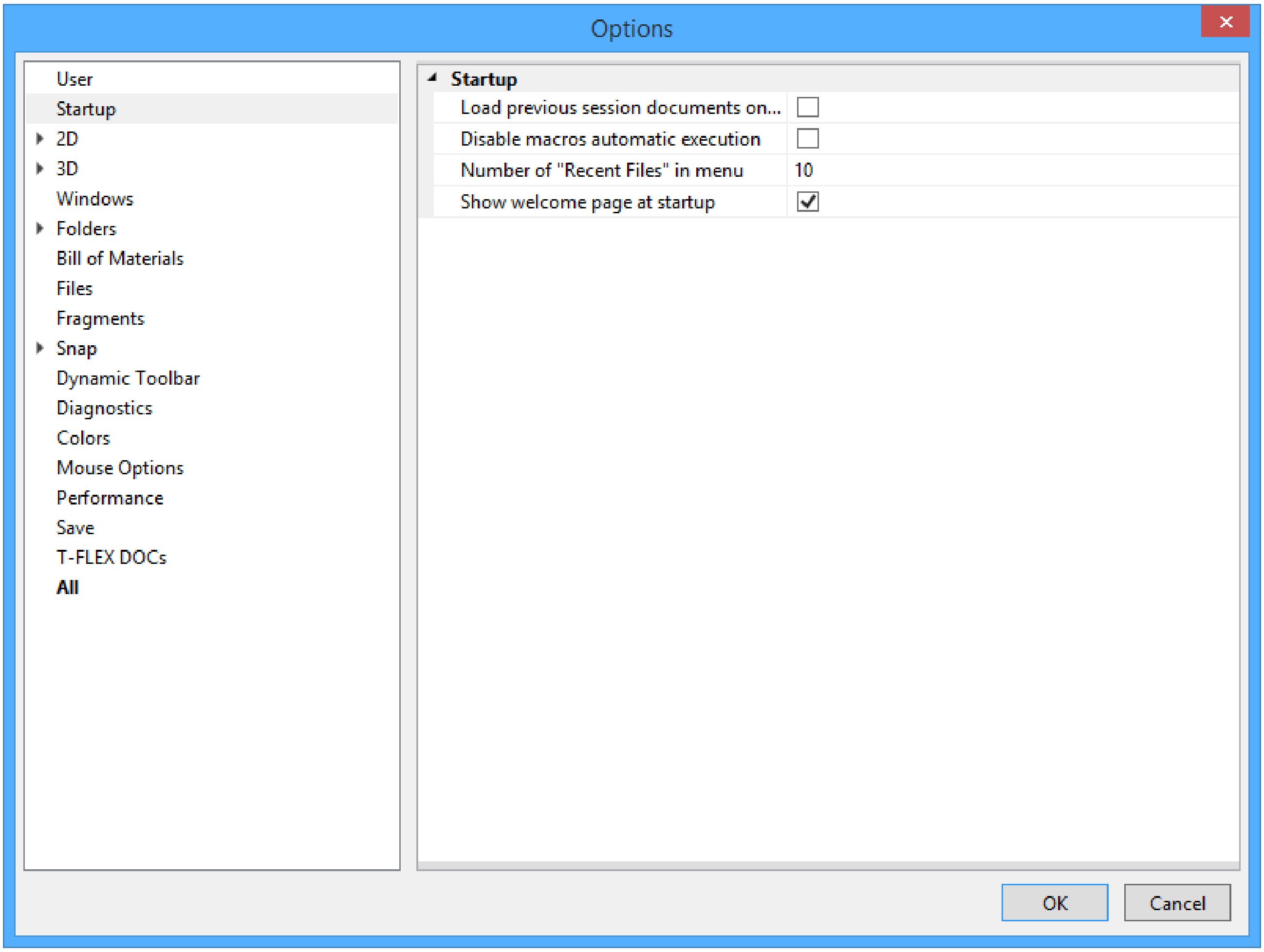
This tab of the dialog specifies the user name, the name of the company using the system and its license number.

The user **name** and the **company** name are saved in each newly created document file. Additionally, these parameters, along with the license number, are automatically included in the text of the message to the technical support group generated by the command **Help** > **Technical support...**

License. License number automatically included in the text of the message to the technical support group generated by the command **?** > **Technical Support....**

A message to the technical support group is based on a template whose location is defined by the parameter **Technical Support Message Form**.

“Startup” Tab



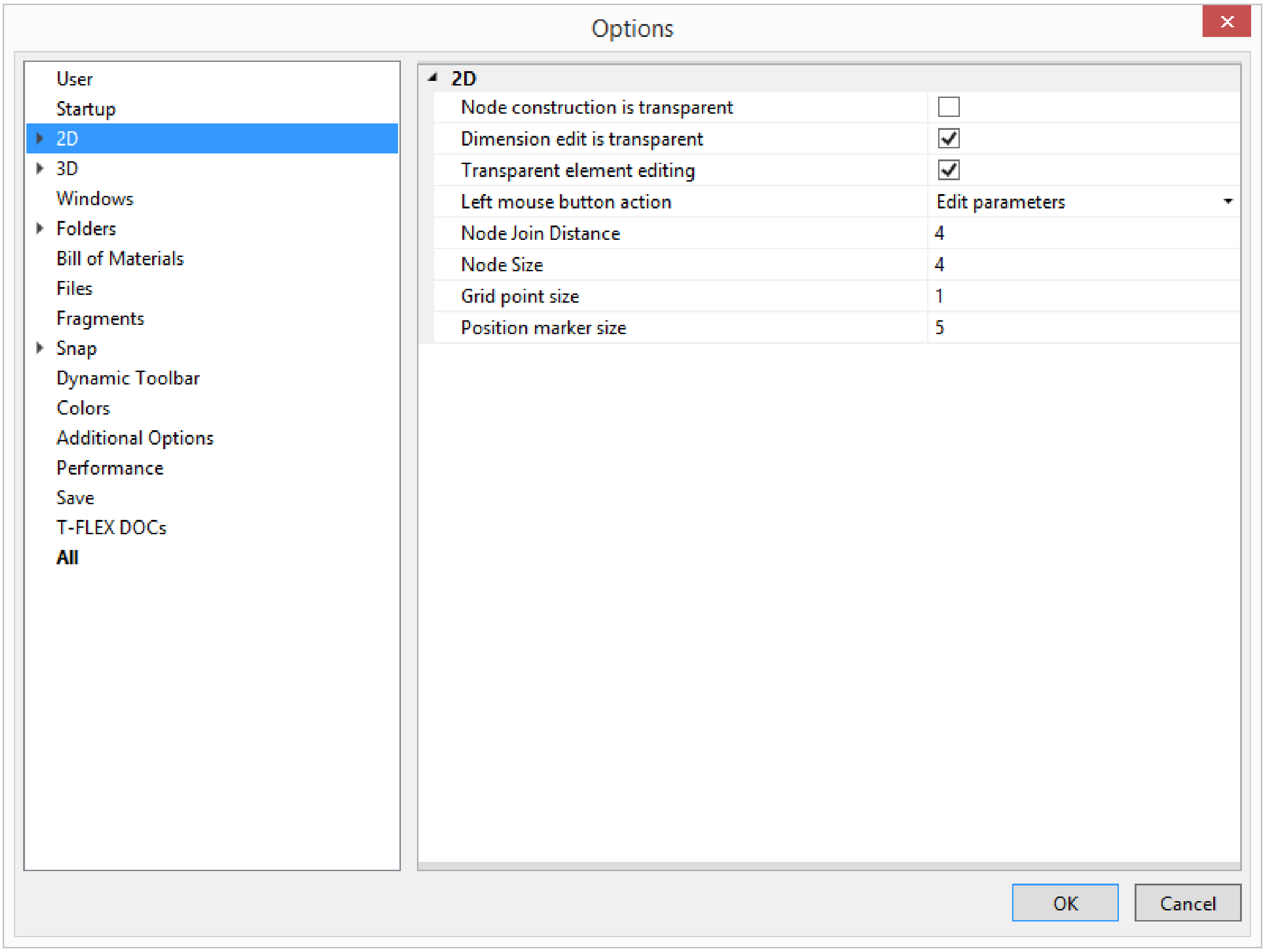
Load Previous Session Document on Start. If this flag is set, then upon the next start of T-FLEX CAD the documents will be automatically loaded that were open at the time of closing the previous system session.

Disable macros automatic execution. Allows to disable automatic macros execution, appointed to document events (opening, closing, saving and so on). This mode can be useful when user is debugging macros.

Number of “Recent Files” in menu. Defines the number of items in the list of files opened in recent sessions. The number should not be greater than 16. This setting affects the menu **File** > **Recent Files** and the **Start Page** dialog box.

Show welcome page at startup. Defines whether to launch the “Start Page” dialog box on the application startup.

“2D” Tab

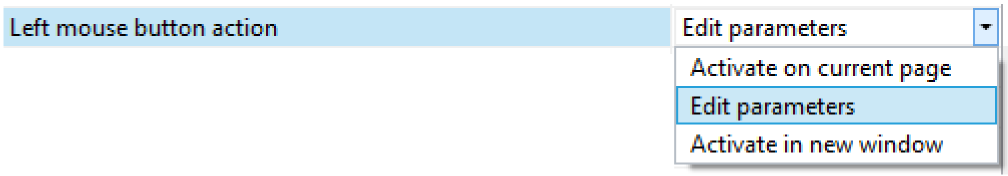


Node construction is transparent. Setting this parameter allows calling the node creation command from within any other command. To do so, type “**N**”. This will not abort the current command.

Dimension edit is transparent. This option toggles on/off the transparent mode of the command **Parameters > Dimensions**. The latter command allows selecting dimensions on the drawing and editing their nominal values. The construction entities that are driven by the dimension are being identified and relocated (if possible) according to the new value.

“Transparent“ Element Editing. When this parameter is turned on, upon selection of 2D elements in the command waiting mode with the help of , the command of editing a selected element is automatically started. If this parameter is turned off, nothing is happening after choosing a 2D element, the system just waits for the user's commands. This parameter is turned on by default.

Left mouse button action. This parameter defines the action performed for drawing views on the left mouse button push while in commands. The action is selected from the pull-down list.



At your choice, pressing in different modes will either bring up a context menu or cancel the current command. For example, if the entry **Menu in command...** is selected, then the context menu on pressing the right button will be duplicating the automenu while in a command.

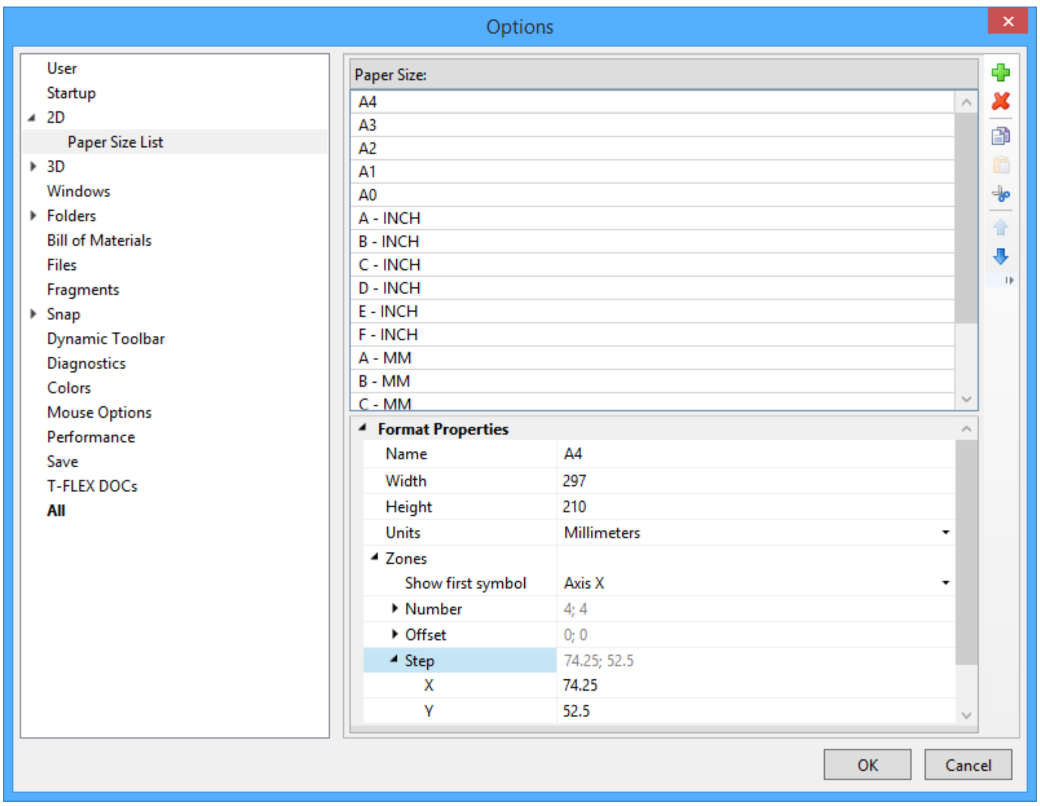
Node join distance. Sets the radius in pixels for locating nodes on the screen. The join distance radius is used when creating new nodes in the "free drawing" mode. If the cursor is within the join distance of some node then this node will be selected instead of creating a new node.

Node size. Sets the size in pixels of a node on the screen.

Grid point size. Sets the size of grid points on the page in pixels. Grid on the page can be customized by **Edit>Grid** command.

Position marker size. Specifies the size of position marker. The position marker specifies position of the dimension and value on it upon editing.

"Paper size list" Tab



Paper Sizes. In the section you can edit the list of drawing paper sizes. This list can be edited in the drawing customization functionality (the **Document Parameters** dialog box, **Paper** tab under the **ST: Set Document Parameters** command).

In the section you can set **Name**, **Width** and **Height** of a general or custom format. It also defines the measurement Units of the T-FLEX CAD system.

Besides, one can set the parameters for dividing the drawing into **Zones**:

Step. Defines the X and Y dimensions of one zone.

Offset. Defines the X and Y offsets of the area being divided into zones with respect to the point (0,0).

First char, **X** and **Y**. Define the characters to begin with when itemizing the zone columns and rows respectively.

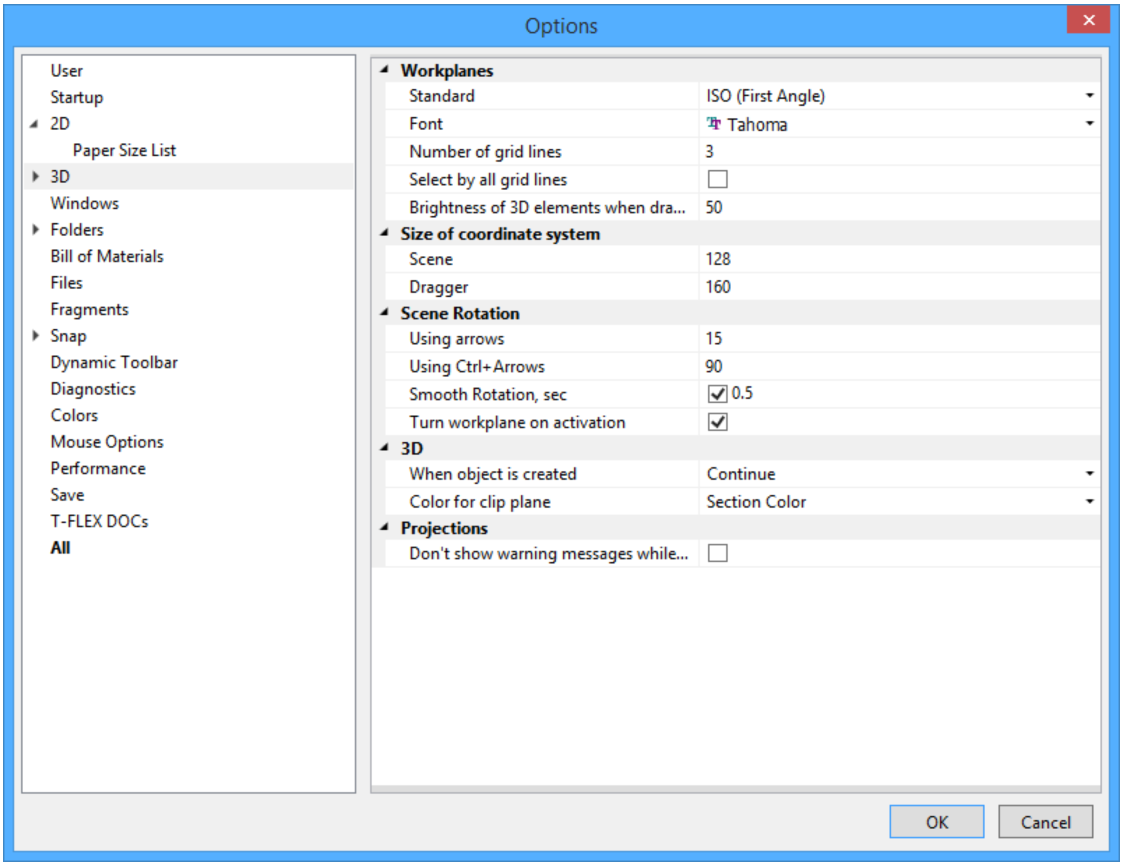
Number, **X** and **Y**. The number of zone columns and rows respectively.

Direction. Defines the itemization direction for zones: left to right or right to left, top down or bottom up.

First displayed symbol. Defines, which of the zone-defining symbols (in the X or Y axis), will stand first in its notation.

“3D” Tab

This tab is specific to the three-dimensional version of the system. It defines the settings used while working with a 3D model.



The **Workplanes** group of settings define various parameters of workplanes:

Standard. Defines the situation of the three default workplanes per the selected standard as follows: **ANSI** – frontal elevation view, bottom-up plan view, right-hand side view; **ISO** - frontal elevation view, top-down plan view, left-hand side view.

Font. The selected font is used for displaying the name of the workplane or workplane's type (depending on settings) in a 3D window.

Number of grid lines. This parameter defines the number of intermediate lines in the image of a workplane in the 3D view representation.

Selection of workplanes in the 3D view by default is restricted to picking at the outer lines (the border) of the workplane. If necessary, the selection can be expanded on all the lines of the workplane image, both the border and the inner grid, by setting the flag **Select by all grid lines**.

Brightness of 3D elements when drawing on a workplane, % specifies brightness of 3D elements, when drawing on a face or surface.

The following group of parameters allow setting the **Size of coordinate system** image on the 3D scene.

Dragger. Defines the size of coordinate system-like draggers used in various 3D commands.

Scene. Defines the size of the coordinate system image displayed in the lower-left corner of the 3D window.

The **Scene rotation** group defines the modes of spinning the 3D scene by certain increments, as follows:

Using Arrows. Defines the angle in degrees of rotating the 3D scene per a keystroke when using the two pairs of arrow keys, plus the third pair <Page Up> and <Page Down>.

Using Ctrl+Arrows. Defines a second spin-with-key mode for rotating the 3D scene by a different angle. This is similar to the "Using Arrows" parameter, except is used in combination with the <Ctrl> key.

Smooth rotation. This flag sets the smooth 3D scene rotation mode during reorientation to a standard view. The input box on the right-hand side defines the reorientation duration in seconds.

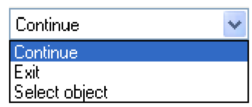
> Note that setting the mode of rotating the 3D scene with respect to the global axes (the command **3RS: Rotate About Global/Local axis**) makes the 3D scene spin with respect to the axes of the world coordinate system. Otherwise, the 3D scene will spin with respect to the axes of the screen coordinates.

Turn Workplane on Activation. If this flag is set, the 3D scene will be reoriented on calling the **Activate Workplane** command so that the active workplane becomes parallel to the screen.

3D group

When object is Created. This attribute defines the system behavior upon creating a 3D element as follows:

Continue. The system remains within the current 3D command after creating any element.



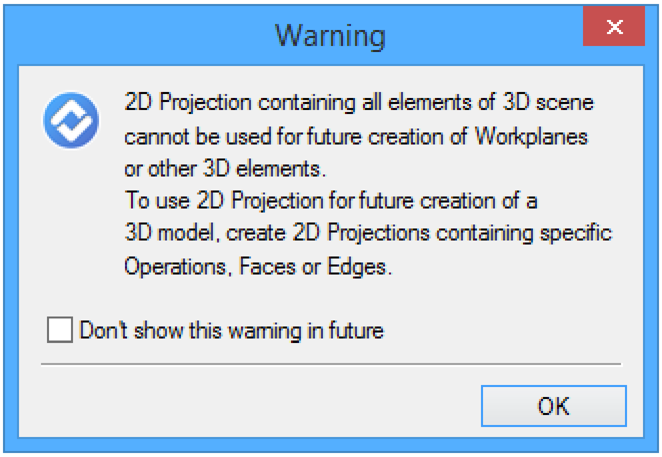
Exit. The current command automatically completes upon creating any element.

Select object. The command automatically completes upon creating any element. If the created 3D element is a construction one, it is placed on the clipboard (gets selected). This mode may be convenient when the user creates an element and instantly proceeds working with it. The following is an example of such common sequence of actions: create a workplane – activate it – use it for creating an extrusion profile.

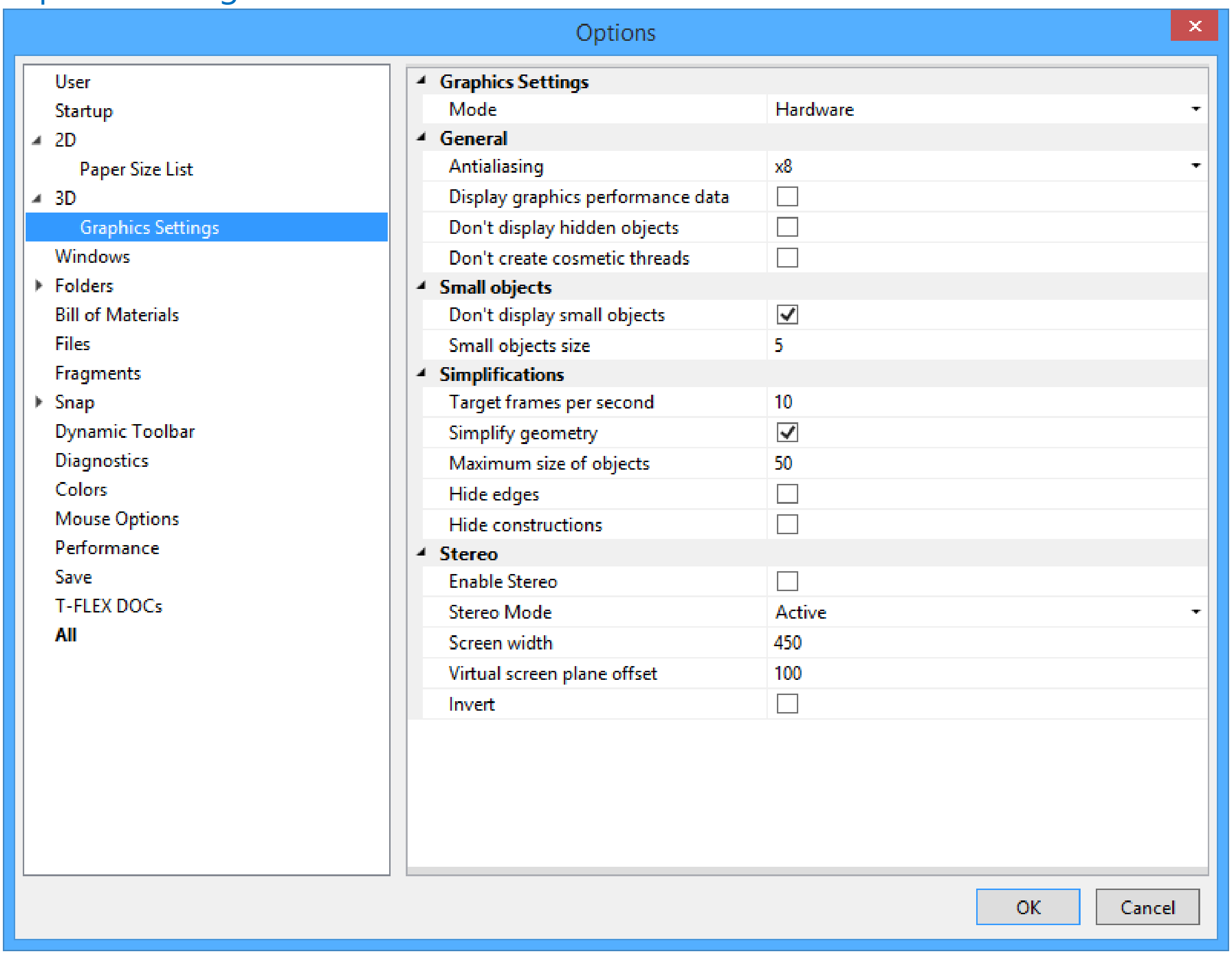
Color for clip plane. You can select S**ection color**, **Body color** or **Body material** from the drop-down list. It will be used for clip plane displaying.

Projections group:

Don't show warning messages while creating projections. If this flag is set, the warning message will not be displayed while creation of 2D projection on the plane created on the base of all elements of 3D scene.



"Graphics Settings" Tab



Graphics Parameters group:

Mode. The option allows us to select one of the two modes: Hardware or Software. The software mode is suited for the case when the graphic card is outdated or absent (the mode of remote desktop) and graphics is not accelerated by the board.

The hardware mode provides more efficient and productive work of the system.

When choosing a parameter of the "3D graphics parameters" dialog the information about application of the selected parameter will be available for the user in the lower part of the dialog window

Group of parameters **General**.

Antialiasing. This parameter controls the number of pixels used for drawing the smoothed image. This parameter allows us to decrease the "saw-toothed nature" of the line on a 3D model. The "Smoothing" parameter is available only when using the Hardware mode. By default the "Smoothing" parameter is disabled. .

Display graphics performance data. If the given mode is enabled then in the upper left corner of the work window–are displayed the graphics system's current parameters that affect the system's productivity. By default this parameter is disabled.

Don't display hidden objects. Enabling this option allows us to speed up rendering the image due to ignoring the scene's objects that are totally hidden by other objects. This parameter is available only when using the Hardware mode.

Don't Create Cosmetic Threads. When this option is activated, the cosmetic threads are not displayed in 3D models. This option has action only on the image created in the 3D window and does not affect 2D projections and dimensions on the threaded surfaces.

Group of parameters **Small objects**.

The **Small objects size** parameter allows us to manually select the maximum size of objects on the screen in pixels which will not be drawn in the mode of the given optimization.

Enabling the **Don't display small objects** mode does not allow the system to draw the objects with sizes smaller than the given maximum size.

Group of parameters **Simplifications**.

The settings parameter **Target frames per second** gives us a capability of setting up the optimum speed for redrawing 3D bodies.

The **Simplify geometry** option allows us to simplify the visible geometry depending on the size of objects on the screen. When for specified values of frequency of frames per second the detailed image of the model cannot be formed, the system determines which bodies have the images taking the smallest area in the frame. These bodies (their images) are drawn in a simplified manner in the form of parallelepipeds. Thus, the image of the entire model is simplified until the time of rendering the frame is smaller than the frequency of redrawing the screen.

The **Maximum size of objects** parameter allows us to manually select the maximum size of objects in pixels which can be rendered as parallelepipeds when enabling this optimization.

Hide edges. If this flag is set, then the outline edges will not be drawn in dynamic picture mode and in automatic rotation mode.

Hide constructions. If there are many constructions in the assembly (for example LCSs from 3D fragments), the scene rotation can be slowed down because of their rendering. If the flag is set, the constructions will be suppressed.

Stereo group allows to setup stereoscopic images viewing. For viewing of stereoscopic images of 3D models, it is required to use 3D glasses.

Enable Stereo. Activates stereoscopic displaying of models.

Stereo Mode. You can select mode of stereoscopic images creation from the drop-down list.

Active mode uses quad buffering to receive stereoscopic image. Quad buffered stereo provides each eye a unique view from a slightly different perspective by using four buffers (front left, front right, back left, back right) rather than the traditional two buffers (front, back).

Vertical. Images for right and left eyes are placed under each other.

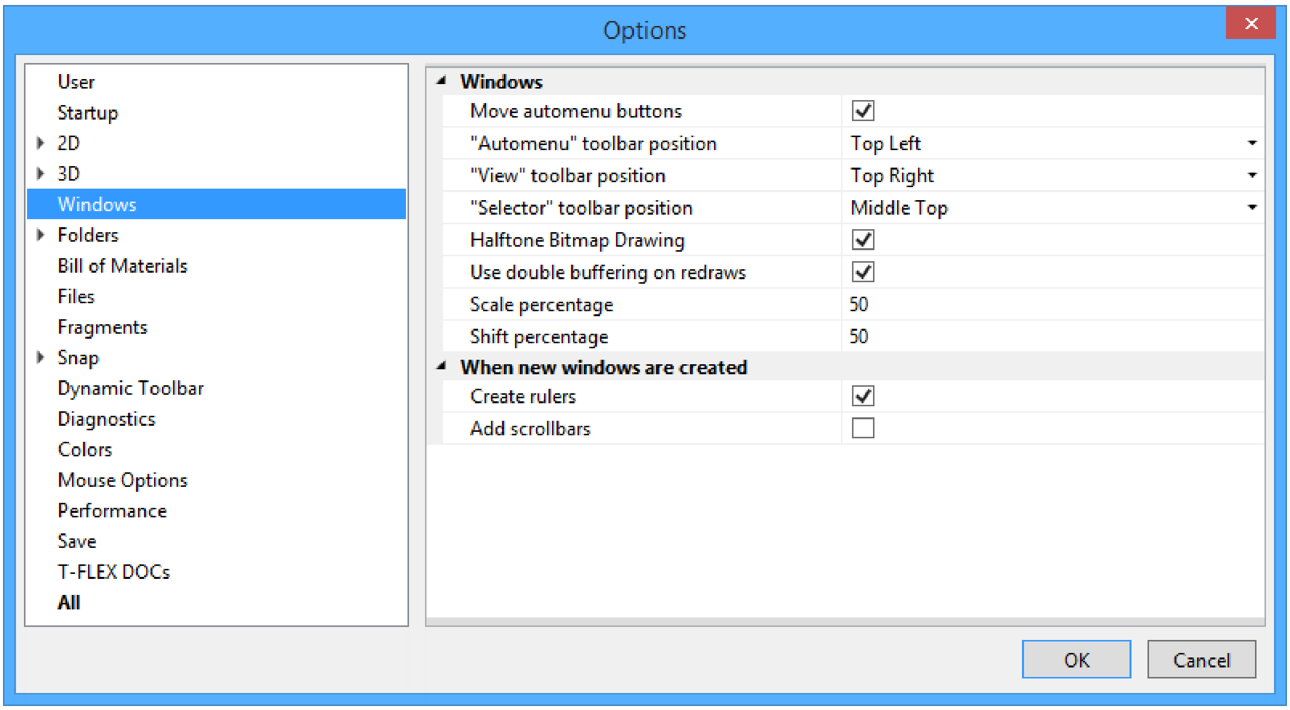
Horizontal. Images for right and left eyes are placed horizontally near each other.

Screen Width - width of the display or projection (if you use a projector) in millimeters.

Virtual screen plane offset specifies the offset of the stereoscopic image plane in percent according to the current position.

Invert swaps the images for right and left eye.

"Windows" Tab



Move automenu buttons. Places automenu options in columns if there is no space for the whole automenu.

You can setup the following parameters for ribbon mode using the drop-down lists:

Automenu toolbar position, View toolbar position, Selector toolbar position allows to select a location of the corresponding toolbars on the screen.

Use double buffering on redraws. This flag sets the double buffering mode for redrawing 2D document windows that enhances perception of redrawing by removing flickering.

Halftone Bitmap Drawing. This option turns on the halftone mode of displaying bitmaps: raster pictures, inserted into T-FLEX CAD documents, ray-tracing results, etc. By default this parameter is turned on.

Scale percentage. Specifies percent for zooming of the current window visible part.

Shift percentage. Specifies percent for panning of the current window visible part.

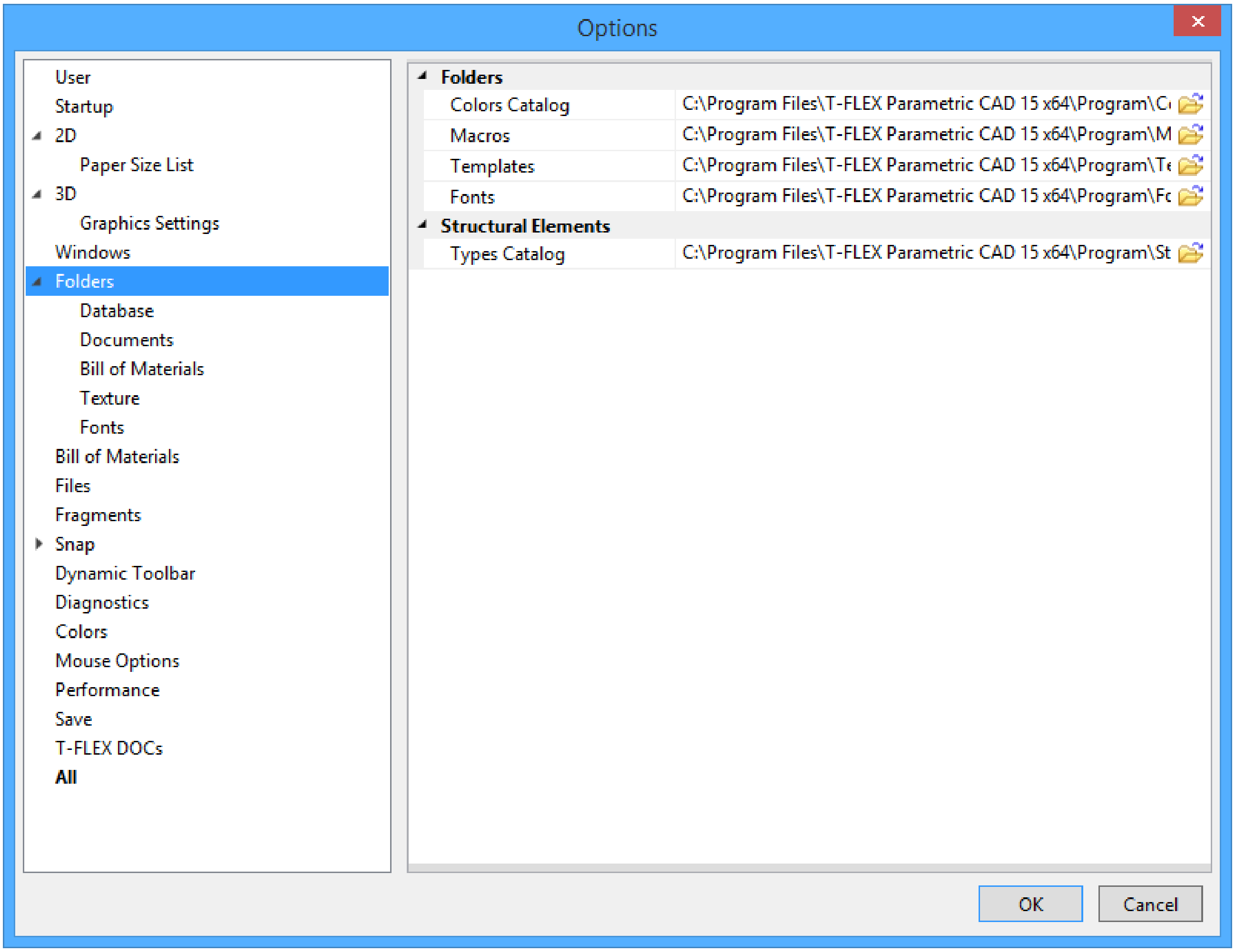
> These parameters can be also set in **ZW: Zoom area** command**.**

When new windows are created group of parameters defines the following modes:

Add Scrollbars. Displays scrollbars when opening document windows.

Create Rulers. Similar to the previous, if unset, the rulers are not created on opening the window.

"Folders" Tab



Folders group:

Colors Catalog. This setting specifies the path to the folder that contains colors catalogs. For example, the colors are used to create color-based materials and set colors on the "Colors" tab.

Macros setting specifies the path to the folder, which content will be displayed in the **Macros** window.

Prototype Folder. This setting specifies the path to the folder whose content will be displayed in the "Welcome" dialog box on the application startup and in the **FP: Create New Document Based on Prototype** command dialog box (**File > New From Prototype…**).

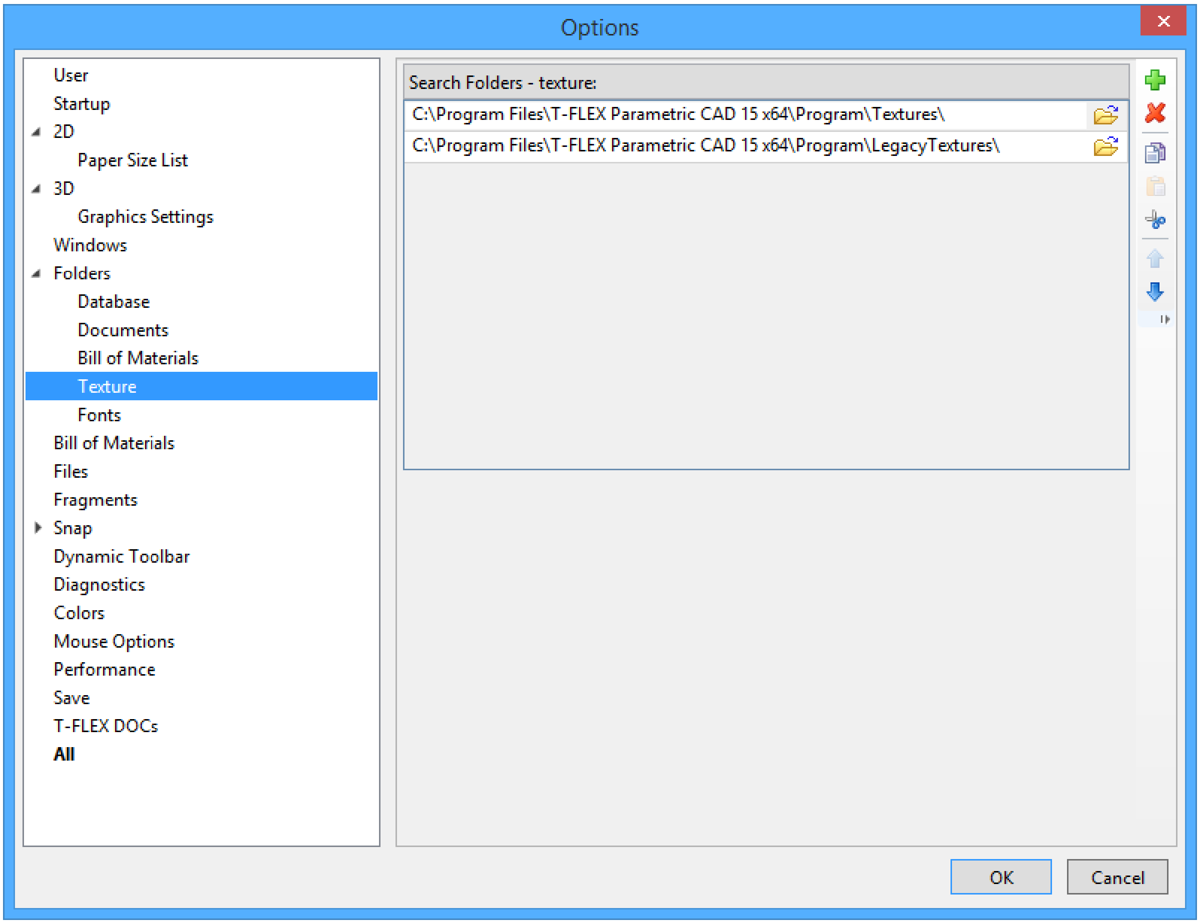
Font Folder. This parameter indicates the path to the folder in which the files of description of SHX fonts are located. If necessary, the user can select the fonts which are not from the standard folder.

Structural elements group:

Types Catalog. This setting specifies the path to the folder which content will be displayed in the **Editor of structure elements types**.

"Database", "Documents", "Bill of Materials", "Texture", "Fonts" tabs

These tabs are used to define a list of additional folders for searching files that could not be found in the standard folders.



The  button adds a new string for search folder. The  button brings up a browser window for selecting the desired folder on the disk. Select the folder on the tree using the cursor. Upon confirming the selection with the **[OK]** button, the browser window closes and the name and path to the selected folder will appear in the selected file type input box.

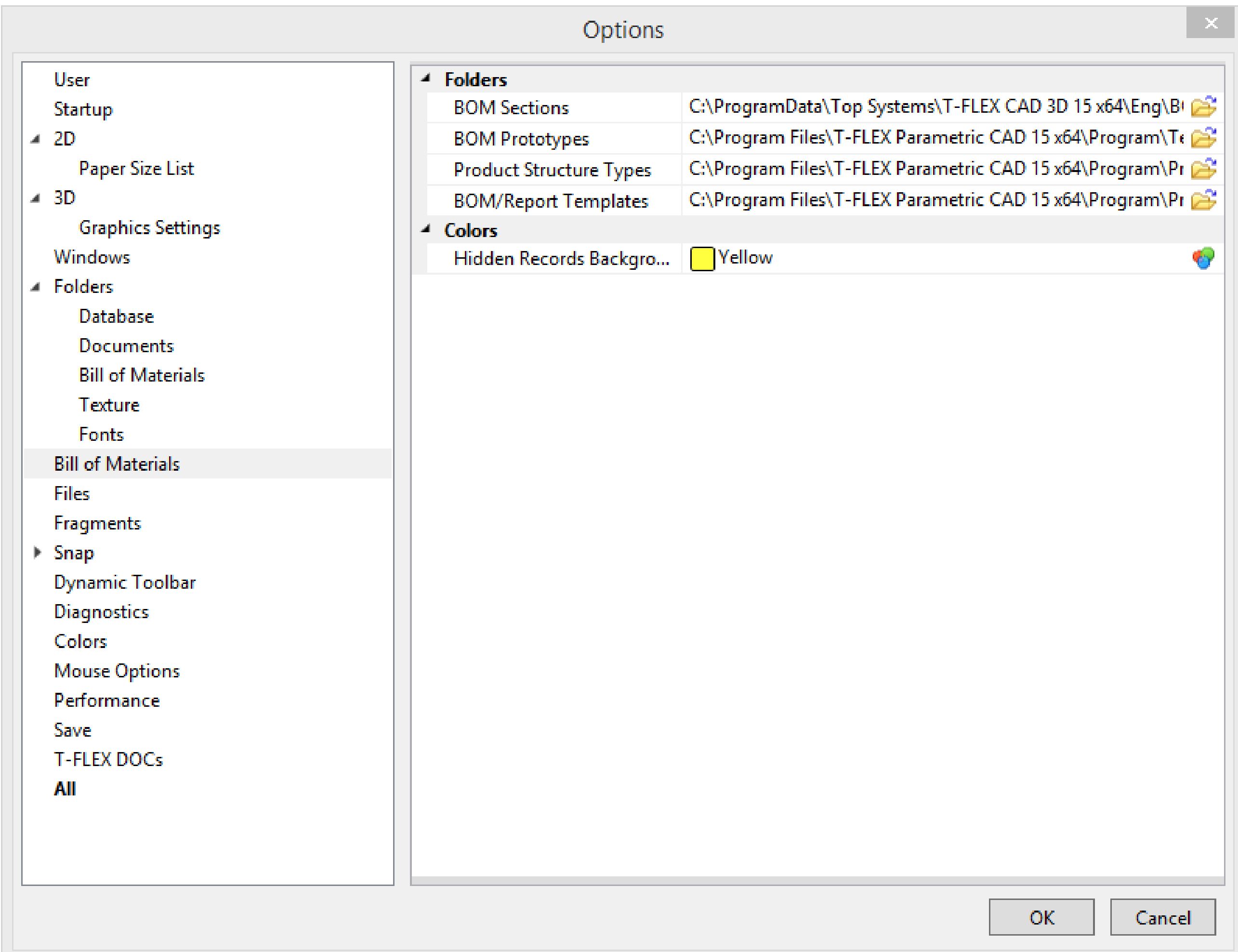
The button allows deleting a folder selected from the list of defined folders.

The and buttons move the selection frame up and down the list of defined folders.

To work with search folder strings using the clipboard use buttons , , .

Example: suppose we open a drawing file with a fragment assembled in it, while the path to the fragment has changed. In this case, a message will be displayed about an error opening the fragment file. The fragment itself will not be displayed on the drawing. This error can be fixed by either changing the path to the fragment, or using the described tab and defining an additional folder where the fragment file is located. In that case, the system will be automatically searching for the fragment file in the additionally defined folder as well, and the error opening the file will not occur.

"Bill of Materials" Tab



Folders Group:

BOM Sections. Defines the name and the path of the database file keeping the set of BOM groups. The specified database will be the one used in creating new and reading already created BOMs.

BOM Prototypes. Defines the path to the folder of template files used for creating new BOMs. The files from this folder will be displayed in the **BC: Create Bill of Materials** command dialog box.

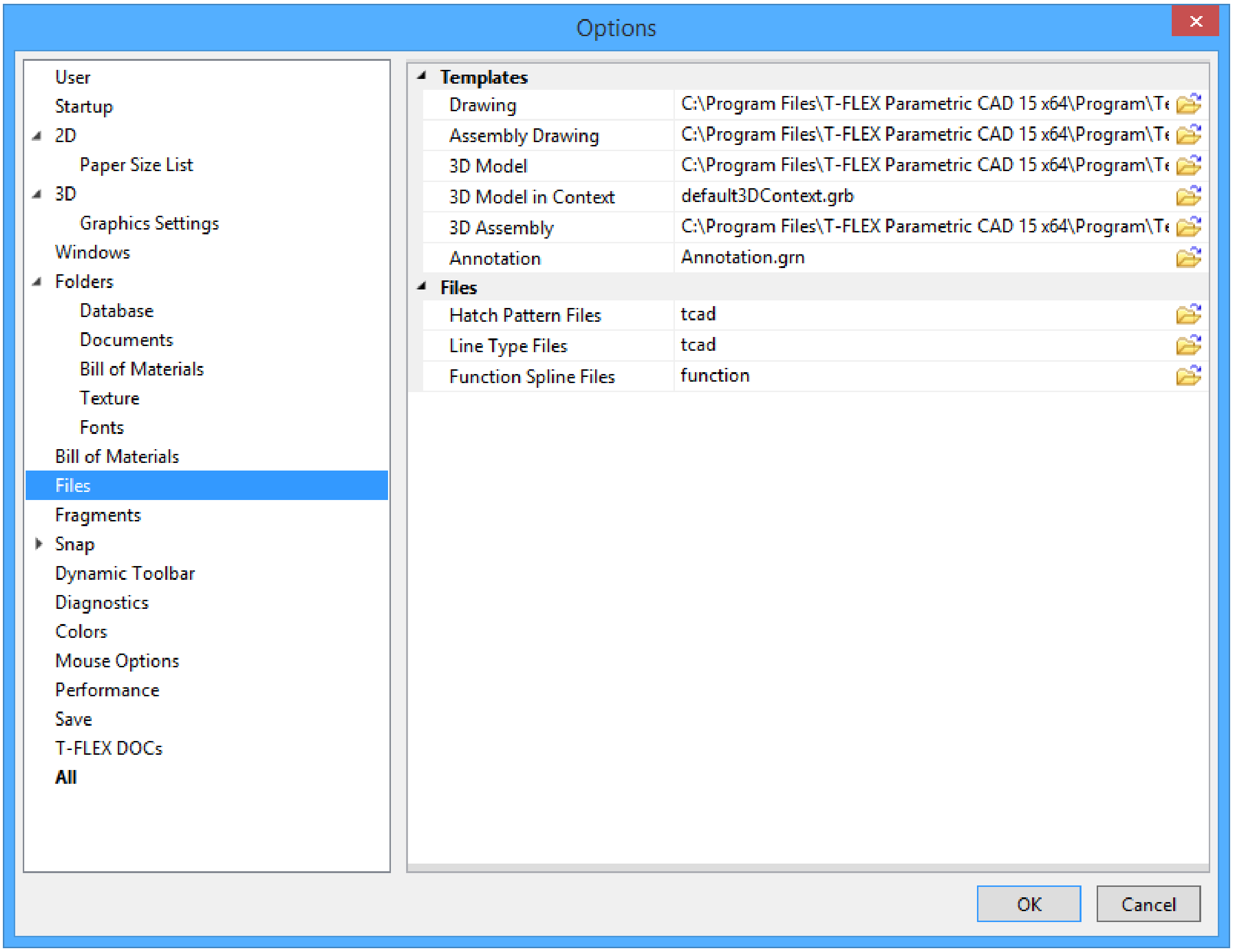
Product structure Types. Defines the path to the folder where XML files with product structures descriptions are saved.

BOM/Report Templates. Documents from the folder will be displayed in the templates list when you create a new report based on the product structure.

Colors Group:

Hidden Records Background Color. Defines hidden records background color. Hidden are considered the records deleted from the BOM but yet still stored in its inner data structure. The way of displaying hidden records in a BOM is defined accordingly.

“Files” Tab



Templates group:

Drawing. A prototype is a T-FLEX CAD drawing file, whose data is used for initializing a new drawing. You can create several prototype files. In the case the path is not specified with the name of the prototype, the system will search for it in the application folder (PROGRAM).

One can save a prototype file using the command **File** > **Save as Prototype**. In this case, the prototype will be saved in the folder *...\PROGRAM\ Template*. To create a new drawing with the same settings as in this prototype, use the command **File** > **New From Template...**

Assembly Drawing, 3D Assembly. A product structure of each assembly template includes records for assembly forming. It’s their only differ from standard templates.

Assembly drawing template

3D Model. This is the prototype file with the 3D window settings used for creating a new 3D model.

3D Model in Context. This is the prototype file that contains the settings of the 3D window that are used upon creation of a new 3D model of the fragment in the context of the assembly.

Annotation. This is the prototype file for creating a new annotation using the annotation editor. The file path is automatically assigned upon installing the Annotation Editor.

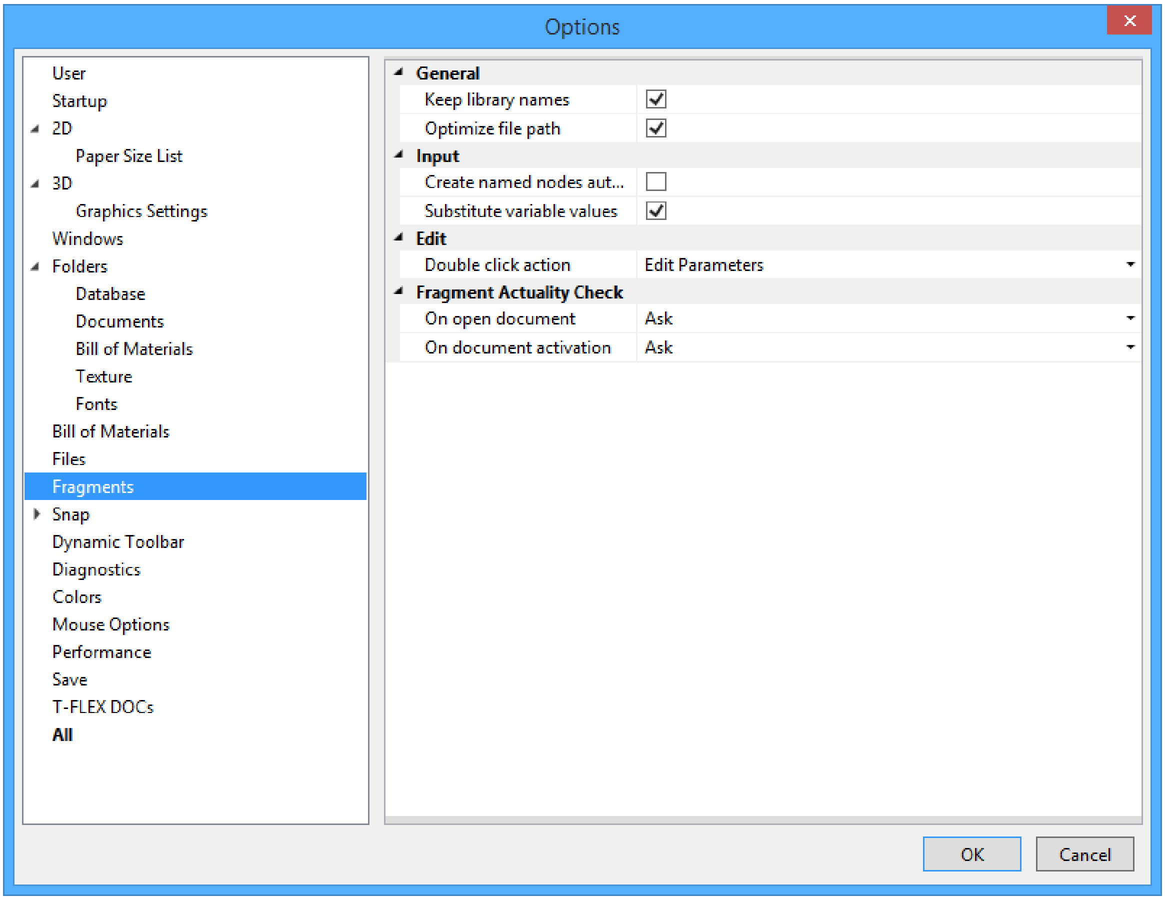
Files group:

Hatch pattern. The system uses hatch pattern files corresponding to the respective AutoCAD ".PAT" files. The file TCAD.PAT, included with the system installation, defines the hatch filling patterns. You can define any hatch pattern file of your own.

Line type. The system has several built-in line types (continuous, thin, waves). The rest of types are defined by the line type file. The system uses the file TCAD.LIN. Its format complies with the line type files of the AutoCAD system. You can define any type file of your own.

Function spline. This special file contains data for setting up the menu of the Function Spline creation command. (A function spline is a kind of a construction line.) To create new functions, modify the standard file named "FUNCTION.DAT" or create a new file by sample, and set its name in this input field of the dialog box.

“Fragments” Tab



This tab defines the options used in inserting and editing fragments. Parameters on this tab can also be set from the fragment parameters modified under the fragment insertion and editing commands using the **[Options...]** button.

Keep Library Names. If set, library names will be saved on inserting a fragment from a library. Otherwise, the absolute path of the respective library folder will substitute the library name.

Optimize File path. If set, the library name or the path will not be entered in the case the current assembly document and the respective fragment are in the same folder. This helps moving both the document and the fragment file to another folder without changing folder settings.

Input. This group defines the flags used while assembling fragments:

Create Named Nodes Automatically. This flag defines whether the new nodes will be created on the current assembly drawing based on the named nodes from the fragments being inserted. The nodes created from fragments can be used for further construction, dimension placement, etc.

Substitute Variable values. In case this parameter is set, upon the fragments' insertion, their external variables are automatically set equal to the values, assigned for these variables in the model of the fragment. Otherwise, the values of the variables are not prescribed.

Edit. Examples of editing fragments:

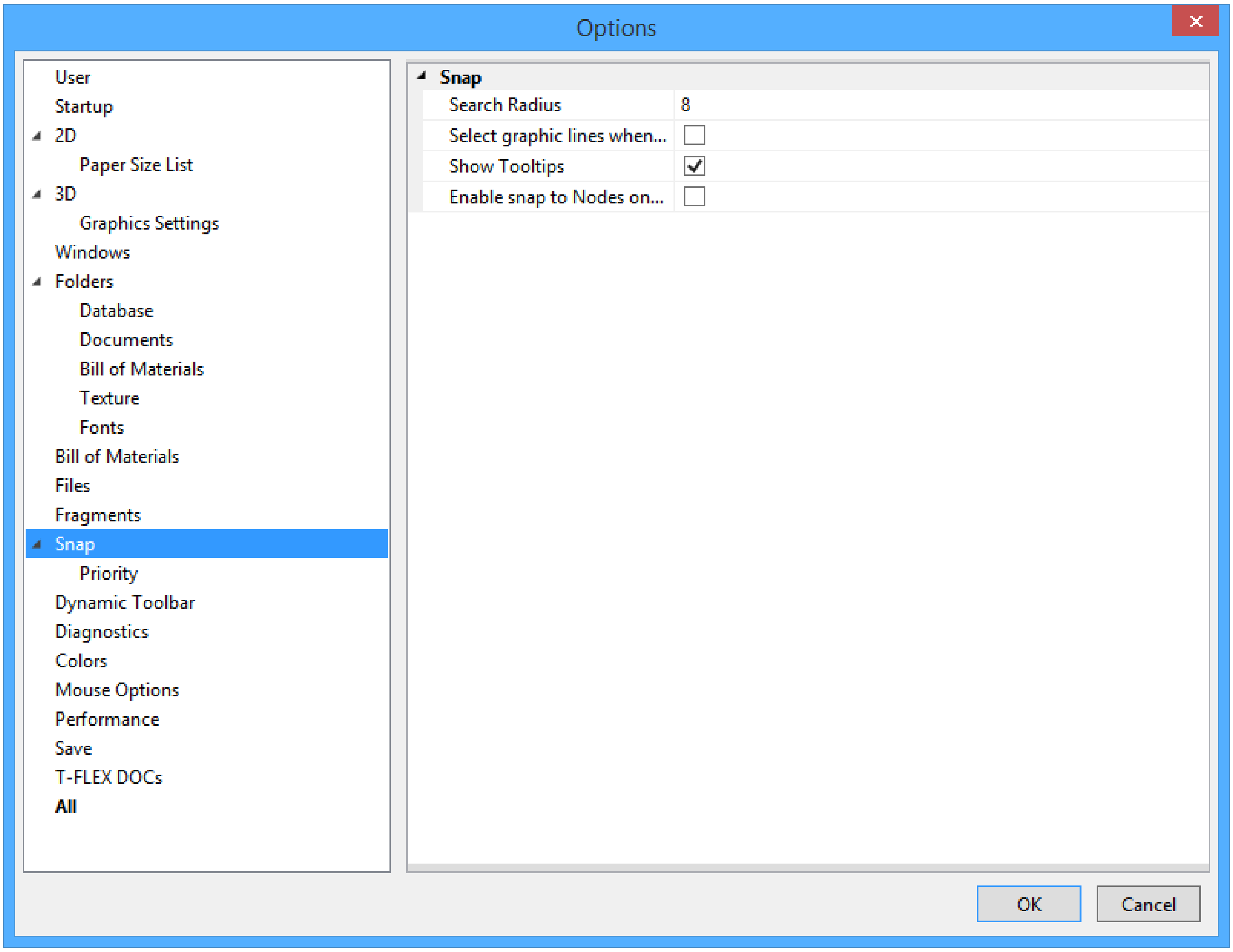
Double click action. This attribute defines what kind of action occurs on the left mouse double-click during fragment editing.

Fragment Actuality Check. This group of parameters defines the system behavior when the files are modified of the fragments that enter the current document (the current 3D assembly), in various situations:

On Open Document. The parameter defines the system behavior upon opening a 3D assembly (if it was found that a fragment files were modified since the time of saving the assembly). The following choices can be made from the list: **Update** - the fragments will be updated, **Don't Update** - skip updating fragments, **Ask** – prompt the user about how to proceed upon finding modified fragments. The default setting is "Ask".

On Document Activation. This parameter defines the system behavior upon returning to a document window (when simultaneously working with multiple T-FLEX CAD documents). In this case, the state of the current 3D assembly is also checked for the consistency with the fragment documents stored on the disk. If the check finds fragments, whose documents were modified after the last check, the system will proceed according to this parameter setting: **Update**, **Don't Update**, **Ask**. The default is "Ask".

"Snap" Tab



This tab serves to define parameters that are used when working in the object snapping mode (including the cases of sketch creation).

Search radius. Sets the radius in pixels for searching the elements of the system on the screen. This parameter is used in new element creation in object-snapping mode. Keep in mind that this parameter setting overrides the **Node join distance** parameter defined on the "Preferences" tab.

Select Graphic Lines when creating hatch contours instead of Construction Lines. Setting this parameter allows selecting graphic lines when constructing hatches and 2D paths. This is required in cases when the construction lines coincide with graphic lines. This option helps set up object snapping so as to have the desired elements selectable in the complicated cases upon the cursor approaching,– for example, graphic lines (if the parameter is set). At the same time, it is still possible to select other elements by using the keyboard commands (<C> – select a circle, <L> – select a line, etc.).

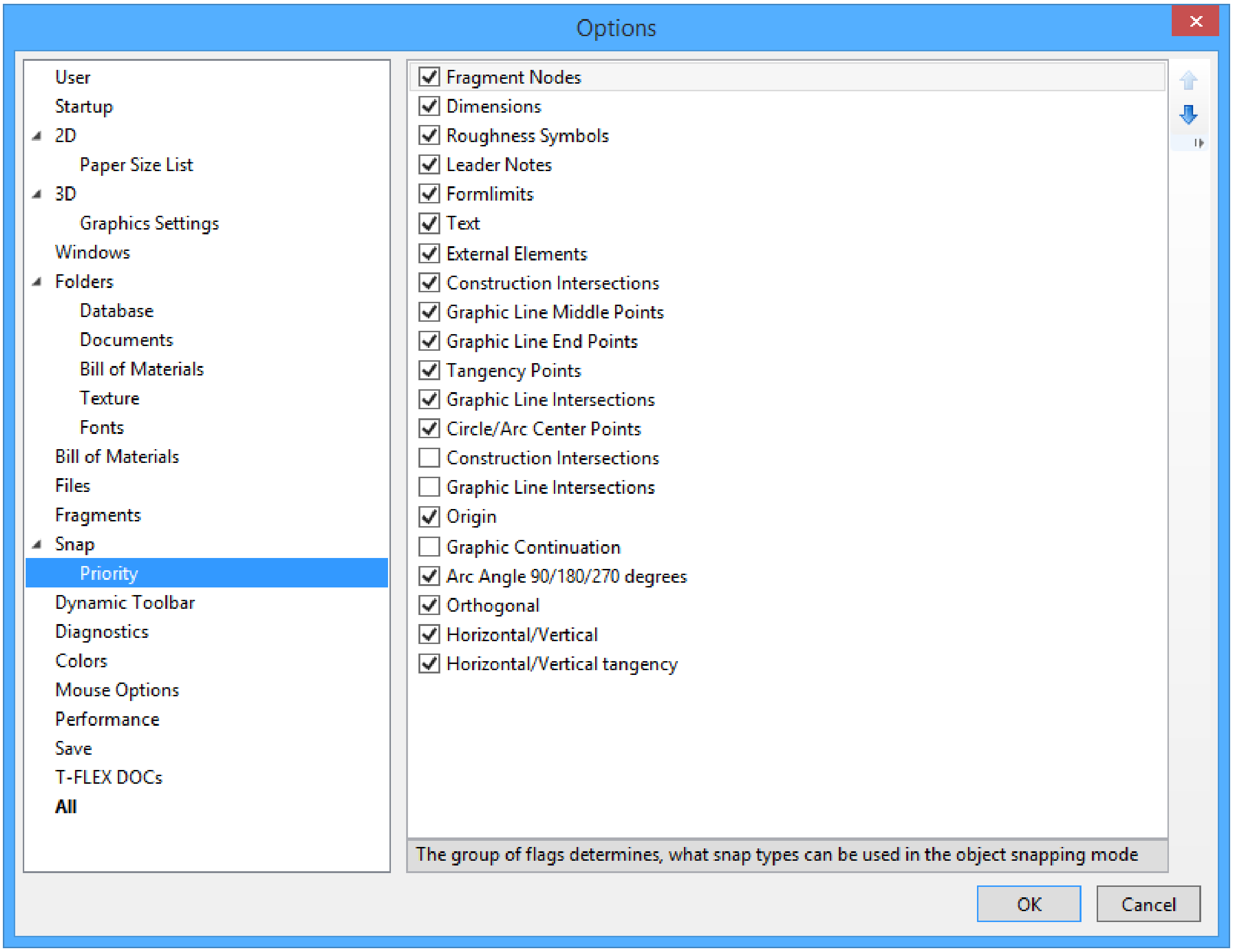
Show Tooltips. Sets the pop-up tooltip display mode when selecting elements on the drawing while in object-snapping mode.

Enable Snap to Nodes on Frozen Layers. If this flag is turned off, snapping to nodes, located on “frozen” layers is not possible. Upon turning on this flag, snapping to such nodes becomes possible.

Priority Tab

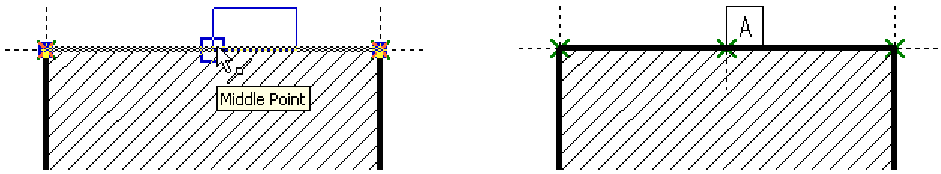
Priority Tab determines, what snap types can be used in the object snapping mode.

> Most snaps can be managed even outside the command **SO: Set System Options** – by using icons on the “Snaps” toolbar.



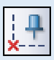
For example, suppose, a tolerance needs to be placed next to a dimension. To achieve that, enable the flag **Enable snaps to/Dimensions**. Call the command **Draw** > **Tolerance**. As the cursor approaches a dimension defining point, it becomes marked with a rectangle. If you then click , a node will be created, and an annotation element – tolerance will be created.

To snap a tolerance to the midpoint of a graphic line, enable the flag **Enable snaps to/Graphic Line Middle Points**. In this case, while in the tolerance creation command, midpoints of graphic lines will be getting highlighted upon cursor approaching. If you then click , a node will be created, and the tolerance annotation element snap to it.



A 2D node created using any-type snap can be free (not maintaining a relation with the elements based on which it was constructed, after the creation) or constrained (tied) - where the node relation with the source elements is maintained.

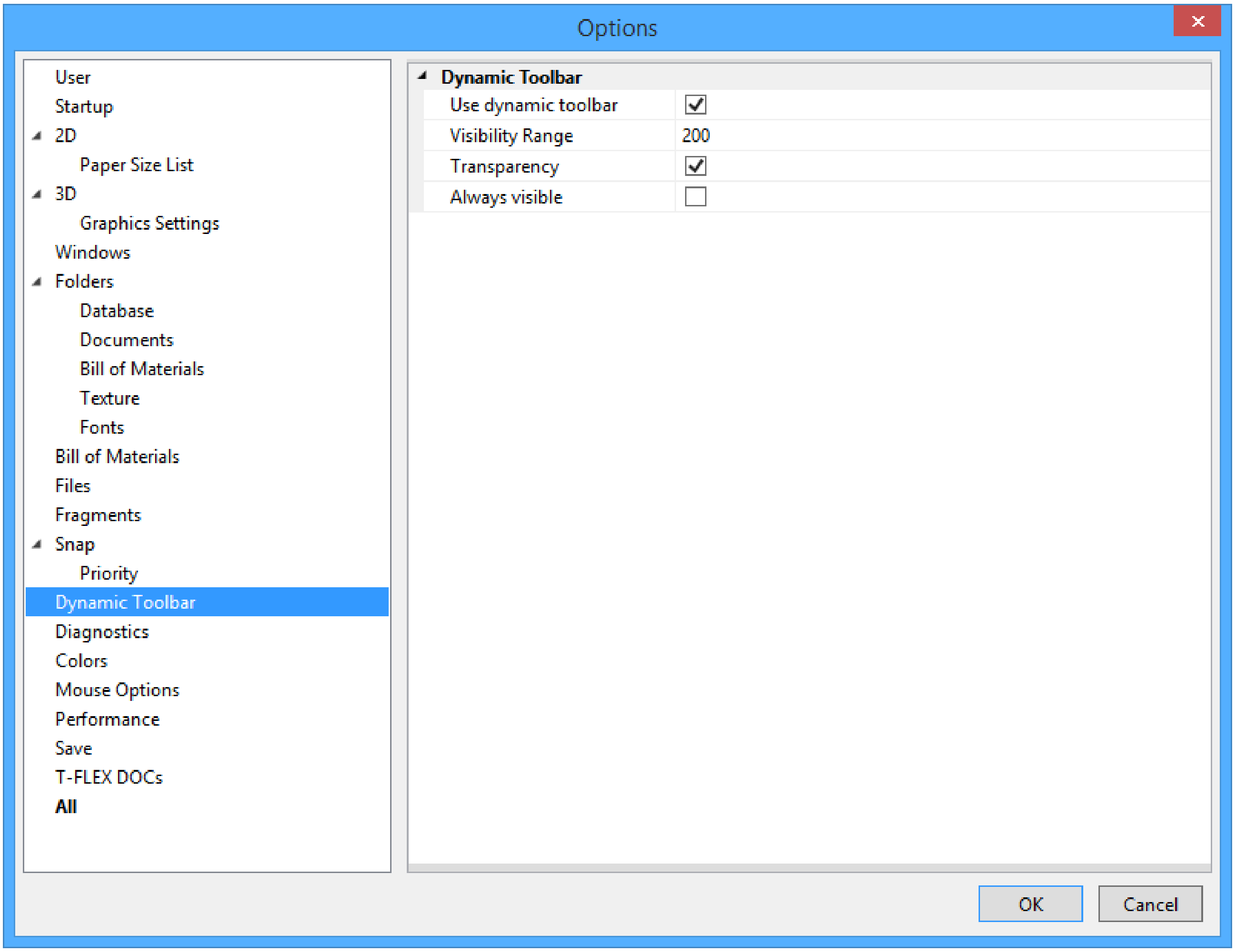
When using snaps to a construction line intersection, a circle center, end points of graphic lines, defining points of drawing annotation elements (dimensions, leader notes, roughness symbols, tolerances), as well as 2D fragments – then constrained nodes are always created.

When using all the rest snap types, the state of the auto parameterization mode is significant (the  icon on the “View” panel). If the auto parameterization mode is enabled, then a constrained node is created. With the auto parameterization mode disabled, either a free node is created, or a point with appropriate coordinates (when creating a leader note, roughness, tolerance, cropped view symbol and 2D fragments).

The buttons  and  serve to modify the priority of object snaps. Snap priorities determine, in what order the system will be offering them to the user in the case when several snap possibilities are found. The list **Enable snaps to** has the snaps positioned in the decreasing priority order.

For example, snapping to construction lines has higher priority then “Vertical/Horizontal” snapping (this can be seen by their position in the list). Therefore, when constructing 2D elements using snaps, the system will first offer the user snapping to construction lines, and only after that - snapping to vertically/horizontally aligned nodes.

Dynamic Toolbar Tab



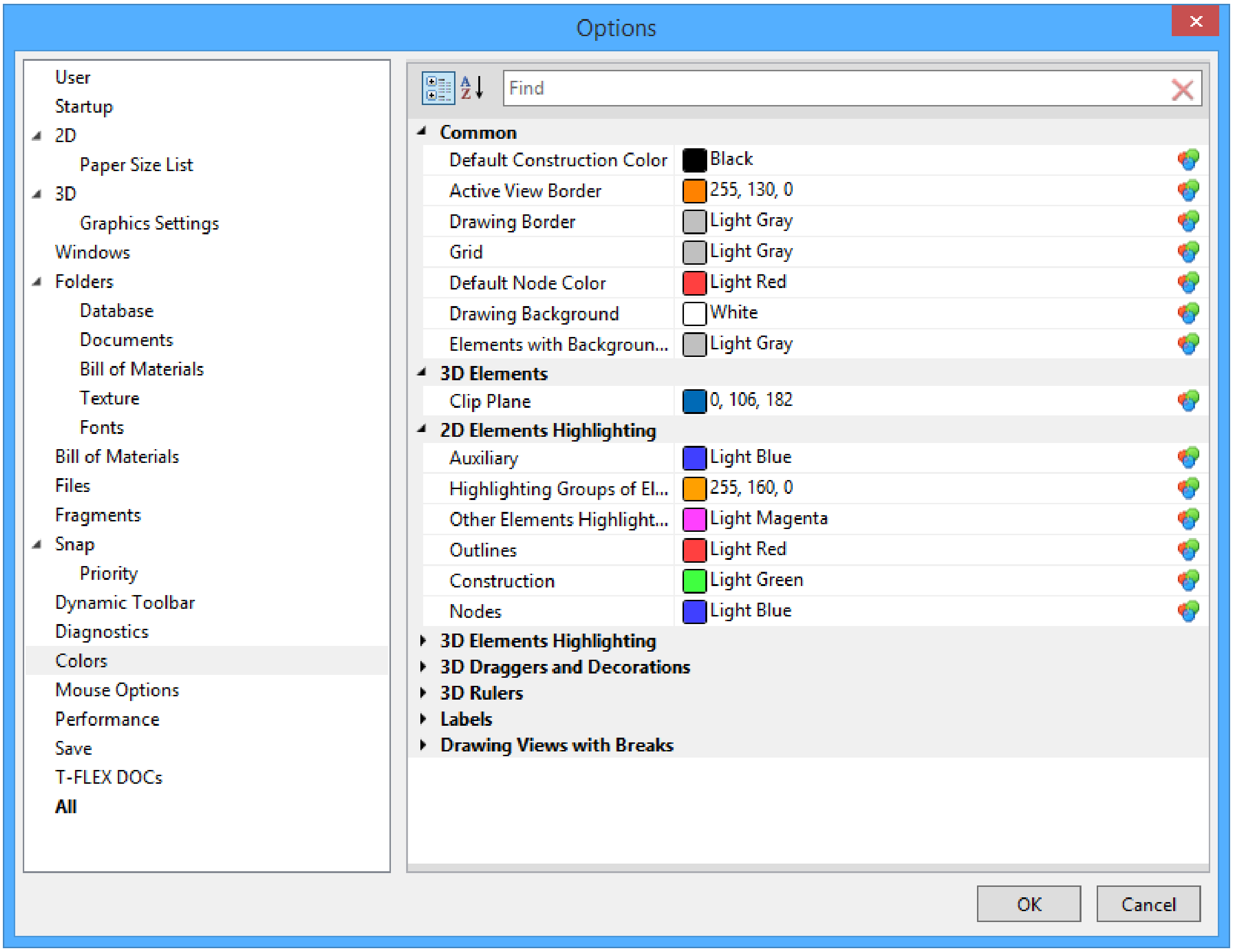
Use Dynamic Toolbar. This parameter controls the image of the dynamic toolbar upon selection of 2D and 3D elements in the command waiting mode with the help of . This toolbar includes commands frequently used for the selected element. For 2D elements the dynamic toolbar is displayed only when the parameter "Transparent Element Editing" is turned off.

Visibility Range – defines a zone around the dynamic toolbar in pixels. The dynamic toolbar is displayed on the screen if the cursor is inside this zone. As soon as the cursor goes outside of this zone, dynamic toolbar disappears.

Transparency. If the flag is active, the transparency of the dynamic toolbar depends on the position of the cursor. The farer the cursor is the more transparent is the toolbar. If the flag is off, the transparency is not changed when moving the cursor.

Always visible. If the flag is active, the dynamic toolbar is visible on the screen until the command is finished. The cursor position does not affect displaying of the toolbar.

"Colors" Tab



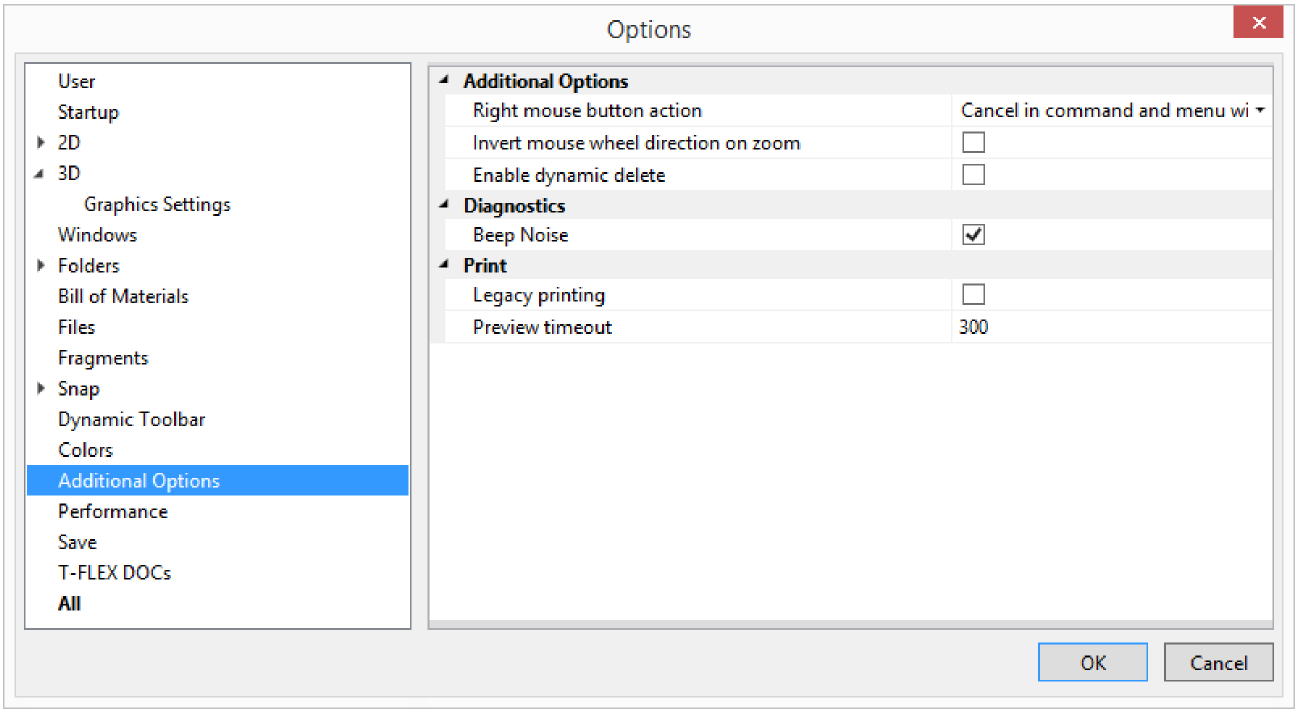
This tab defines the colors used for drawing 2D and 3D model elements on the screen, as well as the application window colors.

The "**Common**" group of colors defines the default colors of various elements of the 2D document window. It also includes such common parameters as the background color of the application windows and the frame color of the active pane in the case the document window is split. These settings do not affect the documents whose respective colors are explicitly defined by the command **ST: Set document parameters**.

Additional groups, such as **3D**, **3D Elements Highlighting**, **3D Draggers and Decorations**, **3D Rulers** are specific to the 3D version of the system, and define various colors of the 3D scene visualization.

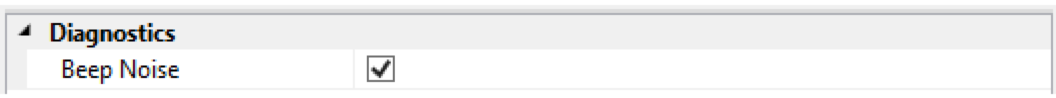
The entered settings can be saved into an external file with the extension "**.tfc*" for future reuse. This can be done using the **[Save...]** button. To load color settings from an external file, use the **[Load...]** button.

“Additional Options” Tab



Diagnostics group

Beep Noise - Messages in **Diagnostics** window will be accompanied by a beep noise.



Right mouse button action. The parameter specifies action type, which is performed upon pressing . You can select one of the types using the drop-down list. According to your choice after pressing in different modes the corresponding context menu will appear or the current command will be cancelled.

For example, if the **Menu in commands**... item is selected, the context menu duplicates the automenu upon pressing during the command execution.

Invert Mouse Wheel Direction on Zoom. This option serves to switch to the opposite the zooming direction in the system windows with the mouse wheel spinning.

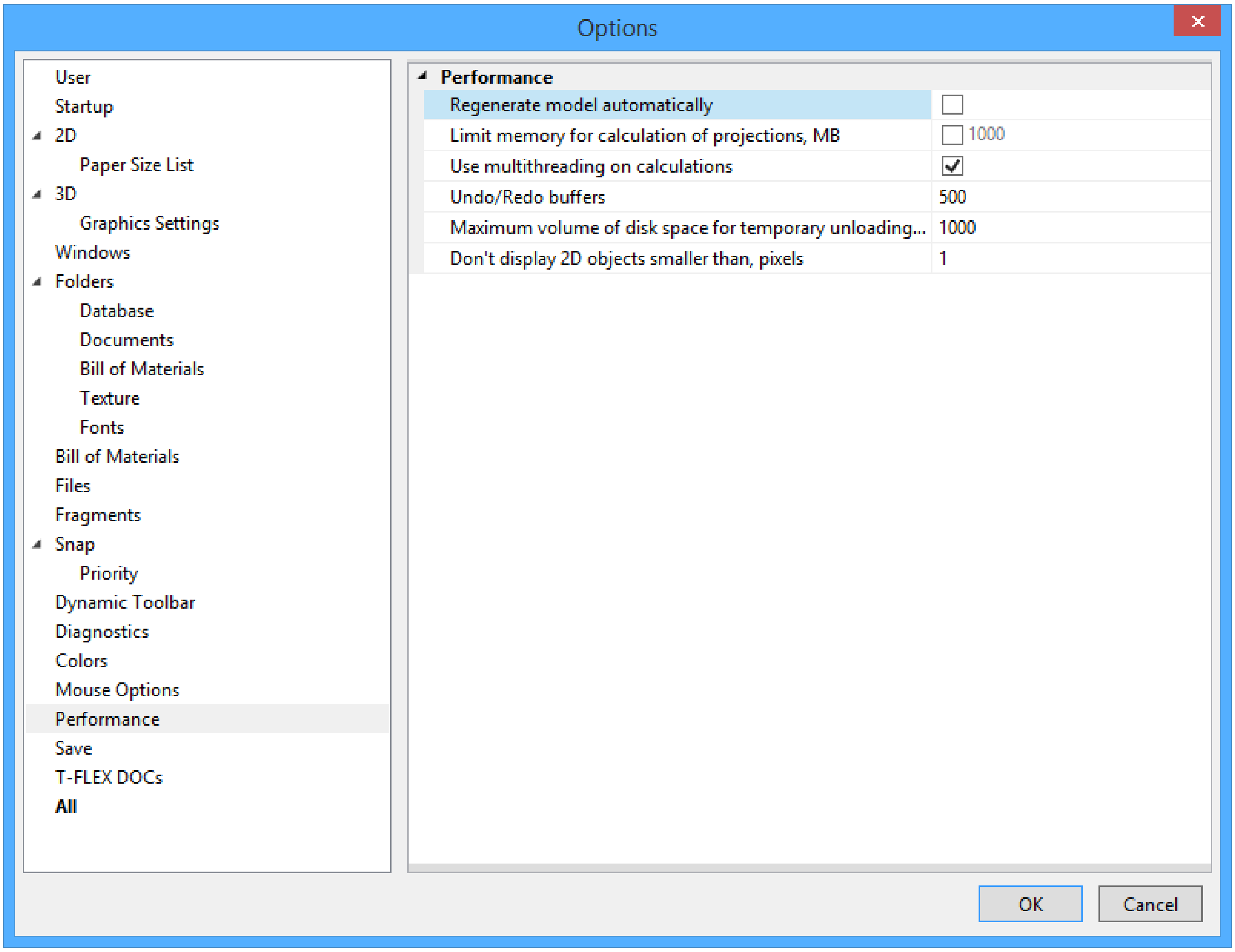
Enable Dynamic Deletion. This flag allows us to activate the mode of “quick deletion”. To delete any 2D element, it is sufficient to move the cursor to the element (the element will be highlighted) and press <Del>.

Print Group

Legacy printing. Use legacy **Print** command.

Preview timeout. If the maximum drawing time at print preview exceeds the specified value (milliseconds), redrawing is not performed and **Show** button appears.

"Performance" Tab



Regenerate model automatically. When this flag is set, the 3D model will be regenerated automatically in case of parameters change (upon exiting the variables editor, changing construction line positions, etc.). If the flag is cleared, then the automatic regeneration does not occur in such cases. To recalculate the model, you would have to manually call the model recalculation command **Tools > Regenerate** or <Alt><F7>.

Limit Memory for Calculation of Projections, MB. This parameter enables to set the upper bound on the size of the memory allocated for recalculation of 2D projections. By default, the following limitation is set:

- For 32-bit operating system Microsoft Windows – 300 MB;
- Upon using 32-bit version of the T-FLEX CAD with Windows x64 – 1000MB;

- For 64-bit version of the T-FLEX CAD x64 with Windows x64 – no limitations.

Use Multithreading on Calculations. This flag activates the mechanism of multithreaded calculations inside the system on computers having multi-core processors or on multiprocessor systems. This enables to increase the speed of regenerating the models. Multi-threaded data handling is used upon calculating geometry of 3D operations, calculating finite-element meshes.

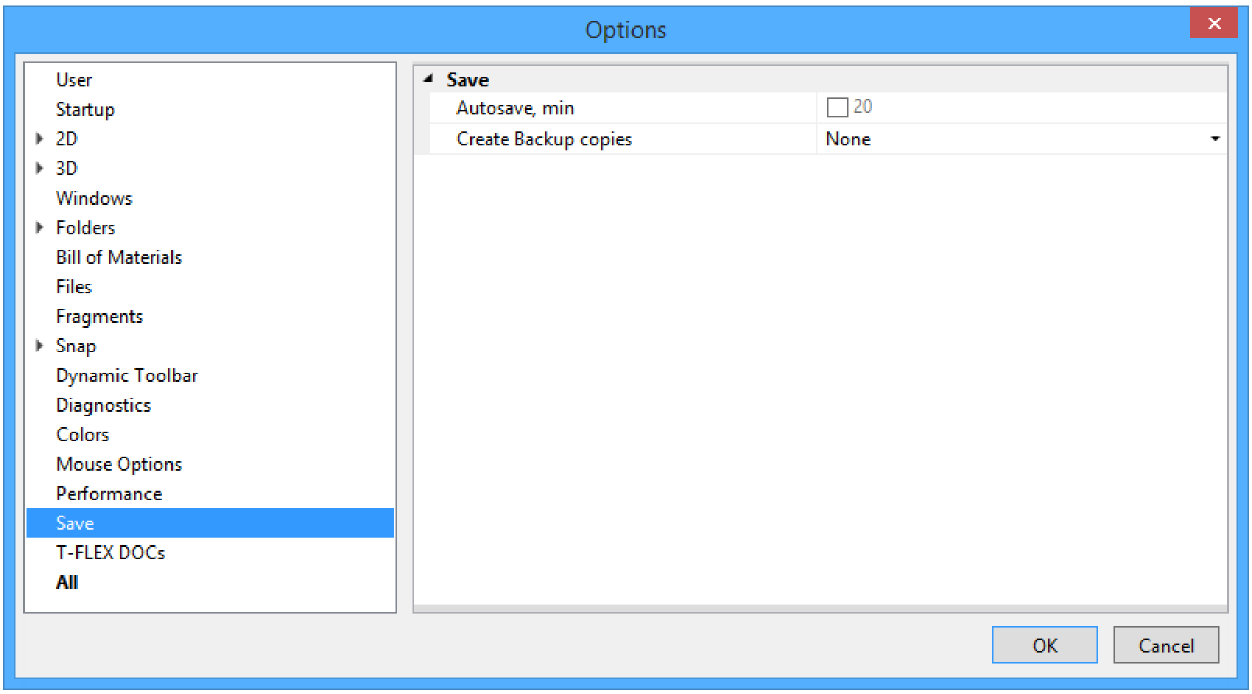
Undo/Redo buffers. You can specify the maximum number of user actions to be remembered by the **UN: Undo Changes** and **RED: Redo Changes** commands. This parameter should be specified before opening a file.

Maximum volume of RAM for temporary unloading parts in large assembly mode, MB. Upon working with large assemblies some of their files are unloaded to the hard drive and some are stored in the RAM. This parameter allows you to specify maximum RAM to operate with large assemblies.

Don't display 2D objects smaller than, pixels. Objects, which size is smaller than the specified value, will not be displayed on the drawing. This option helps to speed up manipulations with the large drawings.

"Save" Tab

Autosave every. When set, the current document is automatically saved after the specified period of time. Saving the model occurs only when switching from one command to another, and is not done while working within the same command, in order to preserve the model data integrity.



Create Backup copies. This flag sets the mode of creating backups on saving document files. The parameter can take the following values:

None – no backup.

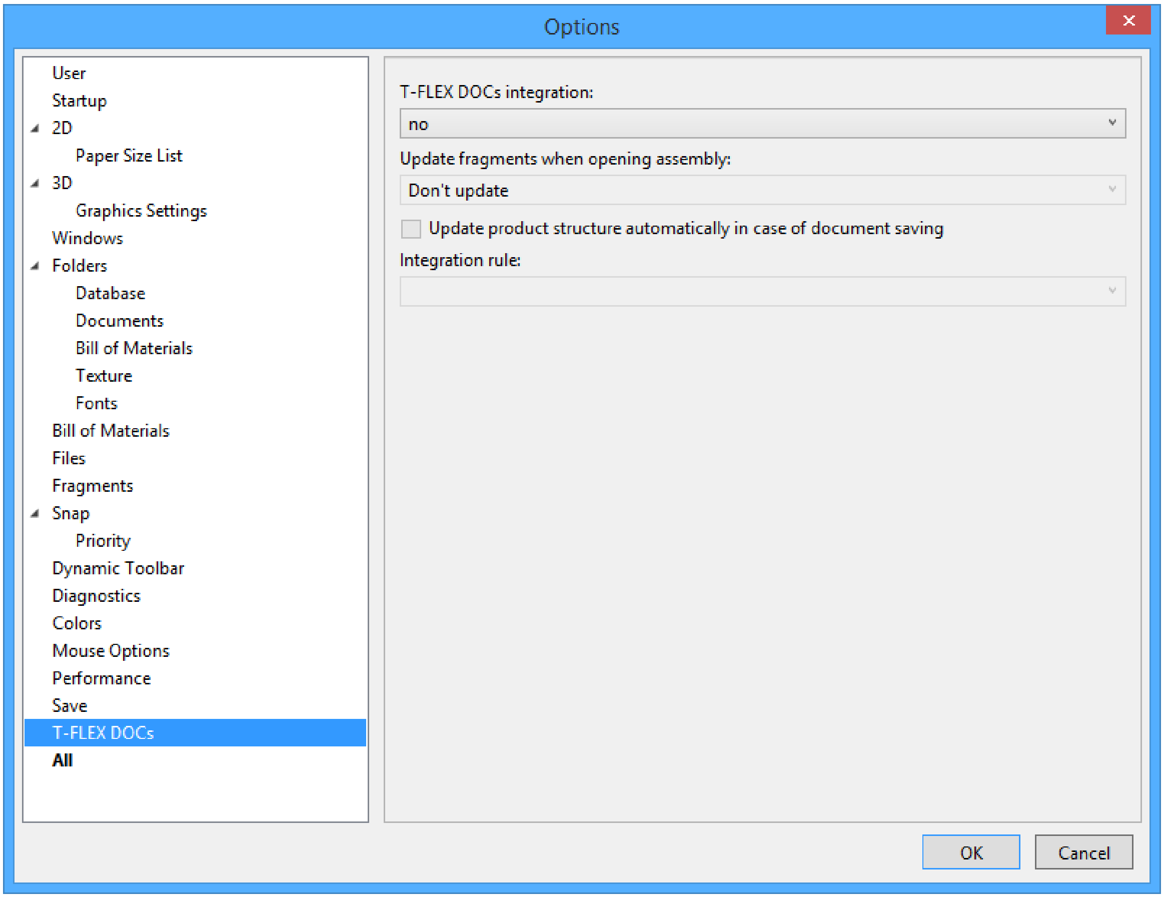
On first save – A ".BAK" file is created on the first save of a document after opening it for editing.

On every save – The previous saved file version becomes the backup copy on every document save.

New on every save – backups are created on each save with different extensions, as ".B01", ".B02", etc.

«T-FLEX DOCs» Tab

This tab defines parameters of the joint work of T-FLEX CAD and T-FLEX DOCs system. The settings for the given tab are available for editing only if T-FLEX DOCs system is installed on a given work place.



The **T-FLEX DOCs Integration** parameter specifies one of the modes of work with T-FLEX CAD:

No — the system works in the regular mode. This mode is chosen by default;

With files and documents of DOCs – the system supports the work with both types of documents: T-FLEX CAD and T-FLEX DOCs ;

Only with documents of DOCs – control over opening and saving the documents is carried out by T-FLEX DOCs system.

Update product structure automatically on saving file. If this flag is enabled, when saving a file the command of saving the product's structure into DOCs is automatically invoked.

The **Update files of fragments when opening assembly** parameter sets up the mode of verification of the actuality of files of fragments, pictures, etc., upon opening and regeneration of the assembly:

Don't update – The file will be downloaded from the server only if it is absent. This is the most optimal variant in terms of efficiency but there is no synchronization during the joint work.

Update – All files that lack actuality are automatically synchronized. This variant works more slowly but it always guarantees the actuality if something has changed on the server.

Check and ask – The actuality of all files in the assembly at all levels will be checked and the question will be asked if DOCs has files newer than they are in the work folder. This is a compromise variant.

Update product structure automatically in case of document saving. If the flag is set, the command for saving product structure into DOCs is automatically called upon file saving.

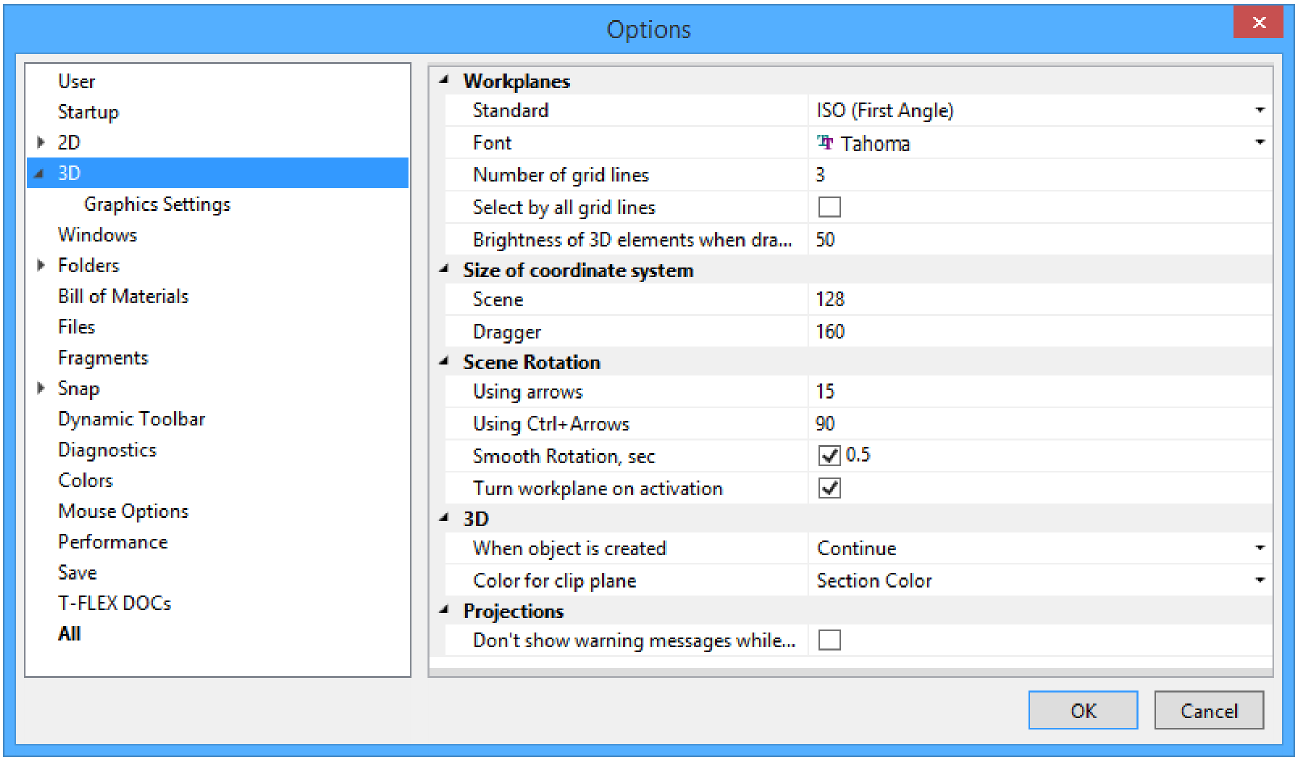
Integration rule. The parameter allows to select a rule from the T-FLEX DOCs "Application integration rules".

Integration rule can be selected only after the integration is done, since integration rules are stored in the DOCs.

Therefore, you need to select integration type and press **[OK]**. After that you should wait for the integration to happen, reopen dialog using the **SO: Set System Options** command and select one of the integration rules from the list.

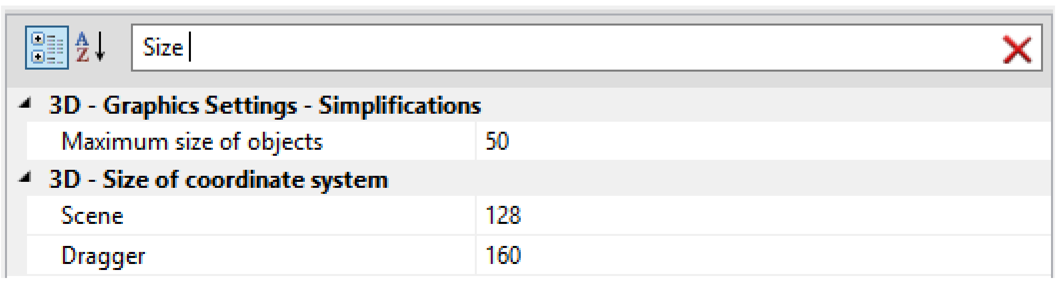
"All" Tab

All existing parameters are displayed on the **All** tab.



The parameters can be sorted by categories or alphabet.

You can enter parameter name or part of the name into the search bar to find it.



CUSTOMIZING TOOLBARS AND KEYBOARD

To move a toolbar, depress over the area of the toolbar free of buttons, and drag the toolbar to the desired location. While being dragged, the toolbar may dock at any border of the main application window or remain floating over the application. In the floating state, the toolbar has a title bar with its own title.

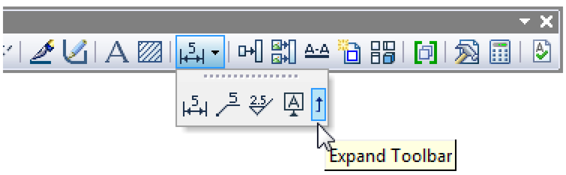
Floating toolbars can be resized. To do so, place the cursor over the toolbar window border (the cursor will assume “resize” shape), and drag the border as desired.

To switch between docked and floating state of a toolbar, double-click over the area of the toolbar that is free of buttons.

Standard toolbars ("Main" and "View") include into themselves several "embedded" toolbars. At the same time, in the main toolbar, by default, only one icon of the "embedded" toolbar (the rest are hidden) is displayed, to the right of which there is a button . Upon pressing this button the "embedded" toolbar is opened together with the remaining icons of the given group.

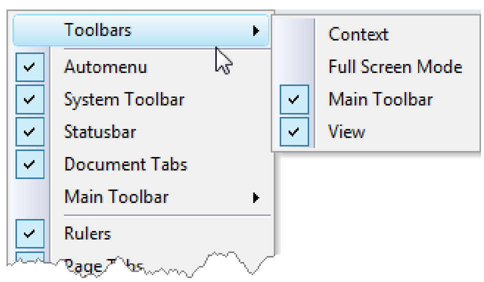
The embedded toolbar can be turned into the regular toolbar. To do this, it is necessary to place cursor into the title area of the embedded toolbar, press and, without releasing the pressed button of the mouse, drag it to any place of the T-FLEX CAD window.

The buttons of the embedded toolbar can be also placed directly on the main toolbar. To do this, it is enough to press the button at the right end of the embedded toolbar.



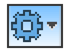
Controlling Toolbar Visibility

To show or hide the desired toolbar, one can do over any displayed toolbar. The context menu will appear with the first item containing the list of all available toolbars.



The currently visible toolbars are checkmarked. To show or hide the desired toolbar, select the respective menu item.

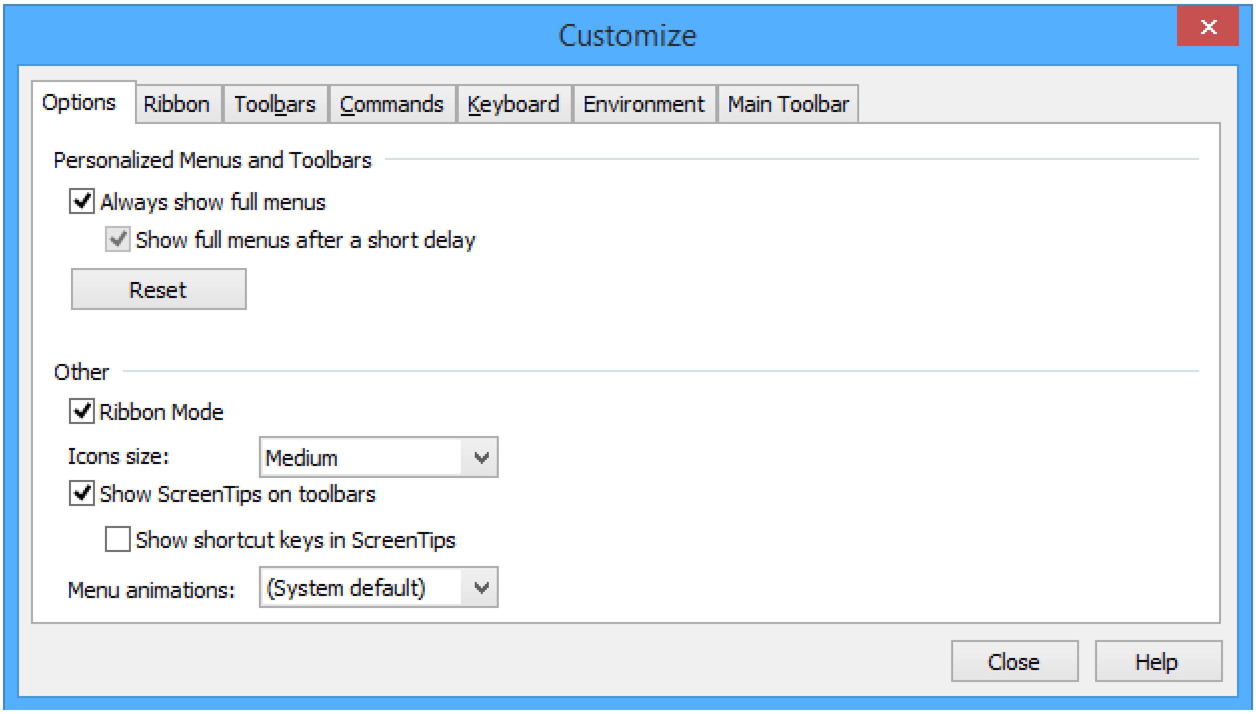
To show or hide several toolbars at once, one can use the item "Customize..." in this same menu, or the command **SB: Show Toolbars**:

Icon	Ribbon
	 → Customize
Keyboard	Textual Menu
<SB>	Customize > Customize

This command brings up a dialog box with tabs supporting various manipulations over toolbars and binding any key combinations to the application commands.

"Options" tab

On the tab **Options** there are parameters controlling the display of the textual menu and toolbars of the system.



The group **Personalized Menus and Toolbars** includes into itself parameters determining the way the textual menus are displayed:

Always show full menus. By default, this flag is turned off. In this case the T-FLEX CAD textual menu are displayed in a shortened version, hiding the menu items which have not been used for a long time. For accessing the hidden items of the textual menu, the button  found in the lower part of the menu is used.

If the flag is turned on, the textual menus are displayed to the full size.

Show full menus after a short delay. This parameter is available only when the flag “Always show full menu” is turned off. By default, this parameter is turned on. It enables to display the hidden commands in the menu just by holding the cursor over the button for some time (without pressing it).

The button **[Cancel]** cancels all changes made in the settings of the standard toolbars by the user.

The group “**Other”** combines the following parameters:

Ribbon mode. Allows to switch between ribbon and textual interfaces.

More information about interfaces can be found in “Quick start” chapter.

Icons size. Magnifying the size of the buttons in the toolbars and automenu. You can select small (16x16 pixels), medium (24x24 pixels) and large (32x32 pixels) icons from the drop-down list.

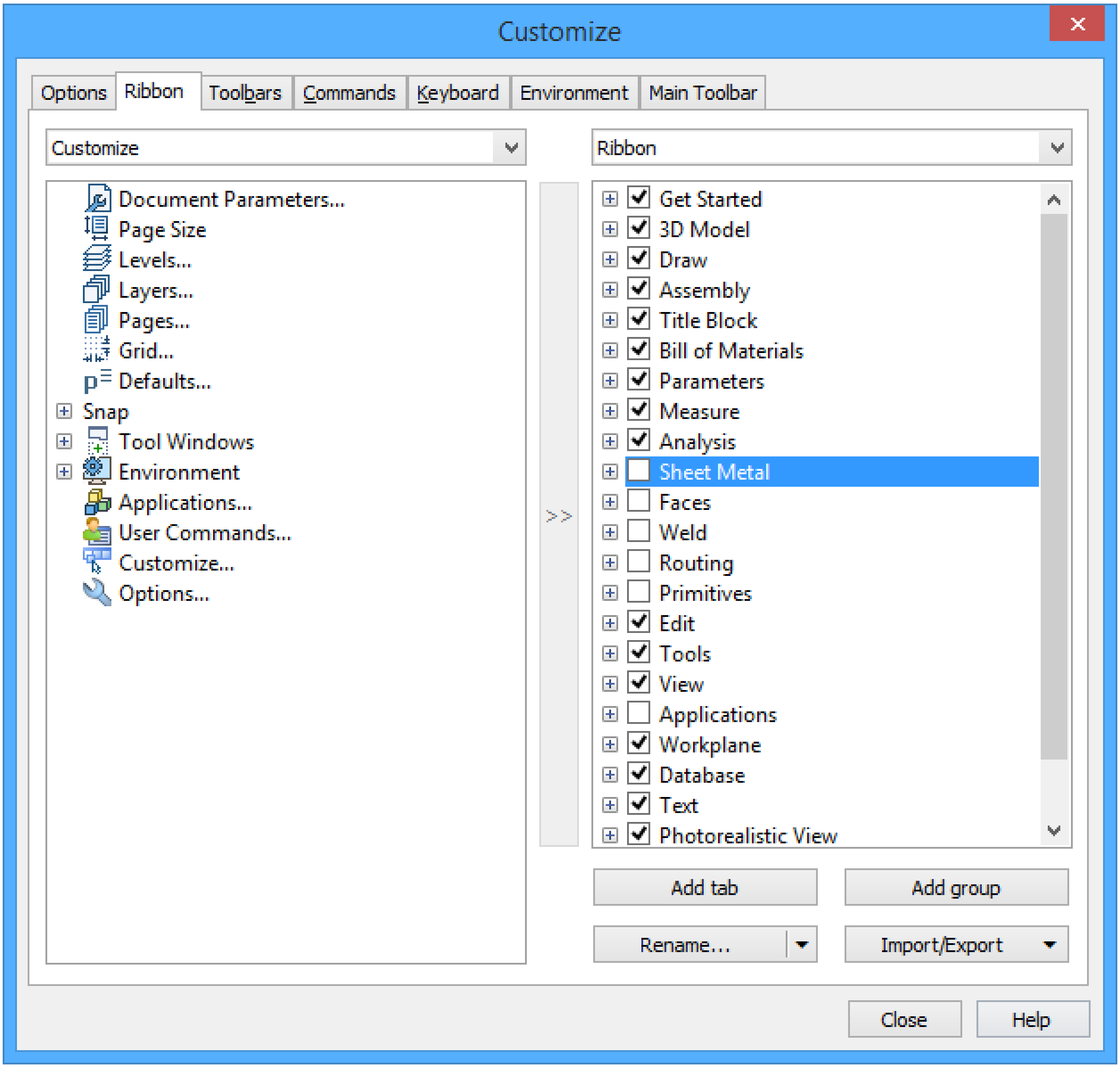
Show Tooltips on toolbars. This parameter activates the mode of displaying tooltips upon pointing with the cursor at the buttons of the toolbars.

Show shortcut keys in tooltips. This option turns on/off the mode, in which not only the name of the command but also the shortcut keys are shown in the tooltip. This option is available only when the flag “Show ScreenTips on toolbars” is on.

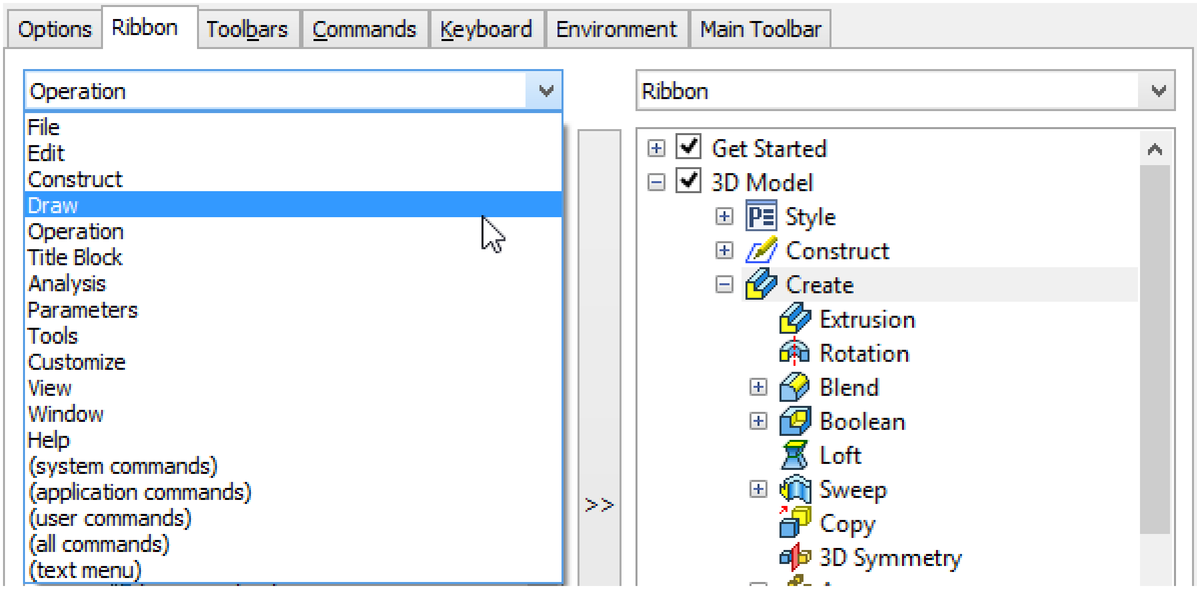
Menu animations. This drop down list defines a special effect used upon opening the textual menu: **System default** (in accordance with the general settings of the Windows), **Random**, **Unfold**, **Slide**, **Fade**, **None**.

“Ribbon” Tab

You can add commands to the **Ribbon** and **Quick access toolbar** on the **Ribbon** tab

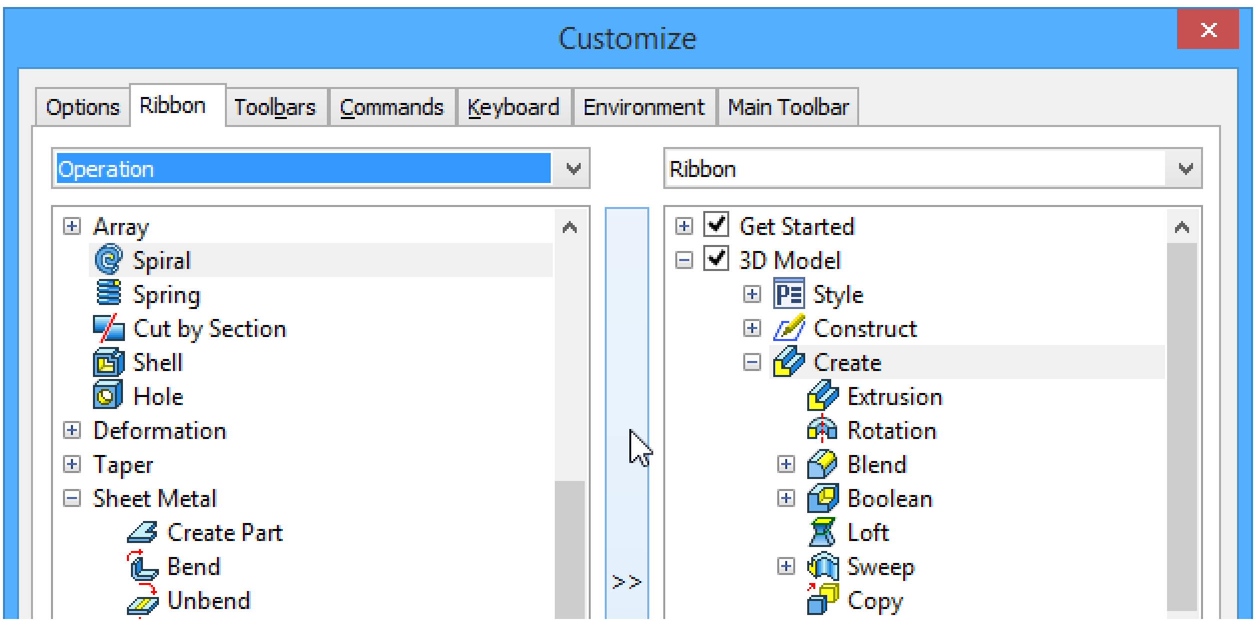


All existing commands are listed in the field to the left. You can select displayed commands using the drop-down list above the field.

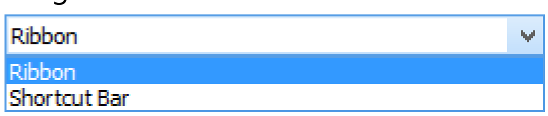


All existing ribbon tabs are listed in the right field. To add a command to the tab you need to:

- Select a command from the left field.
- Select a group on a tab in the right field.
- Press the [>>] button. The command will be added to the group.



Use the drop-down list above the right field to switch between Ribbon and Quick access list.



Add tab button allows to add a tab to the ribbon.

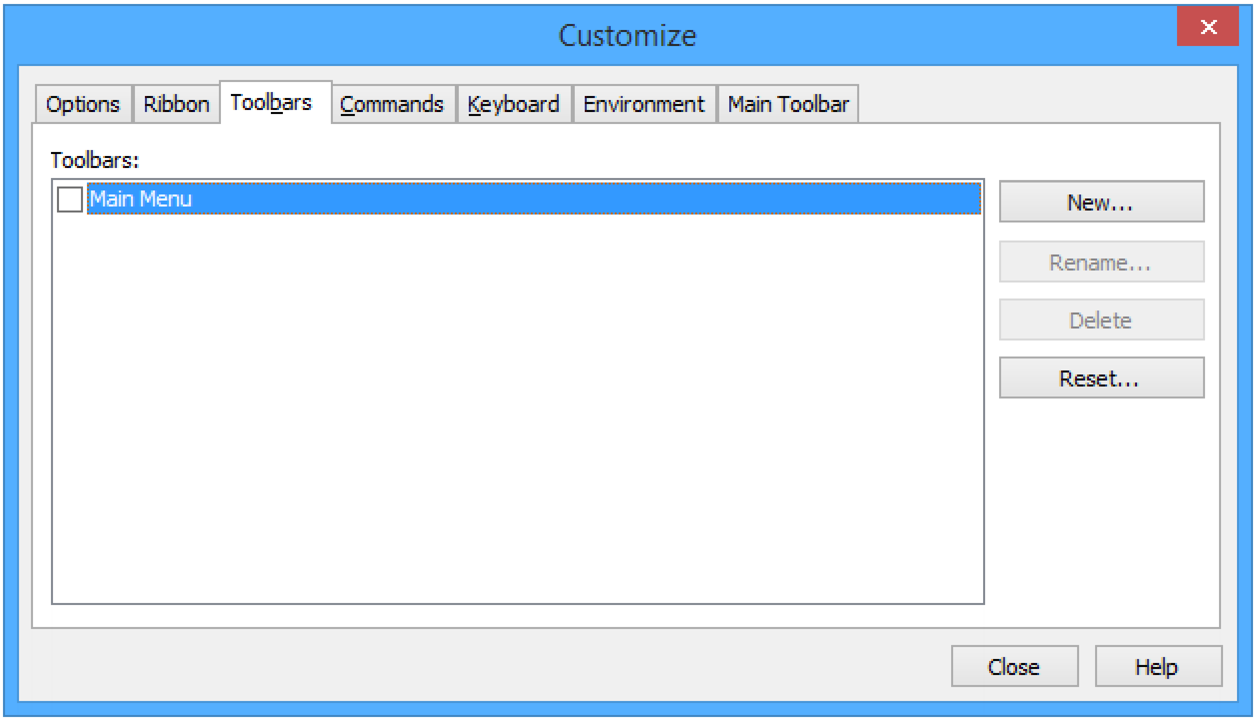
Add group button allows to create a new group on the selected tab.

Rename/Delete buttons allow to rename/delete the selected element.

Import/Export buttons allow to import/export ribbon parameters from/to the XML file.

"Toolbars" Tab

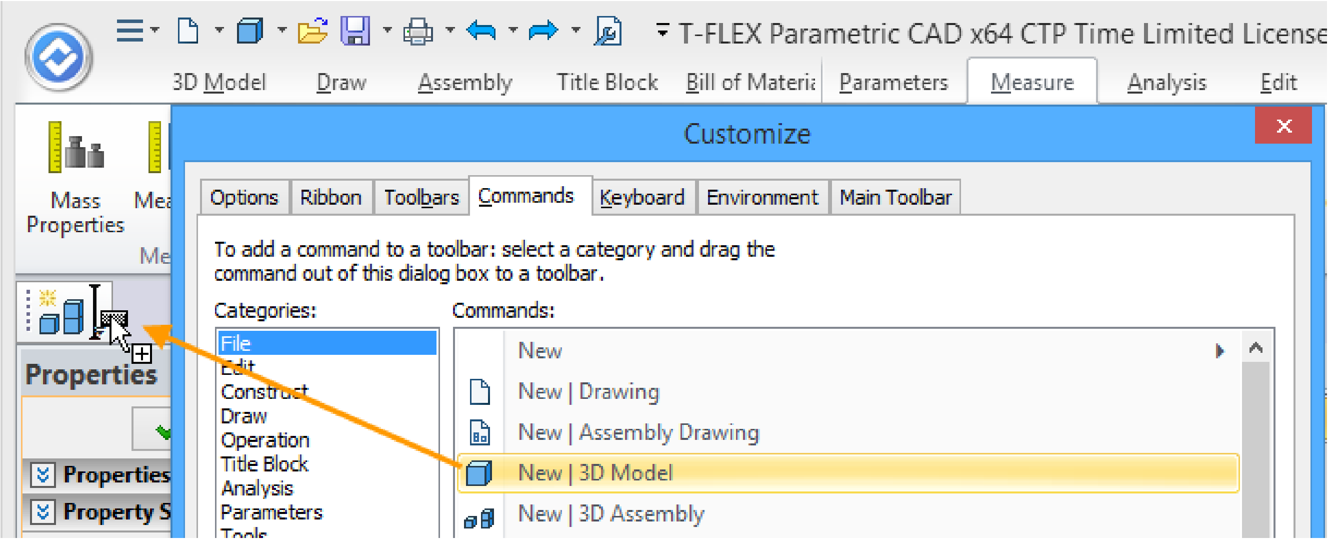
The **Toolbars** pane lists the standard (available) toolbars used in T-FLEX CAD. A toolbar visibility is controlled by setting or clearing the checkmark in the box before the name of the respective toolbar.



The buttons at the right side of the dialog box are used for creating and editing the user's own toolbars, and also for restoring the settings of the standard toolbars (canceling changes made by the user).

Creating user's toolbar. Defining toolbar name

To create a new toolbar, use the button **[New]**. In the coming up dialog box, define the new toolbar name and press **[OK]** button. A floating toolbar comes on screen. Then, open the "Commands" tab. Select the desired toolbar name from the "Toolbars" list and the button from the "Buttons" area, and drag the button into the newly created toolbar using .



The name of the user's toolbar can be also modified after creating the toolbar. To do this, it is enough to select a desired toolbar in the list and press the button **[Rename...]**. This makes the window for assigning the name of the given toolbar appear on the screen again.

Creating new toolbars can be also carried out on the tab **Main toolbar** (see below).

Changing toolbar content

While in the command **Show Toolbars** with the "Toolbars" or "Commands" tab brought up, one can remove or move icon buttons from visible toolbars to other toolbars by dragging with the cursor, using . Separators can be inserted to and removed from toolbars. This is done by selecting a button and dragging it a bit aside. The same action can be done outside any command, by additionally holding down the <Alt> key.

Any icon button on a visible toolbar can be duplicated by dragging it with the cursor, using while holding down the <Ctrl>+<Alt> combination, to any other toolbar or to an isolate location, thus creating a new floating toolbar.

Deleting toolbar

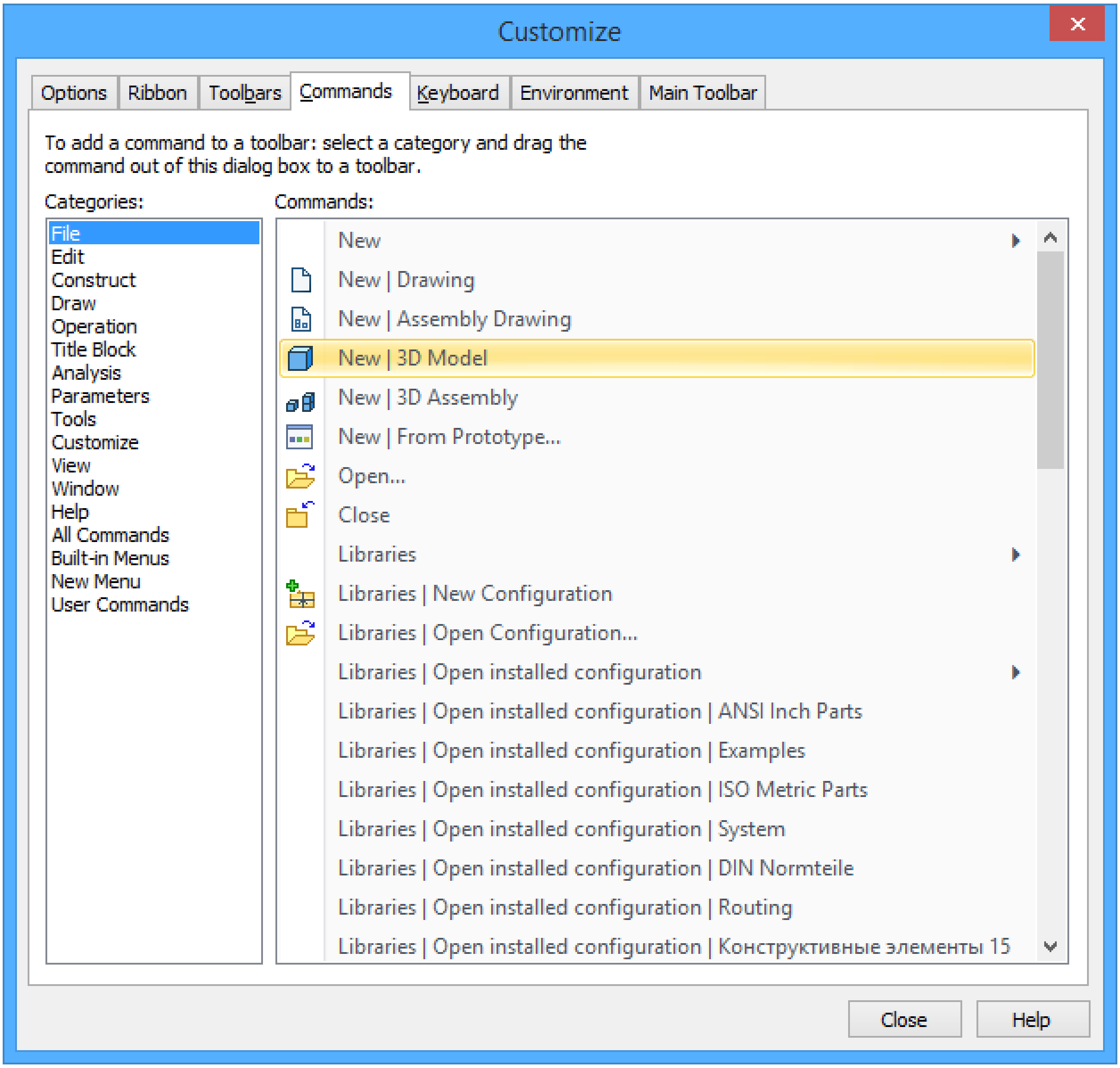
To delete a toolbar, use the **[Remove]** button after selecting the intended toolbar in the list. Note that only user-defined toolbars can be deleted. Standard toolbars can't be deleted.

Restoring content of modified toolbar

To cancel all changes, made in the standard toolbars, the button **[Cancel]** can be used. Pressing this button restores the initial state of the selected standard toolbar.

"Commands" Tab

The list "Categories:" contains the set of categories of the T-FLEX CAD commands. The sets of icon buttons in the "Commands" area are updated upon categories selection.

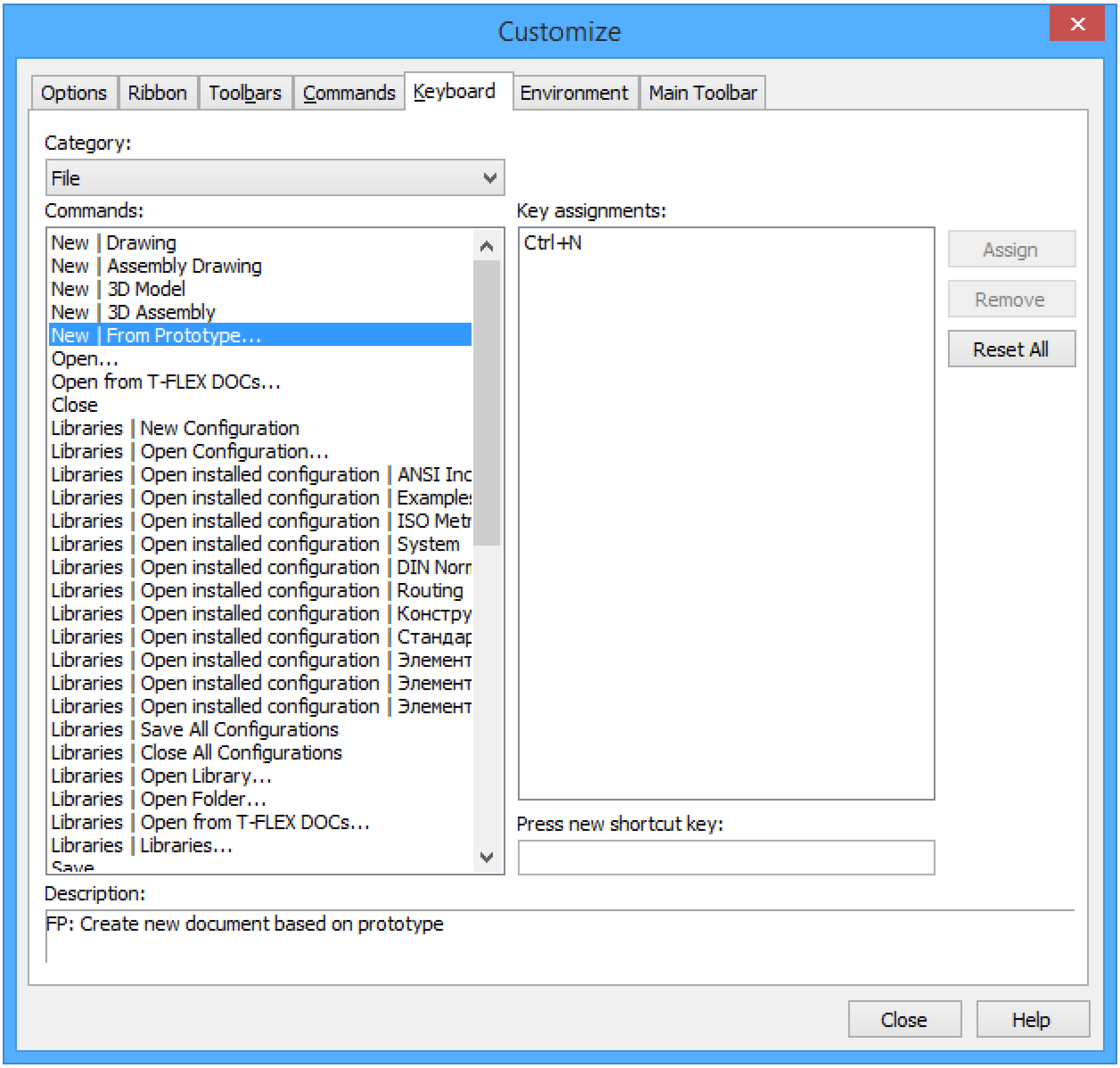


A special category "User Commands" includes into itself user's commands, defined in the dialog box of the command **Setting** > **User Commands...** (see below).

The tab "Commands:" contains the list of the commands, included into a selected category. To add a command to a toolbar, simply drag the button by the cursor, using , to the desired toolbar.

"Keyboard" Tab

This tab defines control key combination bindings of the system commands. The tab provides for creating new key combinations, assigning them to commands, deleting previously assigned bindings, and resetting (restoring the original system settings) of all key combinations.



The drop down list "Category:" enables to choose a category which contains the command being edited. The commands, included into the selected category, are shown in the list "Commands:".

The "Keys Assignments" pane displays all control keys for each particular command currently selected in the "Commands" list. If the pane stays empty upon selecting a command, no key combination is assigned to this command.

To assign a control key combination to a command, enter the new combination in the "Press new Key" input box.

Creating a new control key combination

1. Choose the commands' category in the list **Category:**.

2. Select the desired command in the **Commands** list.

3. Place the cursor in the "Press new Key" input box and click .

4. Set the new key combination using the keyboard, for example, simultaneously press <Ctrl><Alt><F12>. The input box will read, "`Ctrl+Alt+F12`". If a wrong combination is entered, simply enter the right combination over again, without trying to delete anything.

5. Press the **[Assign]** button. The new combination will then appear in the current keys list.

Several control key combinations can be assigned to the same command.

Deleting control key combination

1. In the list **Category:** choose the commands' category.

2. Select the command in the **Commands** list whose control key binding is to be deleted.

3. Select the intended key combination in the **Keys Assignments** list.

4. Press the **[Remove]** button.

Reset all key combinations to the original state

To delete all manually assigned key combinations, simply press the **[Reset All]** button. All key bindings will then be restored to the original state, the way they were assigned at the initial system installation.

"Environment" Tab

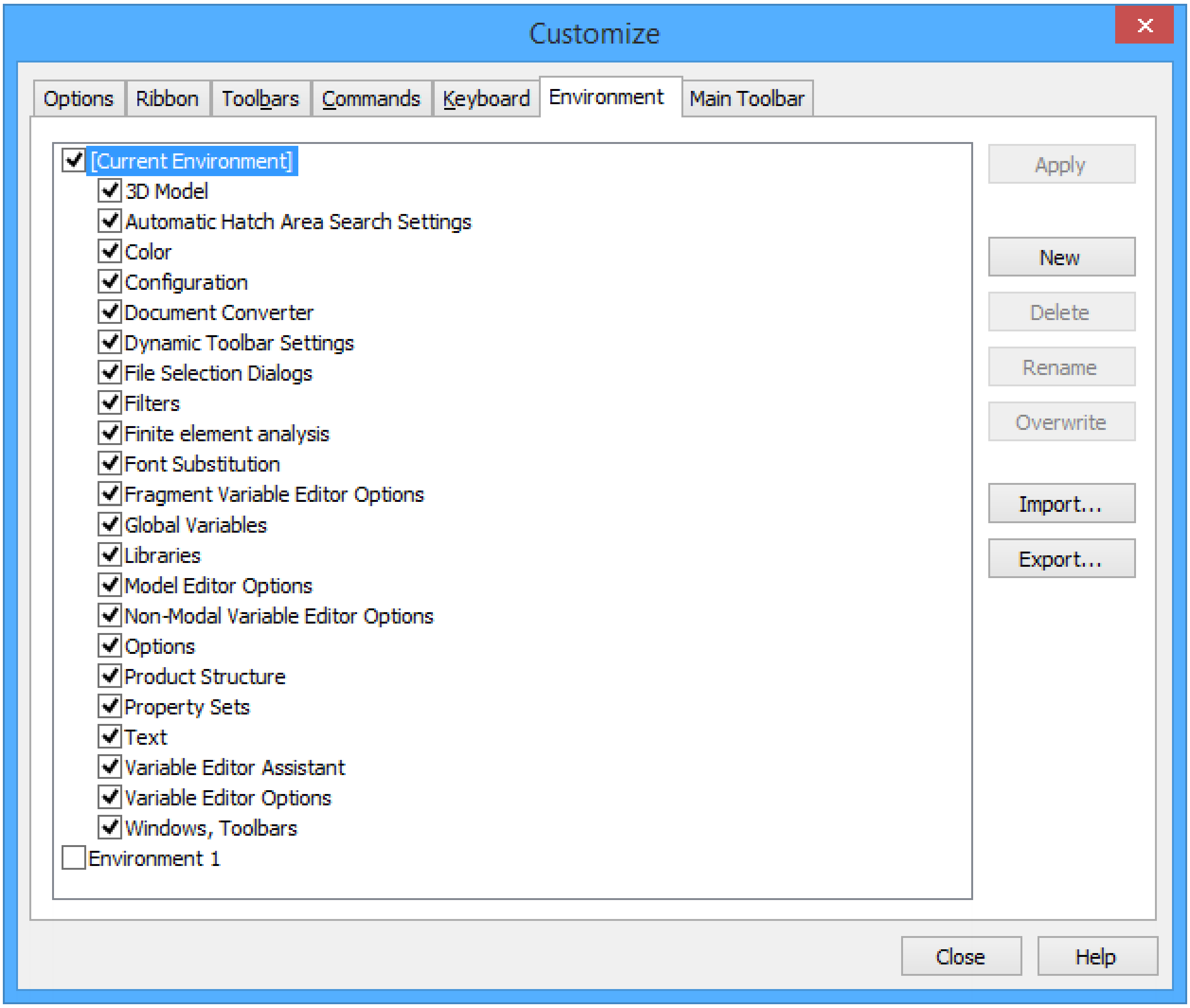
This tab serves to work with Environments.

Environment is a set of system settings that can be saved in the Windows Registry or in an external file with the purpose of a future quick system setup. An environment may include the following settings: the visibility and composition of toolbars, special system windows, library and folder windows; the system settings, default customizations of various commands, etc.

The list of existing Environments is displayed in the left part of the dialog. Initially, the only item in the list is "Current Environment". It denotes the current system settings. In the future, all user-created Environments are added here.

Already existing Environments can be renamed or deleted. To do that, select them in the list (using) and click the respective button (**[Rename]** or **[Delete]**).

To apply an existing environment, select it in the list and click **[Apply]**.



You can create a new Environment either based on the current system settings ("Current Environment"), or based on another existing Environment.

To create a new Environment, you need to:

- Select and Environment in the list, whose settings need to be copied to the new Environment. You can also select the "Current Environment" for this purpose – in this case, the current system settings will be saved in the newly created Environment;
- Once an Environment is selected, you will see the list of setting groups stored in it. By default, all parameter groups are marked with checks in the list. That means, all of that will be copied to the new Environment. To avoid copying some settings, clear the checks before the respective group names;

> The list of setting groups is closed upon another click on the selected Environment. The changes made in it (changes in the setting groups selection) will be remembered until selecting another Environment in the list. To close the list, you can also use <Left>, <Right>.

– Click the button **[New]**. A new Environment with the standard name "Environment 1 (2, 3, …)" will be created. Right after the Environment creation, the system goes into its name editing mode. You can cancel that by clicking outside the Environment name input field. You will be able to edit an Environment name in the future using the button **[Rename]**..

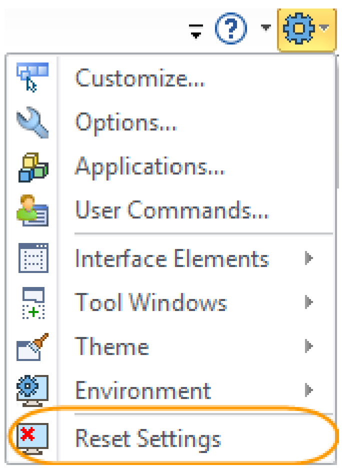
An existing Environment can be altered by overwriting the settings stored in it by the current ones. To do that, select it in the Environments list, mark the groups of settings in its settings list, which you need to replace, and click **[Overwrite]**. Please note that this button is unavailable for the “Current Environment” item.

To save an existing Environment in an external file, use the button **[Export…]**. Upon clicking the button, a standard file-saving window appears. The file name default is the same as the Environment name. The file extension is – "*.2Denv*" for the 2D system version, "*.3Denv*" for the 3D system version.

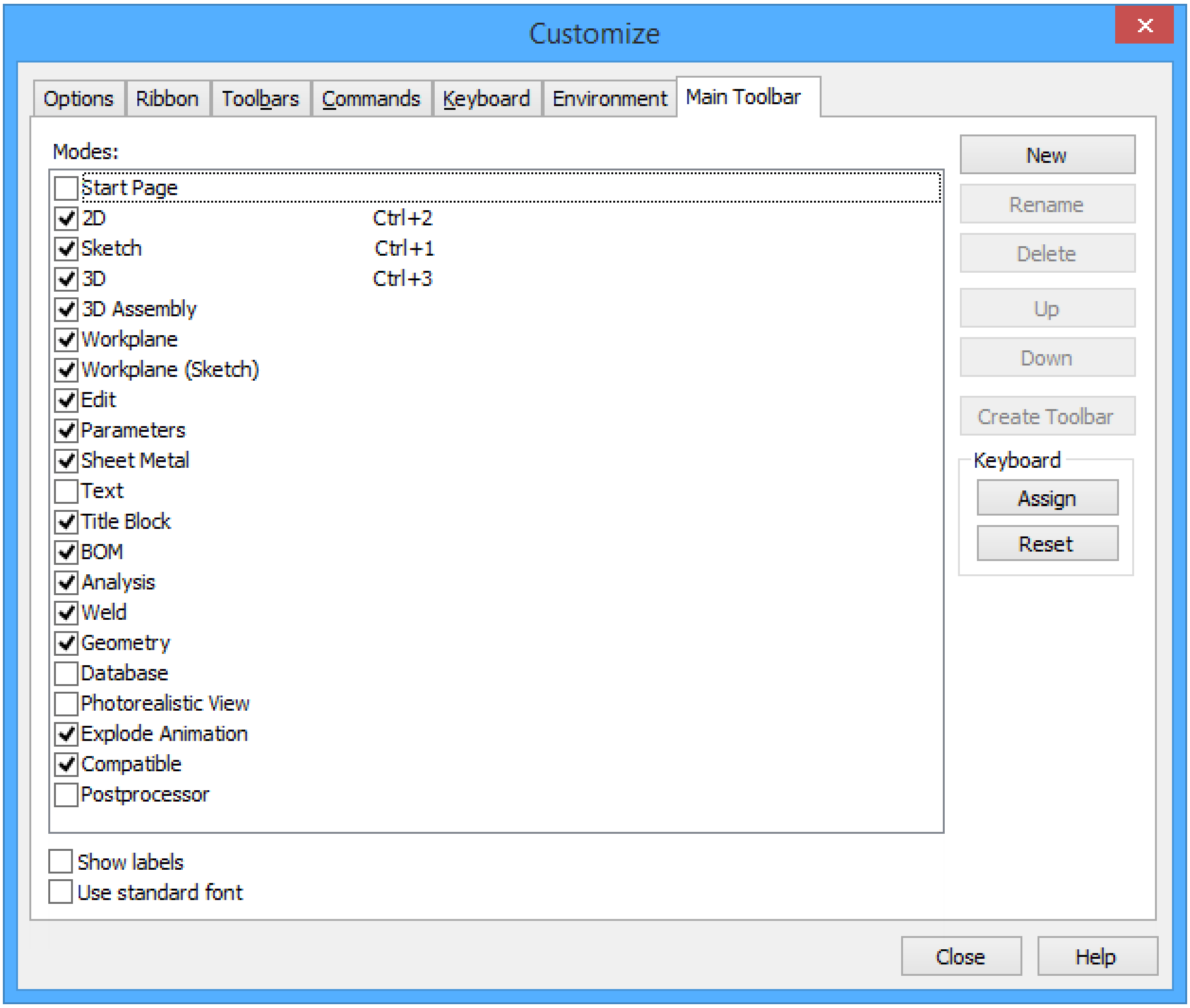
To import an Environment from an external file, use the button **[Import…]**. Upon clicking it, the standard file-opening dialog appears, in which you need to select an Environment file. The Environment imported from the file appears in the Environments list. Besides that, upon importing an Environment the system will offer to apply it.

> The Environment files created in the 2D system version will not be recognized in the 3D version, and vice versa.

The buttons **[Apply]**, **[Delete]**, **[Rename]**, **[Overwrite]** are inaccessible for the “Current Environment” item. But the latter is provided the **Reset Settings** command, using which replaces the current settings by the default ones.



"Main toolbar" tab



This tab displays the list of all main toolbar sets that are defined in the system. The sets marked in the list by the tick are displayed on the toolbar, unmarked sets are hidden. It is possible to control visibility of the sets independently, by removing/putting the tick (with the help of) located next to the name of the corresponding set.

Double pressing on the name of any set activates the set on the main toolbar.

The button **[Rename]** enables to assign another name to any set of the main toolbar. After pressing this button, the window for assigning the new name of the set appears.

Buttons **[Up]** and **[Down]** allow changing the order in which the sets follow in the list.

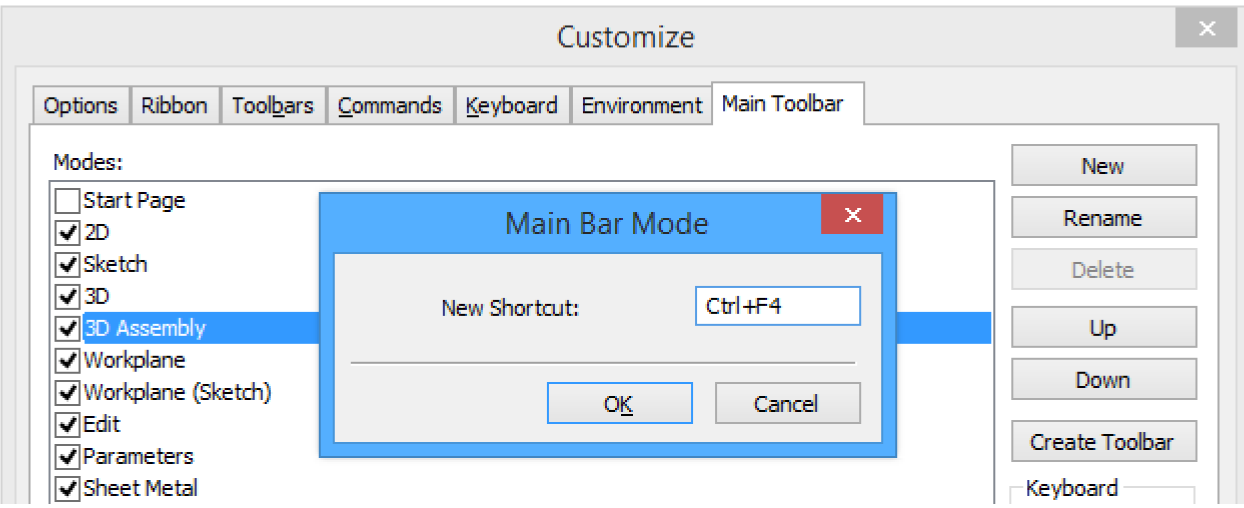
To add the new (user's) set to the main toolbar the button **[New]** is used. After pressing this button the system prompts to assign the name of the set being created. Also, the prompt to add the standard buttons (i.e. related to command used for working with new documents, files etc.) to the new set will appear on the screen. In case of negative answer, an empty set is created, in case of positive one, the set containing only the standard buttons is created. Other buttons can be added to the created set via the tab “Commands” (by using “drag&drop” method).

It is possible to remove the previously created user's set by selecting it in the list and pressing the button **[Remove]** (for standard sets this button is not available).

The button **[Create toolbar]** enables to create a single toolbar on the basis of the set selected in the list. Upon creating the toolbar, it is possible to automatically remove the standard buttons from it (by default, these buttons are present practically in all standard sets of the main toolbar).

To make the work more convenient, several sets of main toolbar commands were assigned the key combinations for quick call with the help of the keyboard (by default, these sets are “2D”, “Sketch”, “3D”). If necessary, the key combinations can be also assigned for other sets (including user's defined ones). To do it, the buttons **[Assign]** and **[Reset]** in the group “Keyboard” are used.

After pressing the button **[Assign]** the window opens up in which the required key combination for the selected set has to be specified. For specifying the combination, it is just enough to press the corresponding buttons simultaneously. If the combination is specified incorrectly, it is sufficient to press the correct combination again. After closing the window with the help of **[OK]** the specified key combination will be displayed in the list to the right of the name of the corresponding set.



To modify already defined key combination for some set, it is enough to press the button **[Assign]** and specify new combination.

For removing the assigned key combination without specifying a new one, it is necessary to press the button **[Reset]** after selecting a set in the list.

The **Show labels** flag controls how the labels to the buttons of the main toolbar are displayed. By default, the flag is disabled, i.e., the labels are not shown.

> The “Show Labels” flag has a similar action in the context menu that is invoked with the help of in the domain of the automenu or any toolbar. These two flags (“Show Labels” in the **SB: Show**

> **Toolbars** command and "Show Labels" in the context menu of toolbars) work in a synchronized manner.

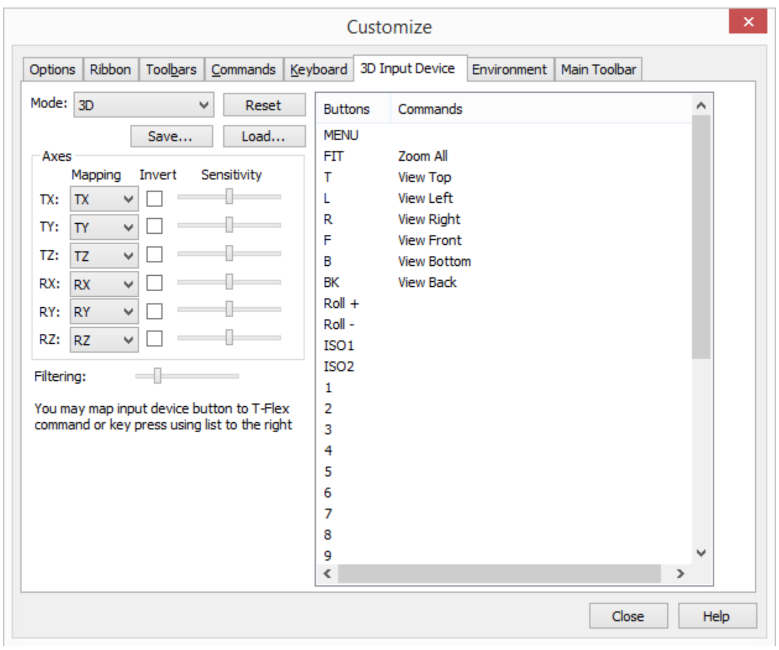
Additional flag called **Use standard font** affects how the labels to the buttons of the main toolbar are displayed when working in Windows Vista. If in the settings of Windows Vista the large font size is selected and the "Use standard font" flag is disabled, the labels to the buttons of the main toolbar will be displayed larger than the labels in the text menu. If the flag is enabled, the font size on the main toolbar is the same as that in the text menu.

"3D Input Device" Tab

T-FLEX CAD supports three-dimensional multifunctional manipulators compatible with the standards Spaceball and SpaceMouse (for example, the products of 3Dconnexion, a Logitech Company, http://www.logicad3d.com). The 3D manipulators allow panning, spinning and zooming objects in the 3D window.

Once such a device is attached, the **SB: Show Toolbars** dialog box gains another tab, "3D Input device".

This tab is used for setting up 3D device working environment. The user can set up device operating options in three modes: for working with 2D drawings, for working in the 3D model window, and for working in the 3D model window in the active workplane mode. Selection of a working mode to set up is done in the pull-down list of the parameter **View mode**.



The user can change:

- Mapping of the device axes on those of the T-FLEX CAD global coordinate system; direction of action (translation or rotation) along each axis; sensitivity of the device to impacts for each type of input (the **Axes** group). The device sensitivity is adjusted by the appropriate sliders. The more a slider shifted to the right the less impact is required on the device for translating/rotating the model along this axis;
- The level of filtration from accidental device disturbance (the **Filtering** parameter). The slider position defines the filtration coefficient in the range from 0 (all impacts handled) in the left-most position to 100% (only strongest impacts are handled) in the right-most position.

  With simultaneous impacts on several axes of the 3D device, the axis of the maximum impact is determined by the system. This impact is considered primary and is handled always. Other impacts are handled only when they exceed the product of the maximum impact and the filtration coefficient;
- Mapping of the device buttons. Each device button can be mapped on a T-FLEX CAD command (the dialog button ), or a keyboard key press (the button ). To cancel a mapping, use the  button.

The buttons **[Reset]**, **[Save...]**, **[Load...]** allow respectively resetting the changes to default settings, saving the new settings in an external file (*"*.t3d"*) and load device settings from an external.

Saving User Settings. Environments

Settings made in the commands **SO: Set System Options** and **SB: Show Toolbars**, as well as in some other commands, can be saved using the Environments mechanism. Created Environments are used to quickly set up the system.

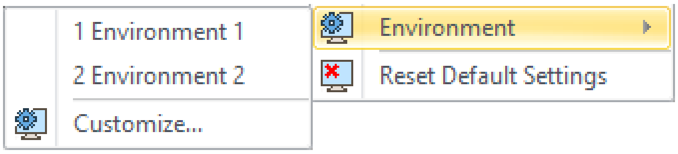
Environment is a set of system settings that can be saved in the Windows Registry: the visibility and composition of toolbars, special system windows, library and folder windows, the system settings, default customizations of various commands. An Environment may also include the settings made in various system windows (variables editor, databases, etc.) and, as was already mentioned, all settings defined in the commands **SB: Show Toolbars** and **SO: Set System Options**.

Environments are created by the user. Created Environments can be edited and deleted.

By using Environments you can quickly set up the system in a desired way by simply applying an earlier saved Environment. You can create an unlimited number of Environments. In this way, for example, you can arrange convenient working of several users on one work seat: save each user's Environment with one's personal settings and apply it when needed. Besides that, Environments can be stored in external files and, therefore, be loaded from external files. This allows porting T-FLEX CAD settings from one work seat to another one.

The main work with Environments (creating, editing, deleting, applying, export/import from an external file) is done in the **SB: Show Toolbars** command dialog on the "Environment" tab. This was described earlier in this chapter, the section “Customize Toolbars and Keyboards”, the topic “'Environment' Tab”. Here we will describe the method to quickly apply already created Environments.

The Environments existing in the system can be quickly loaded using the textual menu **Customize** > **Environment**. All Environments existing in the system are added there automatically.



To use any of the Environments, you just need to go into the said menu and select the desired Environment from the list.

Customize > **Environment** > **Customize…** command serves to quickly open the **SB: Show Toolbars** command dialog on the “Environment” tab.

ADDING USER'S COMMANDS

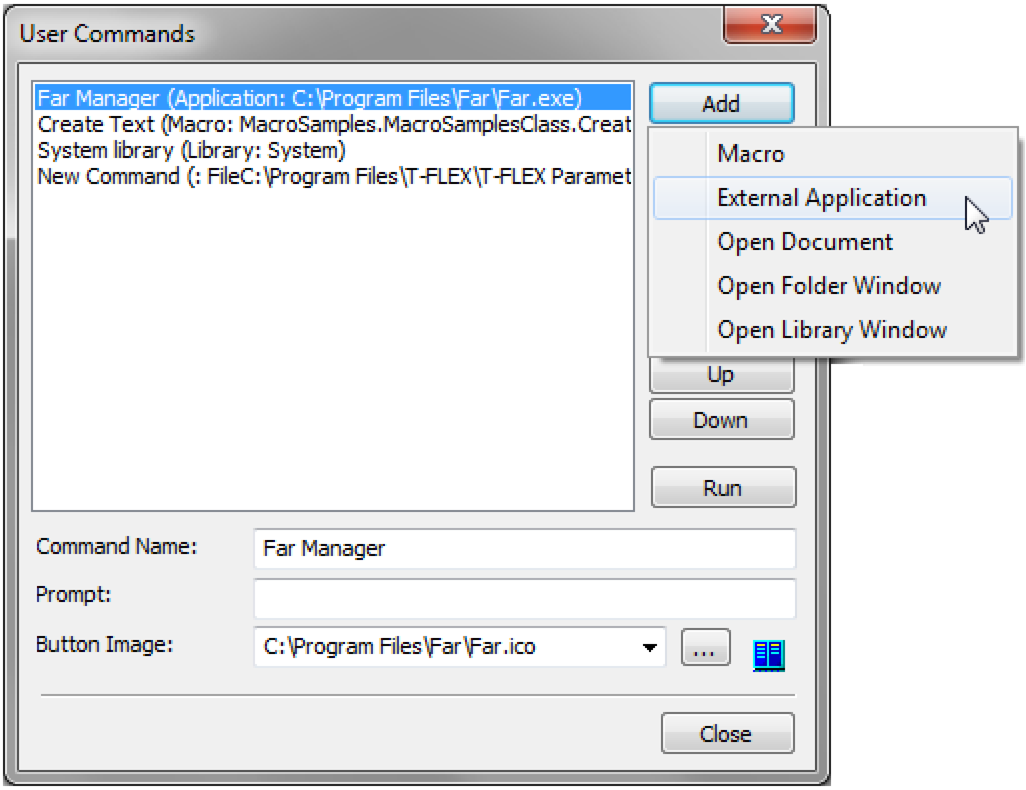
T-FLEX CAD lets a user add to the textual menu of the system or on the toolbars his own commands that enable him to start external applications and macros, open a document, a window with a folder or a library. For the added command an icon can be assigned (file *“*.ico”*).

For adding user's command the following command is used:

Keyboard	Textual Menu	Icon
-	Customize > User Commands…	

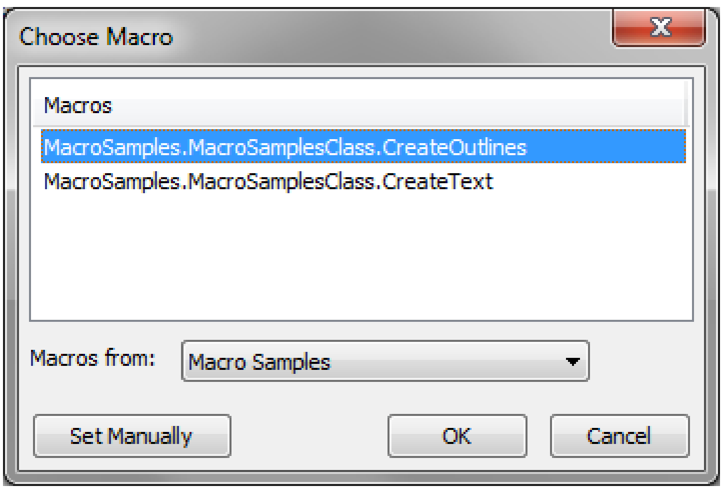
After calling this command the dialog window **User Commands** will appear on the screen. In this window the following items are defined: the command's type (what sort of operations this command will do), command call parameters, command name, a line of a tooltip for a command and its icon for displaying in the textual menu or on the toolbars of the system.

To add the command, it is necessary to press the button **[Add]**.

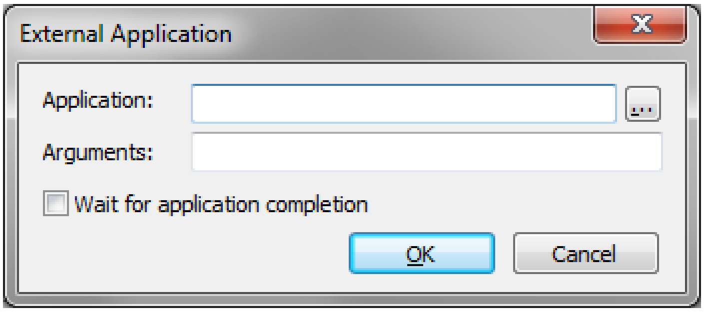


Upon pressing this button a drop down list for choosing the type of the command being added will emerge:

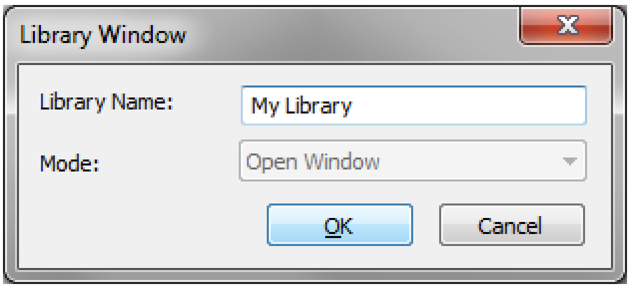
- **Macro** – adding the command for calling a macro. When this item is selected, a window of an auxiliary dialog box for choosing a macro is opened. A macro can be selected from the list or assigned manually;



- **External Application** – adding the command for calling an external application (without exiting the T-FLEX CAD). When this item is selected, a window of an auxiliary dialog box for selecting an external application and start-up parameters will emerge. An additional flag **Wait for application completion** prohibits continuation of work in the T-FLEX CAD before closing the window of external application;



– **Open Document** – adding the command for opening the T-FLEX CAD document. When this item is selected, the standard file selection dialog box will appear;
– **Open Folder Window** – adding the command for opening a certain folder with the T-FLEX CAD documents. Upon choosing this item, the standard folder selection dialog box will be opened;
– **Open Library Window** – adding the command for opening the T-FLEX CAD library. Upon choosing this option, the dialog box for assigning the library name will appear.



For each command being added it is possible to assign a name, a brief tooltip and an icon. To do this, it is necessary to choose a desired command in the list and put the required information into the fields **"Command Name"**, **"Prompt"**, **"Button Image"**.

To remove an unnecessary command, one can select it in the list and press the button **[Remove]**.

For the command selected in the list, the button **[Properties...]** calls for the same dialog box as the one used upon addition of the given command. Thus, one can change parameters of the command (for example, indicate another file for opening or another macro).

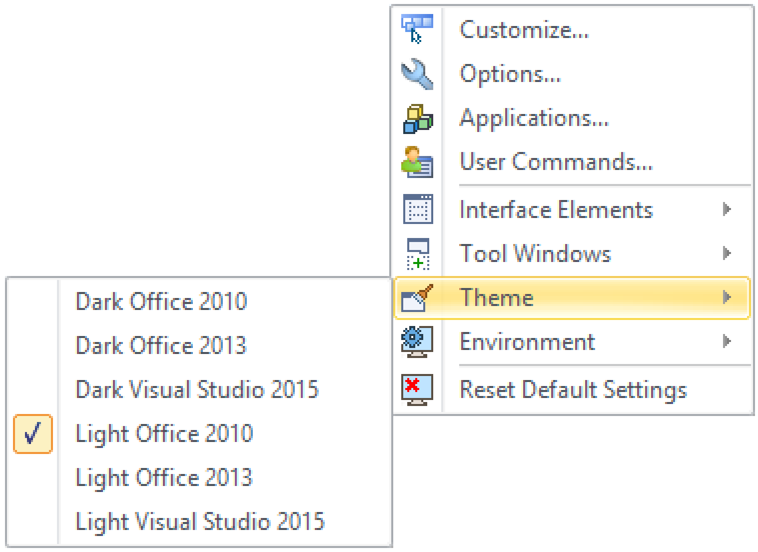
The button **[Run]** calls for the command selected at the moment in the list of the commands.

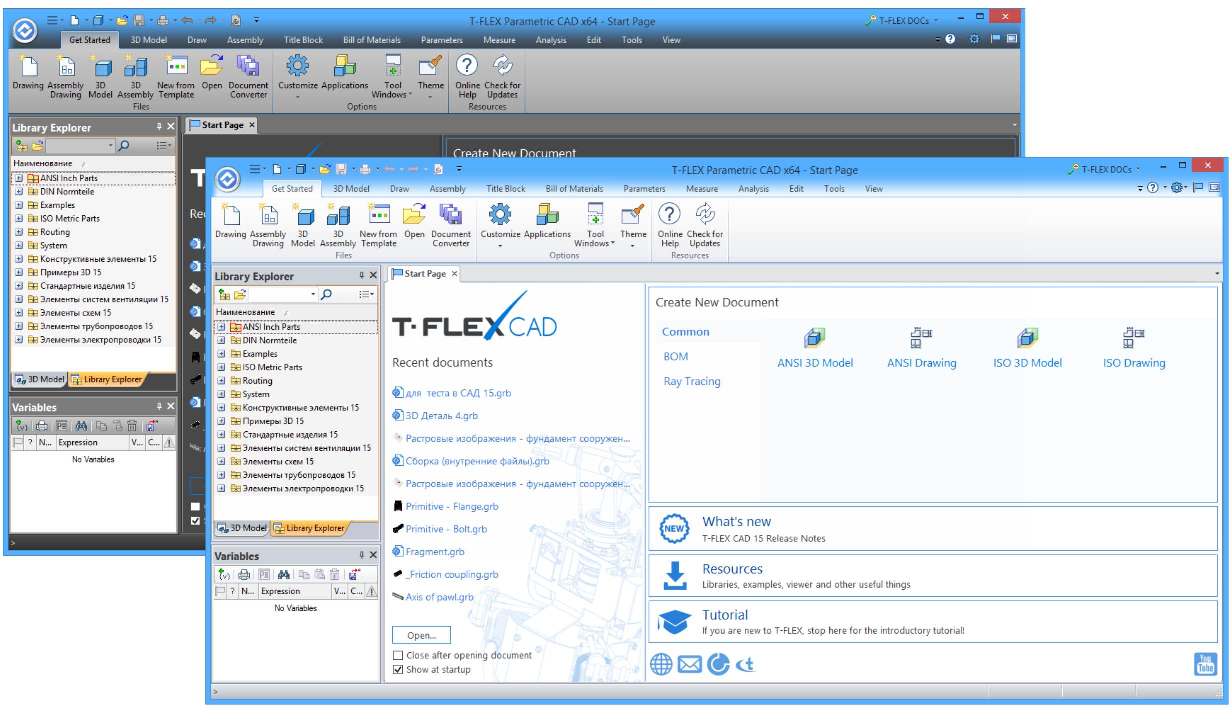
> Upon defining user's commands one should keep in mind that the commands defined in the dialog box "User Commands", by default, are not added to the textual menu or toolbars of the system. Access to these commands is possible only with the help of the button **[Run]** of the given dialog box. To simplify an access to the user's commands, it is possible to add them to the textual menu or toolbars of the system with the help of the command **SB: Show Toolbars** (tab "Commands").

The list of the user's commands can be saved in the external file "*.*tfcmd*" with the help of the button **[Export...]**. The list of the user's commands can be read from the external file with the help of the button **[Import...]**.

MAIN WINDOW DESIGN STYLES

T-FLEX CAD has a capability to modify the design style of the main window of the system. In the context menu of the main window (textual interface) and in drop-down menu the "Theme" submenu is available which allows us to select the desired style of design.





CUSTOMIZING DRAWING

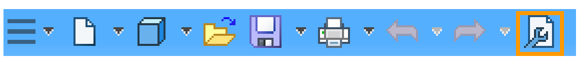
Each T-FLEX CAD drawing has its own settings. These settings include a variety of definitions of both general nature and specific to particular elements. Examples of the former include setting drawing boundaries and scale, while the latter – dimension standards, line thickness, etc. These settings can be defined not only at the beginning, but also at any moment while working on a drawing. All settings are saved with the drawing. IN the case of multi-page document, settings can be defined separately for each page. When creating a new page, its settings are copied from the currently active page.

One can create a prototype drawing with its specific settings. To do so, use the command **File > Save as Prototype…** The prototype file will be saved in the folder …"…*\AppData\Local\Top Systems\T-FLEX CAD 3D 15 x64\Eng\Templates*" (see "Getting Started"). To create a new drawing with the settings as in a prototype, use the command **File > New From Prototype**. The desired prototype file can be selected from the list. (multiple prototype files can be created.)

Any T-FLEX CAD document can be used as a prototype. Initially, the system is shipped with several prototype drawings "*.GRB". These are located in the system folder *(…\Program\Template)*. Their settings comply with several drawing standards. Depending on the choice of the particular drawing standard, new drawings created via **File > New** will assume all settings from the respective prototype. The name of the prototype drawing can be redefined using the command **SO: Set System Options**, on the **Files** tab.

CUSTOMIZING DRAWING. DIALOG OF " DOCUMENT PARAMETERS" COMMAND

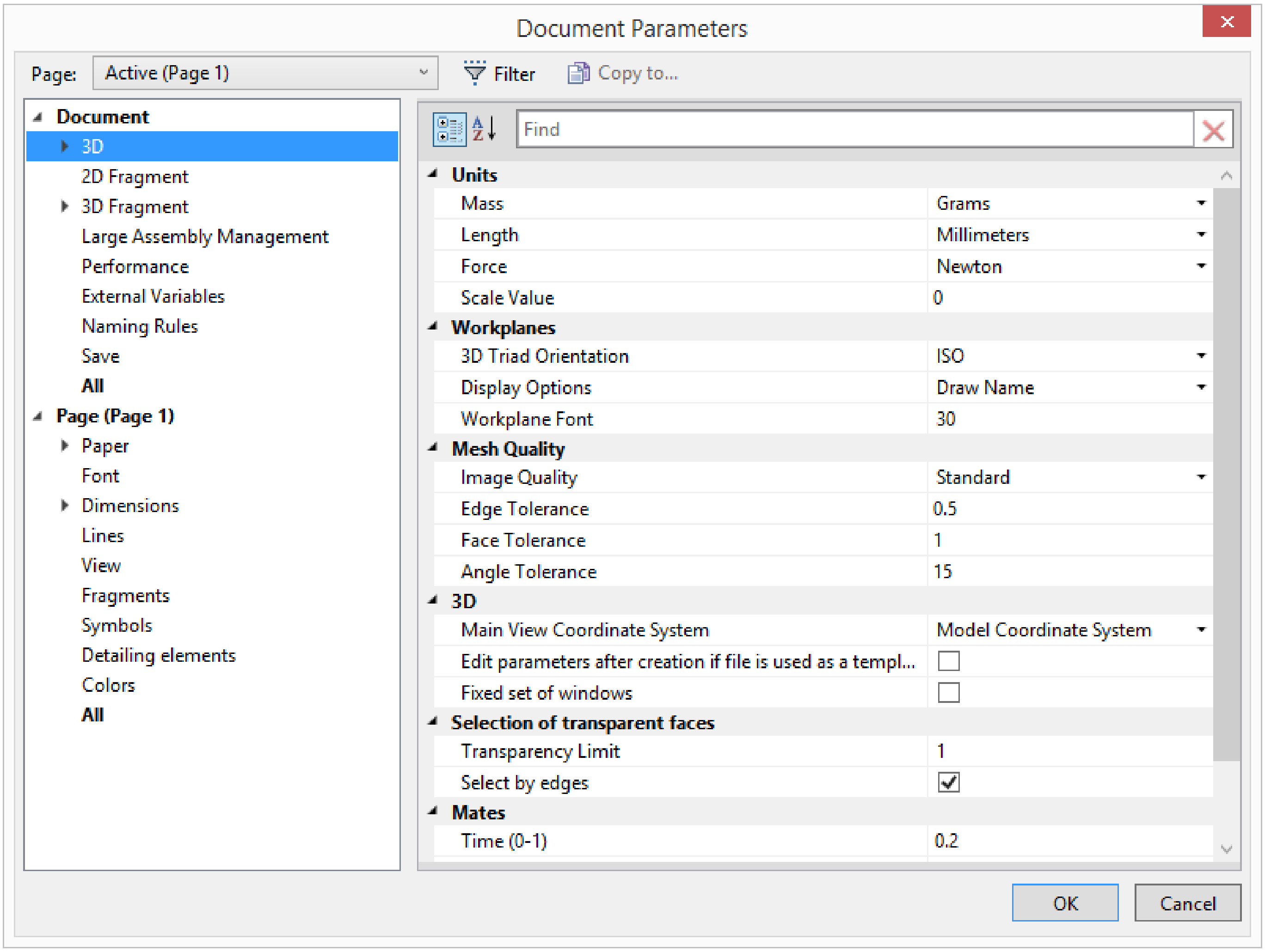
The following command is used for defining drawing parameters, **ST: Set Document Parameters.**

Icon	Ribbon
	Edit → Document → Document Parameters 
Keyboard	Textual Menu
<ST>	Customize > Document Parameters

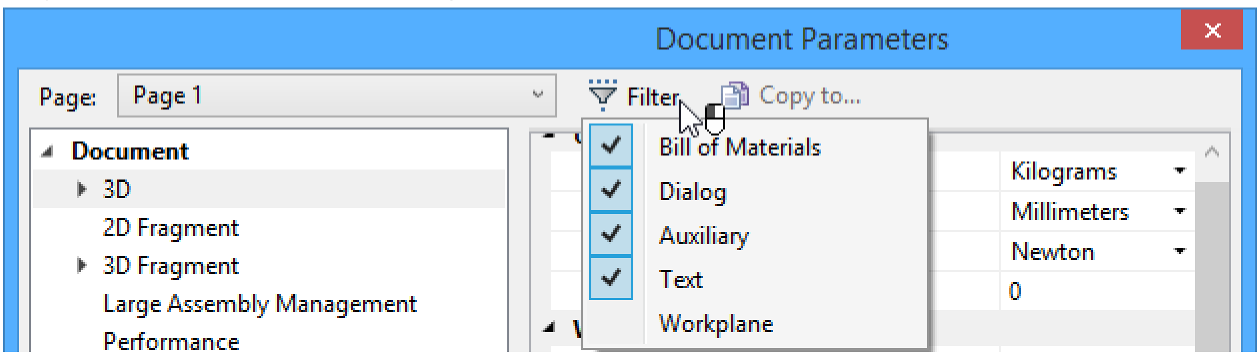
The command brings up a dialog box with tabs holding various groups of parameters.

> When you select a parameter its tip appears in the bottom part of the screen. Such tips are available for the most of the parameters.

In the **Document Parameters** window it is possible to adjust the parameters relating to a whole document or to its pages. For these purposes bookmarks **Page** and **Document** are used.



A page can be chosen in the drop-down list in the top of the window. It is possible to set displayed pages type for the drop-down list using **Filter** list:



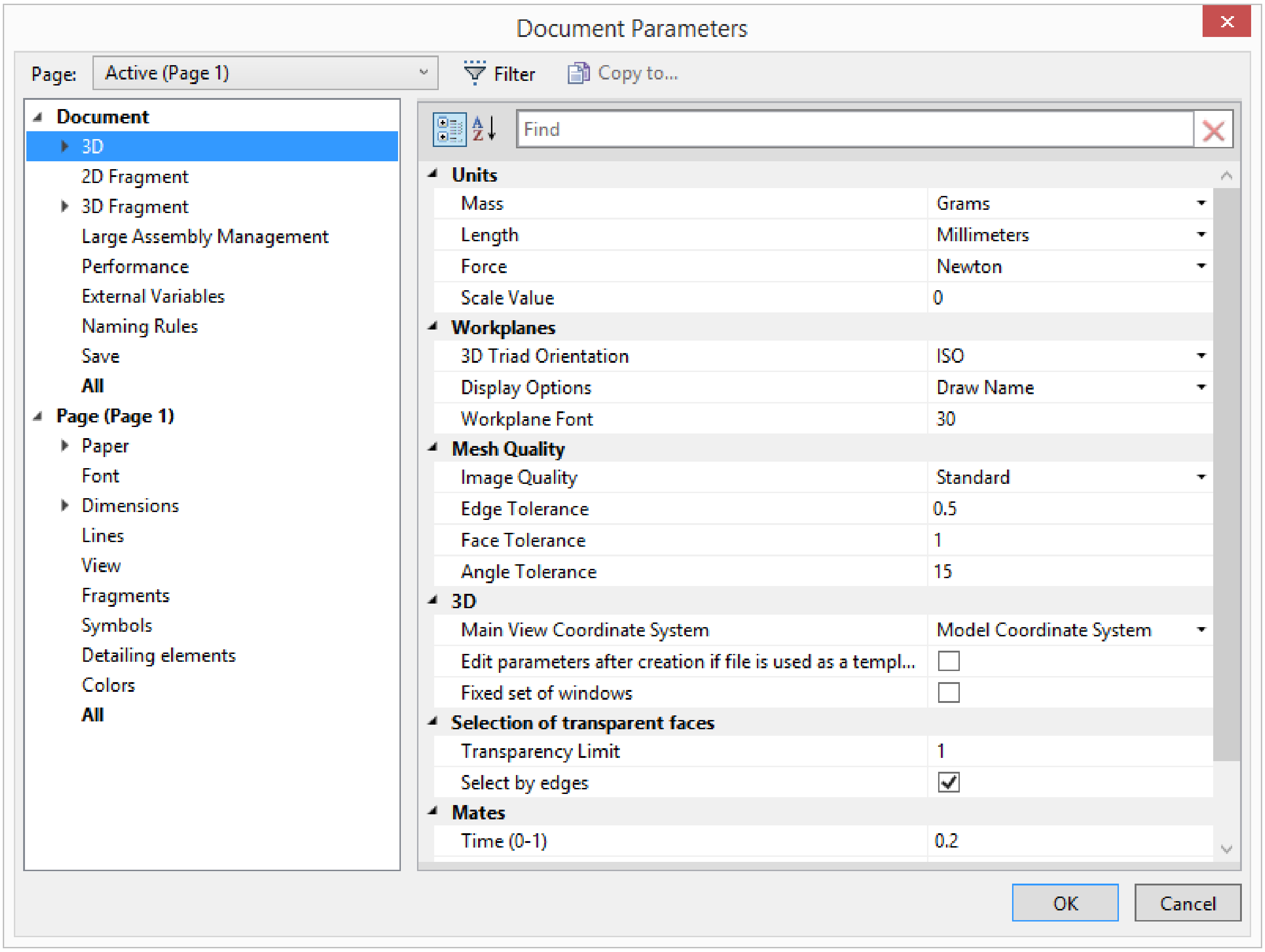
By means of the [Copy to...] button it is possible to transfer parameters of one page to another. It is possible to copy all page parameters at once or parameters from chosen bookmarks.

You can select several pages at once to copy.

"Document" Group

"3D" Tab

This tab collects general parameters related to 3D modeling:



Units section. Defines measurement units of the elements and operations on the 3D scene.

Mass. Defines mass units for solid objects of 3D scene.

Length. Defines length units for 3D objects.

Force. Defines force units for 3D objects. Used in Measure command and loads definition in Analysis studies.

Scale Value. Parameter is used when the value of Length parameter is set to "User". In this case, any arbitrary number of units per meter can be defined. 3D object will take into account this value for Length units.

The **Workplanes** group defines the mode of displaying the workplane names and types. These parameters are displayed in the upper-left corner of the respective workplanes.

3D Triad Orientation. Defines orientation 3D model views in the world coordinate system.

The value **"XY-Top"** corresponds to the choice when axes direction of the "Top view" coincides with the directions of X and Y axes of the world coordinate system.

The **"ISO"** setting defines the ISO standard orientation.

Selected orientation is used when creating projections and in the commands of 3D window viewpoint definition («Front View», «Left View», «Right View», «Axonometric View», etc.).

Display Options. Allows you to display/hide workplane types and names in 3D scene:

Don't display. Workplane name and type are not displayed.

Draw Name. Sets the mode of displaying the workplane name according to the workplane type. Note that changing the workplane type does not affect the name. The name can be changed manually under the workplane parameters.

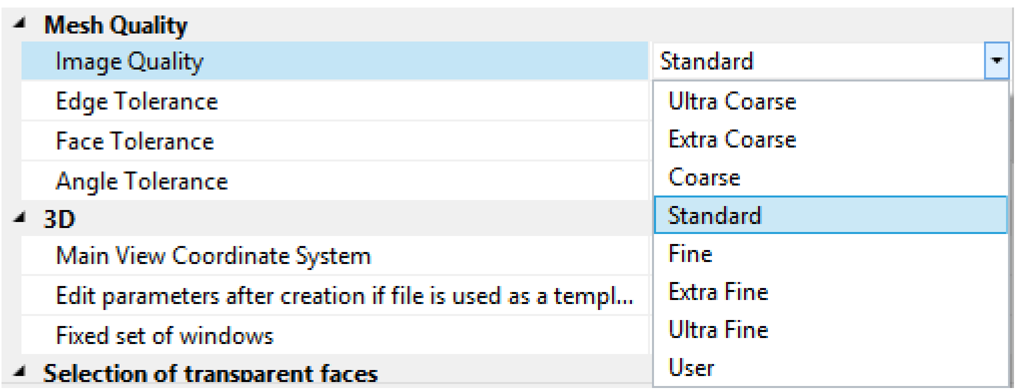
Draw Type. Sets the mode of displaying the workplane type (Front, Left, etc.).

Draw name and type. Both workplane name and type are displayed.

Workplane font. Defines font size for workplane names.

Mesh Quality:

Image Quality. This parameter allows us to specify the degree of refinement of the model mesh consisting of planar triangular faces upon displaying the image in the 3D window. Higher quality of the image of the model increases the number of planar faces which slows down the work for large models or for insufficiently powerful video cards. If possible, it is recommended to minimize the quality of the image of the model.



How many triangles are used for discretization of a specific model given a specific value of the mesh quality parameter can be determined with the help of the "Display Statistics Information" option in the "Graphics Settings" dialog. This dialog is invoked by pressing the "Graphics Settings..." button on the "3D" tab of the **Customize > Options** command.

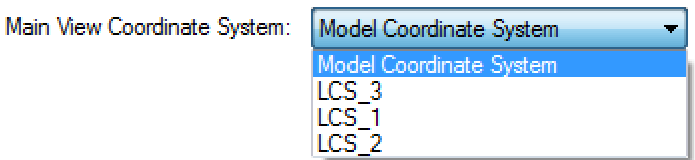
Edge Tolerance – the maximum distance between the curves of the model and edges of the mesh which approximate these curves;

Face Tolerance – the maximum distance between the surfaces of the model and planar faces of the mesh which approximate these surfaces;

Angle Tolerance – the maximum angle, specified in degrees, between the curves of the model and the edges of the mesh which approximate these curves. The angle is equal to the sum of angles between the edges of the mesh and the tangents to the original curve at the end points of the edges. This parameter also defines the maximally admissible angle between the surfaces of the model and the planar faces of the mesh which approximate these surfaces (i.e., the angle between the normals to the surface or face).

3D:

Main View Coordinate System. This parameter defines the orientation of 3D model views, i.e., location of views ("Front view", "Top view" etc.) in the world coordinate system. Its orientation is used when creating projections, as well as in the commands of 3D window view direction («Front View», «Left View», «Right View», «Axonometric View», etc.). This option can be helpful when working with imported geometry, parts created in context of assembly or other analogous cases, when model is oriented discordantly to main directions.



Local coordinate system selected as the main view coordinate system cannot be deleted.

Edit parameters after creation if file is used as template. The given parameter is also used only for the document which will become a prototype. Moreover, the document-prototype must have a set of external variables. Then, if the flag is enabled, upon creation of a new document on the basis of this prototype will immediately appear the window of the editor of external variables of the document being created.

Fixed set of windows. When the given parameter is set up, the current set of views (i.e., 2D and 3D windows) is getting fixed in the document. Upon subsequent openings of the document, modification of this set of views will not be possible. It will not be possible to delete one of the windows by reducing its size to zero (when moving the window's separator), or add a new view (the button of creation of a new view will not be available). This functionality is convenient for creation of templates with a set of views defined in advance.

The “**Selection of Transparent Faces**” group of parameters allows us to control selection of transparent faces in the 3D scene:

Transparency limit. This parameter can take values ranging from 0 to 1. It is possible to specify the transparency limit with the help of a variable. If transparency of some face of a body is higher than the indicated transparency limit, this face will be ignored when it is selected in the 3D scene.

Recall that transparency 0 corresponds to the absolutely opaque body (face), 1 – absolutely transparent. Transparency which is larger than 1 is impossible, therefore the value of the limit equal to 1 (default value) corresponds to the standard behavior of the system.

Select by Edges. If this flag is enabled, those faces whose transparency is larger than the specified limit can be selected by their edges (as in the wireframe mode). When the flag is disabled, such faces cannot be selected.

“**Mates**” group of parameters define performance of mates solver only for regenerating a 3D model.

Time (0-1). Defines the time limit of mates solver calculations. The limit is set as factor between 0 and 1 corresponding to “less” and “more” respectively.

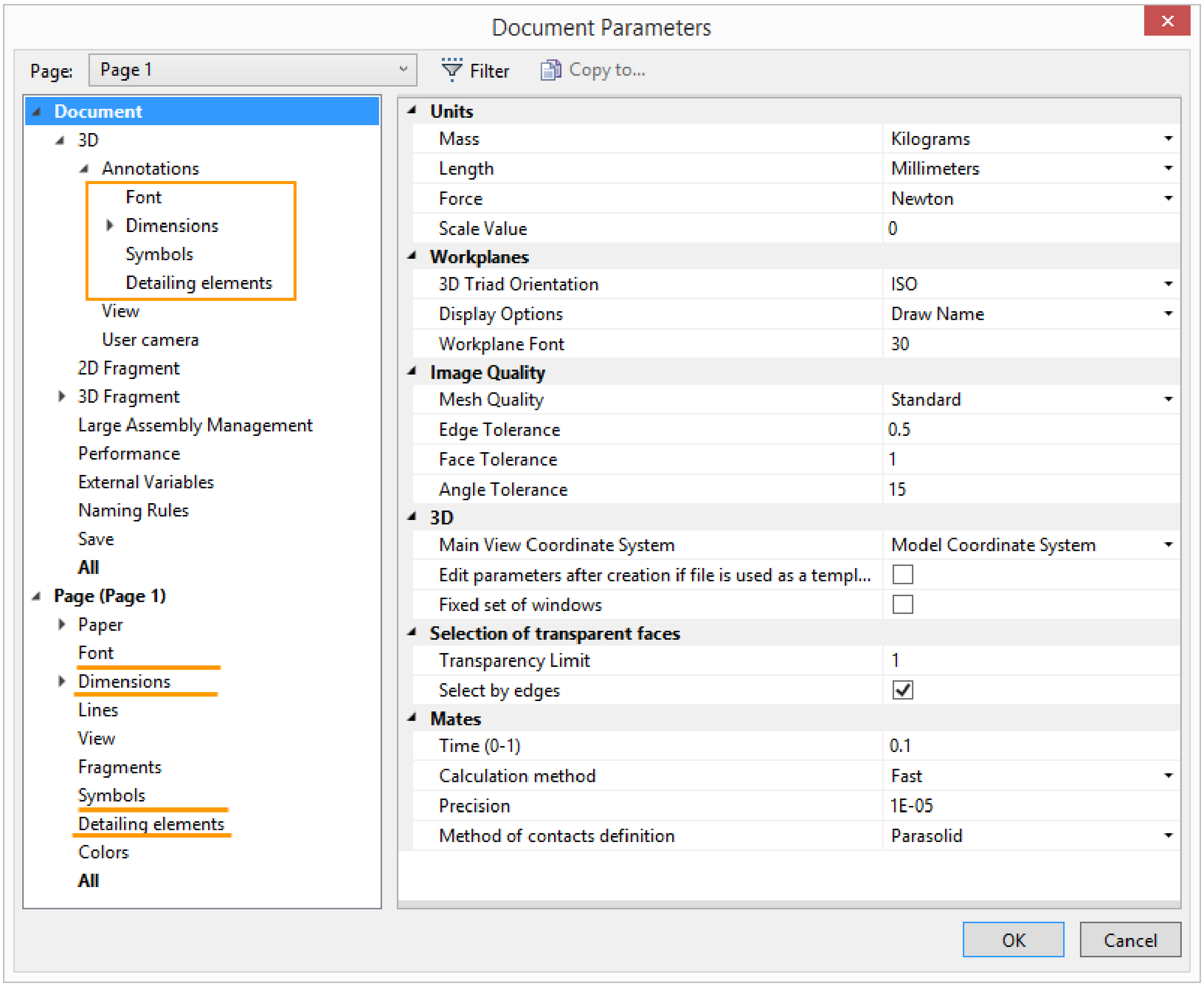
Calculation method. Sets one of two available methods of calculation – precise or fast.

More information about calculation methods can be found in “3D Assemblies - Mates” chapter.

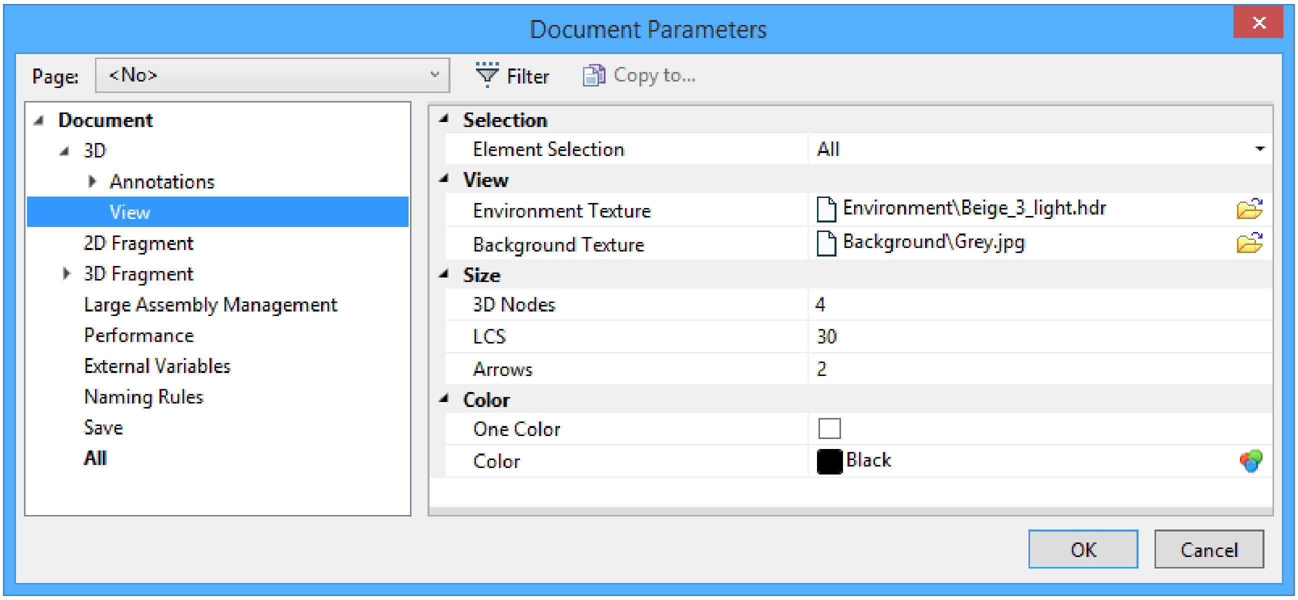
Precision. Specifies precision of mates calculations when regenerating a model. The parameter value can be specified from 0.000001 (10^{-6}) to +∞. The precision value specifies the minimum value of mates deviation. If mates deviation exceeds the specified value, all mates will be recalculated. If there are no mates, which deviation exceeds the specified value, the recalculation will not be performed. This reduces model recalculation time.

“Annotations” Tab

Font, **Dimensions**, **Alternative dimensions**, **Symbols** and **Leader notes and text** tabs are described in the “Page Group” section. The group sets parameters for elements in 3D scene.



"View" Tab



Selection:

Element selection. Defines element selection modes while in drawing and editing commands. Select one of the two modes:

All. When creating and editing elements, all existing elements will be allowed for selection.

Visible only. When creating and editing elements, only the visible elements will be allowed for selection. The element visibility is determined based on element levels and visibility intervals defined in the command **SH: Set Levels (Customize > Levels...)**, as well as layer configurations defined in the command **QL: Configure Layers (Customize > Layers...)**.

View:

Environment texture. File of the image containing the picture of the texture. This parameter allows us to virtually surround the objects of the model by a 3D image which will be displayed on the faces of the model. To specify this texture the format hdr is used.

Surrounding can be also specified in the properties of a 3D view or with the help of moving the texture file from the Windows Guide to a 3D window.

In all windows of a 3D model the same texture of the surrounding is used. In one specific window the use of the texture can be blocked by the corresponding flag in the properties dialog.

Parameters of the material such as "**Gloss**" and "**Reflection**" strongly affect visual properties of the material when using the texture of surroundings.

In case when these parameters have zero values, the surrounding texture will not affect the appearance of the model.

Background texture. File of the image containing the picture of the texture. Reference to this file can be specified in the **Document Paramers** dialog on the **3D > View** tab, in the 3D window properties dialog or by the method of moving the file of the raster image from the Windows Guide to the 3D window. This parameter allows us to use the texture as a background of the current drawing's window.

The same background texture is used for all windows of 3D model. In one specific window the use of the texture can be disabled; to do so there is a corresponding parameter (flag) in the properties of a 3D view. To specify the texture the files of the formats bmp, jpg, jpeg, gif, tga, tif, tiff, png can be used.

The **Size** group includes definitions of **3D node** size (in pixels), and the sizes of **coordinate systems** and **arrows** (in respective measurement units). 3D element sizes can be arbitrarily defined at user preference.

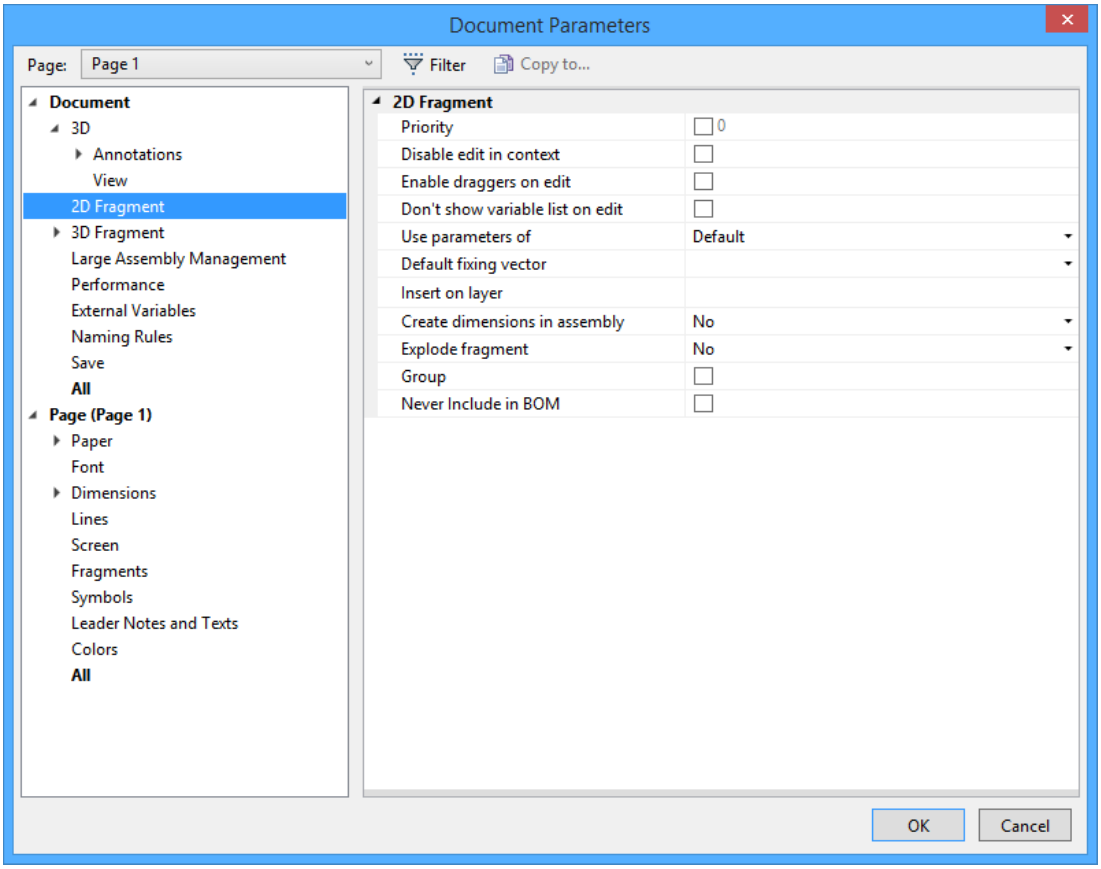
Color:

One color. Sets the display mode of using one color for all 3D elements. This color overrides element own settings.

Color. Defines the color for all 3D elements when one color mode is set. The color can be selected from the menu of colors.

"2D Fragment" Tab

On **2D Fragment** tab you can define the fragment attributes on inserting the document as a fragment.



Priority. Setting this attribute allows to define a priority value to be assigned to the fragment on inserting into an assembly drawing.

Disable Edit In Place. When set, editing this document in the assembly context is prohibited.

Enable Draggers on Edit. If set, the draggers will be enabled for this assembled 2D fragment in the fragment editing mode. The draggers allow modifying the external variables of the fragment by using the mouse.

Don't show Variables List on Edit. This option hides "Preview" and "Variables List" flags in the fragment parameters dialog box on editing. The option works only for the custom dialog boxes of fragment variables.

Use Parameters of. This parameter allows specifying what settings of the drawing will be used upon inserting the given document into the assembly as a fragment:

Default. The default settings specified in the assembly for the parameters of the 2D fragment will be used;

Fragment Document. The settings of the current document will be used;

Current Document. The settings of the assembly drawing will be used.

Default Fixing Vector. This parameter sets the main fixing vector used upon insertion of the given document as a 2D fragment. This parameter duplicates the corresponding flag in the properties of the fixing vector.

Insert on layer. This parameter specifies the name of the layer on which a 2D fragment will be placed when the given document is inserted into the assembly.

Create dimensions in assembly. Option is used to insert dimensions existing in 2D fragment into an assembly. When 2D fragment with dimensions is inserted into the assembly drawing, the dimensions are transformed to separate elements as if they were created right in the assembly drawing. There are three options:

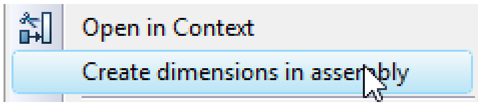
No. Dimensions that already exist in fragment file will be included into assembly. Such dimensions can't be edited because they are part of the fragment.

Automatically. Dimensions will be automatically created after the fragment insertion. When 2D fragment with dimensions is inserted into the assembly drawing, the dimensions are transformed to separate elements as if they were created right in the assembly drawing. These dimensions will not be inserted from the fragment file; they will be created directly in the assembly document. The dimensions will be created according to the dimensions from the fragment file, i.e. their locations and attachments will be inherited from the fragment. You can edit the dimensions. The dimensions will be correctly displayed if you rotate fragment using fixing vector.

> The dimensions will be created only if they are based on graphic lines. Dimensions based on construction lines or nodes will not be created.

If the dimension is considered to be driving in the fragment file, i.e. it's nominal size is set by an external variable, you can change the variable value in the assembly. The fragment and its dimensions will be recalculated according to the new value.

Manually. After fragment insertion, its dimensions won't be created in the assembly. To add dimensions into the assembly you need to use **Create dimensions in assembly** command in the fragment context menu.



The dimensions will be created in the same way as for **Automatically**.

Explode fragment. Allows to explode 2D fragment when it is inserted into an assembly. The fragment is deleted after explosion and all its visible elements copies will be created in the assembly.

No. The fragment won't be exploded after insertion.

Without construction. Only copies of all visible graphic elements from source document will be created.

With construction. Copies of all visible graphic and construction elements from source document will be created.

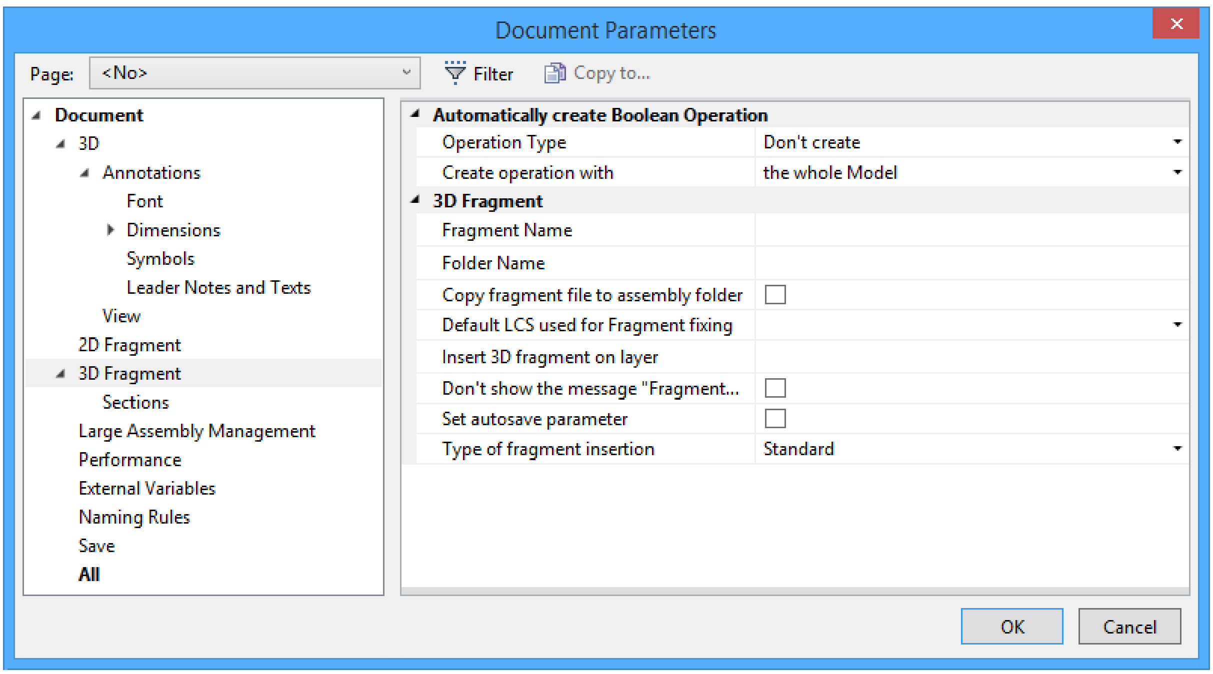
Group. All fragment elements will be automatically grouped after fragment explosion.

The option is used only for exploded elements.

Never include in BOM –fragment data is not included into product structure and corresponding BOM table when the flag is set.

"3D Fragment" Tab

3D fragment tab in **ST: Set Model Parameters** command provides possibility to specify **default LCS used for fragment fixing**, specify **type of insertion**, set **parameters for automatic Boolean operation.**



Automatically create Boolean Operation

When inserting a 3D fragment, one can automatically create a Boolean operation of a specified type. The target body will be the selected body in the assembly, while the tool body - one of the bodies of the 3D fragment, or all fragment bodies at once.

Setting up automatic creation of Boolean operation is performed with the help of several parameters:

Operation Type. This parameter defines the type of the Boolean operation (addition, subtraction, intersection) that will be created upon inserting the fragment in the assembly. The type can be selected from the list.

Create operation with:

When 3D fragment is inserting, Boolean operation of specified type can be applied automatically.

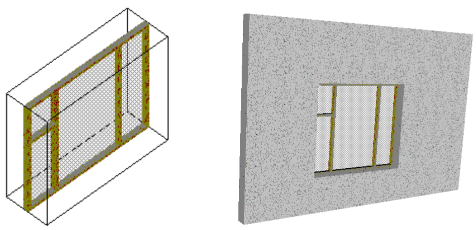
Selected body in 3D assembly will be used as a first Boolean operand, as a second operand – one or all bodies of 3D fragment.

Use the whole Model. With this option checked, the whole 3D model of the fragment will be used as the target body of the Boolean operation.

Use single operation. Selecting this option makes the Boolean use just a single operation within the fragment. The desired operation can be selected from the list of all existing operations.

In the case when a body is used in the Boolean, that is not visible when working with the document of the 3D fragment, it is possible to place it in a special layer marked as **Visible only when model is used as a Fragment.**

Let’s review an example of automatic creation of a Boolean operation on inserting a fragment. Suppose, there is a 3D model of a building wall, and we need to insert a window in it as a fragment. To do so, let’s create the window model in a special way. Create a tool block as a dummy for cutting the opening in the wall for inserting the window. Then call the **Insertion of 3D Model as Fragment**” dialog box (see above). Set the operation type – **Subtraction**, **Use single operation**. From the list of operations select the one that created the tool block.



3D Fragment:

Fragment name. This parameter defines the name with which the fragment will be displayed in the assembly’s model tree.

Folder Name. This parameter specifies the name of the folder into which the fragment will be placed in the assembly’s model tree.

> Fragment's name and folder's name are read from the fragment's file only upon creation of the new fragment. If the values of these parameters are changed, then upon update of the fragment in the assembly the name and the folder will remain the same.

Copy fragment file to assembly folder. If this parameter is turned on, the file of a fragment that is inserted into the assembly is automatically copied into the folder of the assembly document (or its subfolder). A link to the file that has been copied and not to the original one is written in the properties of the fragment.

You can specify a name for the subfolder to the right of the flag. The subfolder will be created inside the folder of the assembly document into which the file of the fragment will be copied. If this field is empty, the file is copied directly into the folder of assembly.

Default LCS used for Fragment fixing is selected from the list. The list contains all local coordinate systems of the model that have the flag **Use for Fragment insertion** set. When inserting such document as a 3D fragment, the thus defined coordinate system will be automatically offered as the source LCS.

If necessary, the permitted degrees of freedom can be set in the properties of the prepared coordinate system, which will insure the correct behavior of the given 3D fragment in an assembly in the mode of moving mated elements (see the chapter "**Mates and degrees of freedom**").

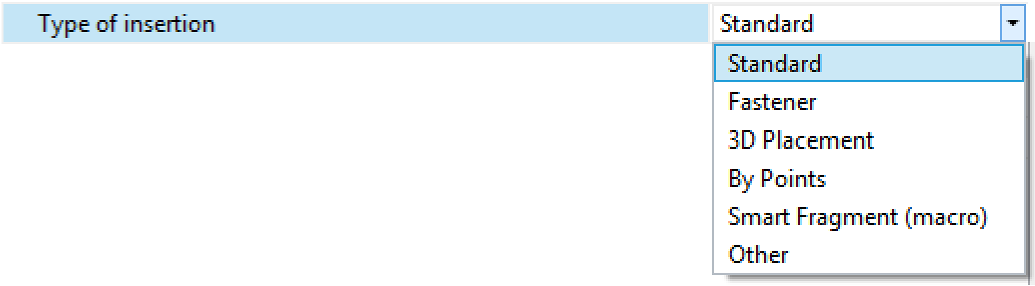
For uncomplicated parts, which can be conveniently inserted into the assembly in the mode of **dynamic snapping**, the given mode can be turned by default.

Insert 3D fragment on layer. This parameter defines the name of the layer onto which the fragment will be placed upon its insertion into the assembly. If in the fragment's document the **Insert on layer** parameter is enabled**,** but upon insertion of the fragment into the assembly this layer does not yet exist, then this layer is created automatically upon request.

Don't show the message Fragment doesn't have bodies. By default this parameter is disabled. In this case the system does not allow for insertion, into the 3D assembly, of the fragment in which the 3D model is absent or suppressed. The corresponding message in the diagnostics window appears as comments. When this flag is enabled the system allows for insertion of "empty" fragment without issuing any messages.

Set Autosave Parameter. If the given parameter is set up, then upon insertion of the current document as a 3D fragment into the assembly, the "Auto Save" parameter is automatically enabled in the parameters of the fragment. The file of such a fragment will automatically be saved upon each saving of the assembly, with the substitution of the values of external variables and adaptive parameters.

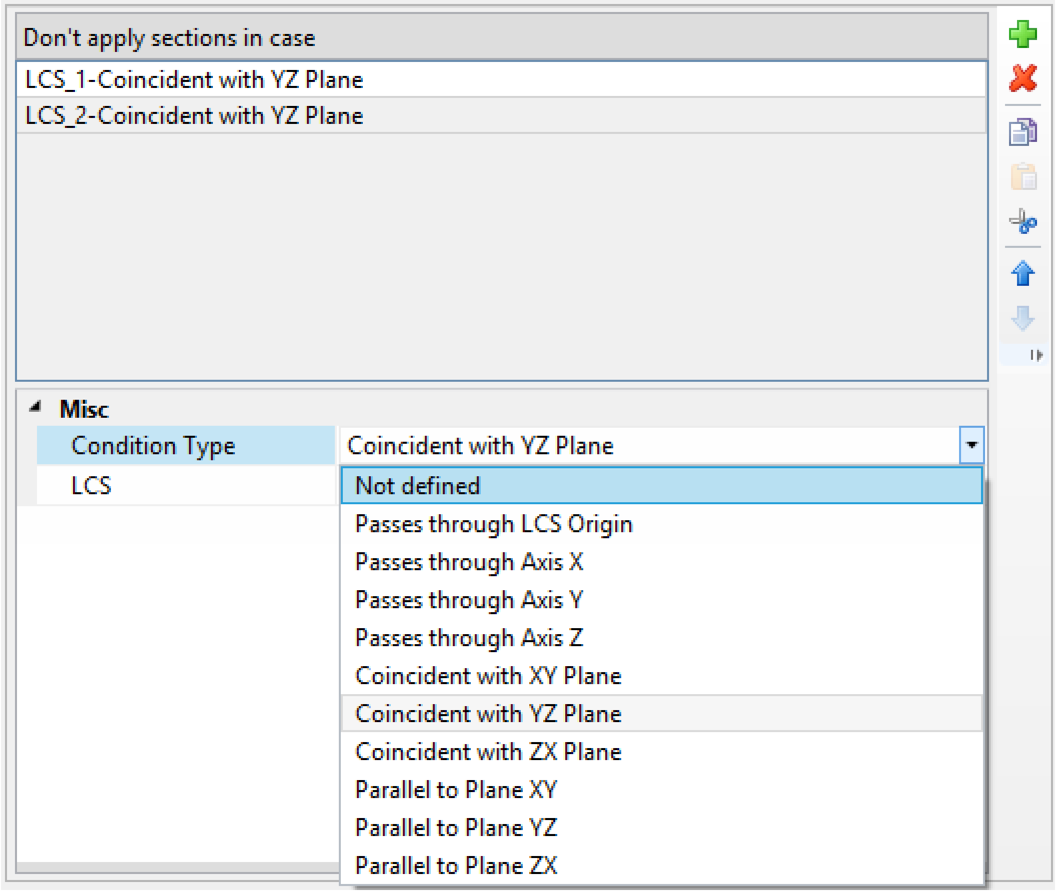
Type of Insertion (for application). The chosen way defines behavior of a 3D fragment at an insert into 3D assembly.



- **Standard**. Upon insertion into the assembly the translation of the fragment is possible only with the help of the manipulator of LCS (dynamic translation is disabled);
- **Fastener**. When inserted into the assembly the fragment dynamically moves after the cursor of the mouse;
- **3D arrangement**. This method of insertion is used for quick creation of the arrangement in 3D scene. In the fragment's file special attachments to the floor, walls, ceiling, horizontal surfaces must be created. These attachments are defined by means of connectors with the specific parameters;
- **By points**. A fragment can be fixed to the selected in the 3D scene points upon insertion. Its size changes according to the distance between the points.
- **Smart fragment (macro)**. This method of insertion is used for parametric fragments whose insertion scenario is described in the program (macro) stored directly in the file of the given fragment or in the external module (DLL). When inserting the file as a fragment the user-specified macro will be executed.

"Sections" Tab

You can define **options for section views on 2D projections** on the **Sections** tab.



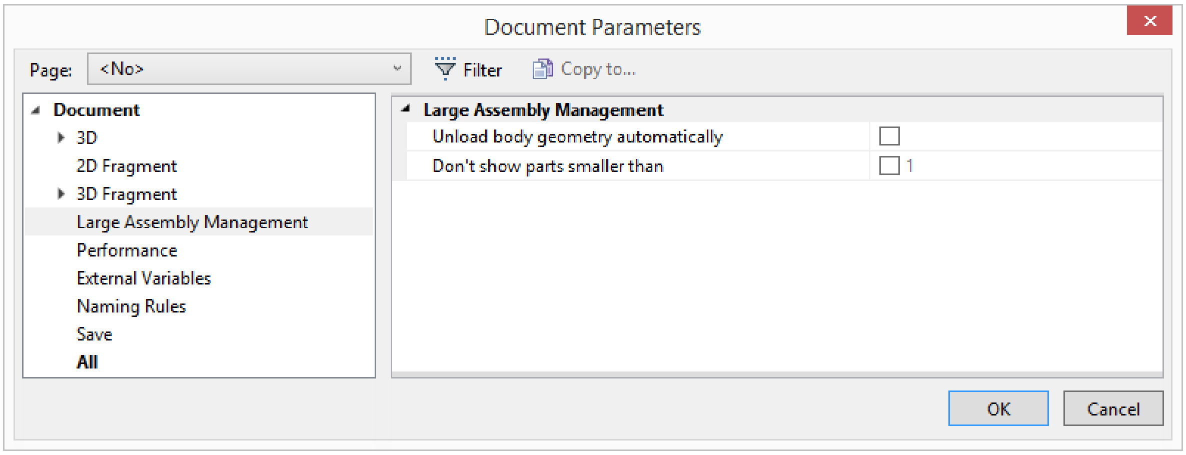
For some models (future components of assemblies) it can be helpful to define special settings for preventing their cutting when used in section views of 2D projections. For example it is known that bolts should not be cut on section views of assembly drawings. This option will help to avoid time consuming manual customization of 2D projection parameters for assemblies that contain such parts.

New condition can be created with the help of  button. Section **Application Condition** section will appear with the lists of conditions and LCS names.

Conditions are organized in relation to the alleged source fragment LCS that exists in a document of a future 3D fragment. For example a part may remain uncut if a section plane goes through a specified axis of the source LCS, or coincides with one of the main LCS planes.

"Large Assembly Management" Tab

"Large Assembly Management" tab serves to define parameters that optimize the use of computational resources and RAM when working with a large assembly model. You can set the following resource-saving modes:

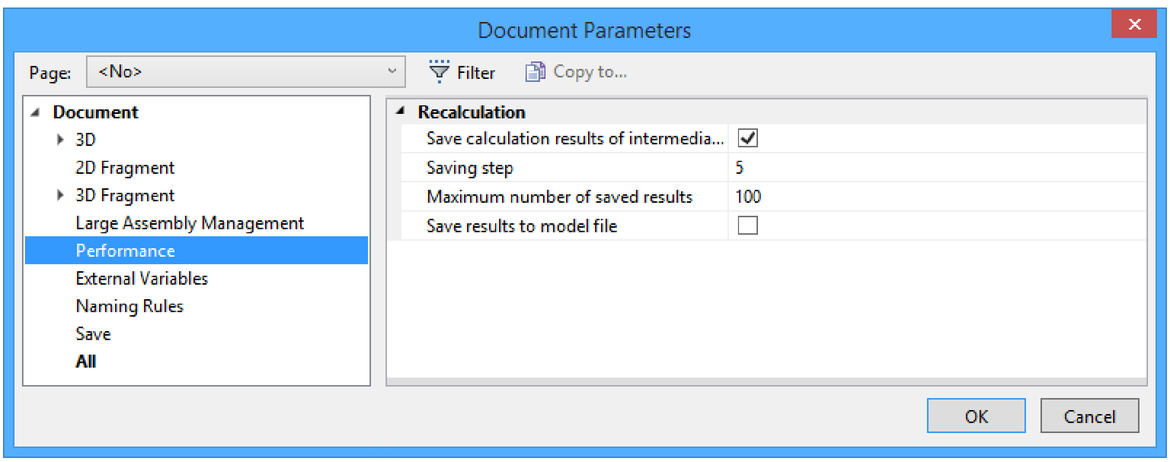


Unload Body Geometry Automatically. When this parameter is set, the model enters the Large Assembly Mode. In this mode, some information about the assembly model geometry is dumped on the hard drive. This information will be loaded as it becomes necessary. A detailed information about the large assembly mode is provided in the chapter "3D Assemblies Creation" of the T-FLEX CAD 3D modeling user manual.

Don't show Parts Smaller than. If this parameter is set, then 3D objects with the size less than the specified (in model units) will not be displayed in the 3D model.

"Performance" Tab

Save calculation results of intermediate operations. Enabling of this option speeds up entrance to editing commands of the intermediate operations, but it slows down the total time of recalculation and allocates more RAM.

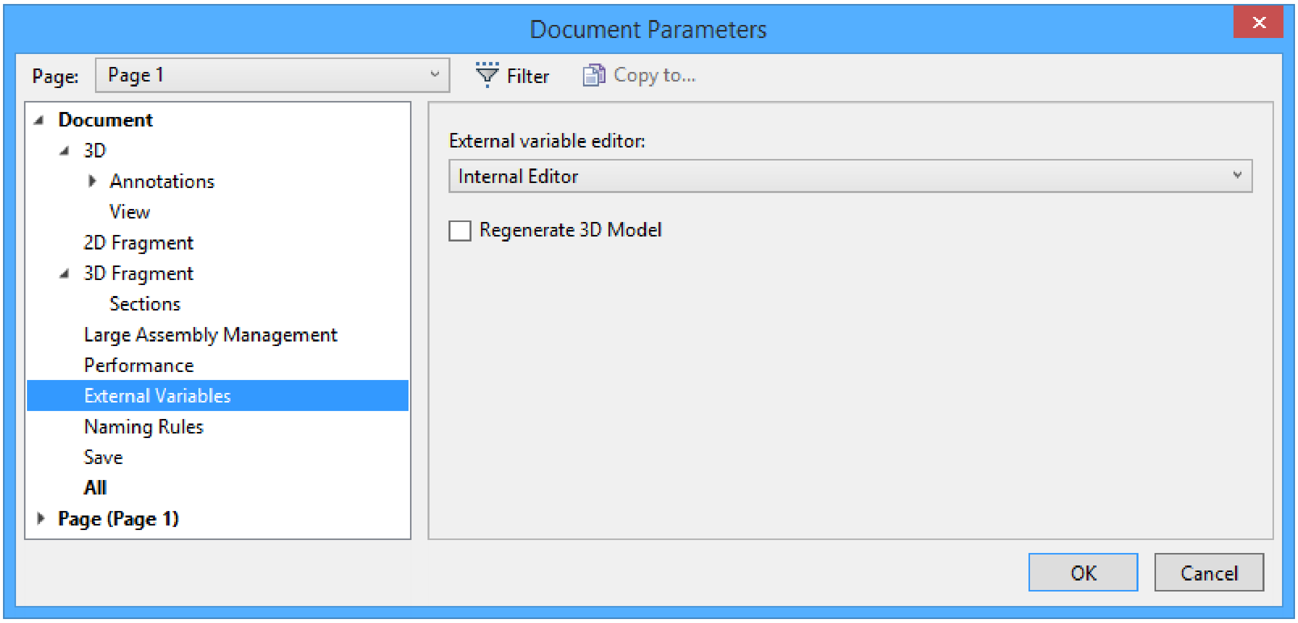


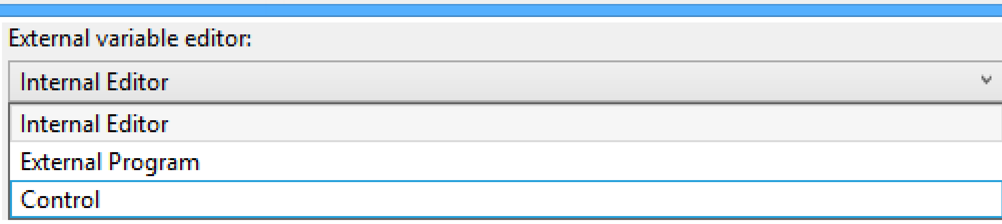
Saving step. System will store intermediate calculation results not for all operations, but for particular operations with the specified step, e.g. for each 5-th operation.

Maximum number of saved results - maximum number if intermediate operations for which the geometry will be saved.

Save results to model file. Store geometrical data of intermediate operations in file, if these operations have such data at the time of document saving. This parameter may speed up entrance to editing commands of intermediate operations after document opening, but will increase file size and opening time.

“External Variables” Tab





External variable editor. This parameter defines the means of editing external variables in the command **M: Model Parameters**.

If **Internal Editor** is set, the model variables are edited using the “Variables Editor” built in the T-FLEX CAD system.

If **External Program** is set, you can define a means of editing model of your own. This implies the user writing a custom application for editing external variables. An example of such application is the format creation functionality that provides a dialog box for filling in the format template. You can use Open API for this purpose.

More information about OPEN API can be found in API help.

Control. This option is used if a custom dialog is created in the document, containing parameters to edit external variables. The document pages to be displayed in the dialog window must be checked in the list of pages.

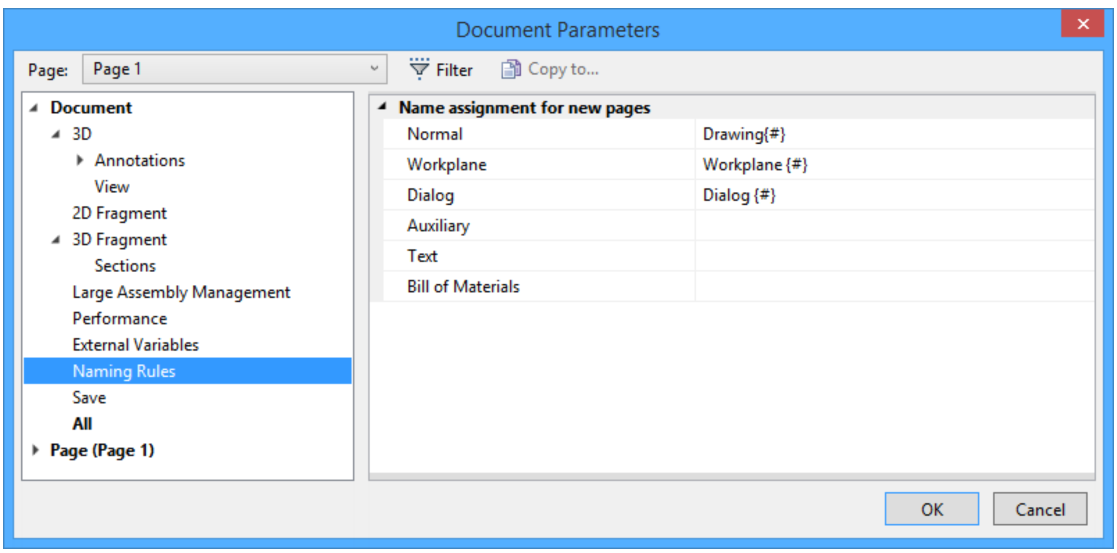
If there are “Dialog”-type pages in the document that were created within the command **TR: Create Control**, then this parameter setting is applied automatically, and the respective page is marked in the list.

Regenerate 3D Model. When this flag is enabled, the 3D model of the document is automatically regenerated upon editing of external variables.

A detailed description of creating a custom dialog and its handling techniques are provided in the chapter “Control Elements. Creating User Defined Dialog Boxes”.

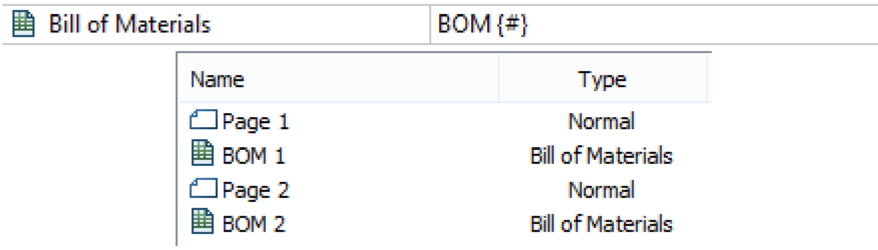
Naming rules Tab

On this tab you can set names and numbering for the following pages: **Normal**, **Workplane**, **Dialog**, **Auxiliary** and **Bill of materials**.

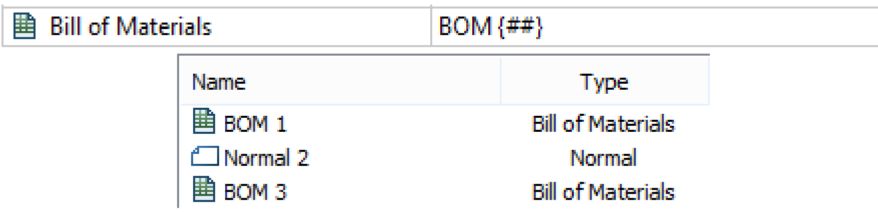


The following numbering can be set:

Sequential numbering within page type. Pages are numbered according to their types.

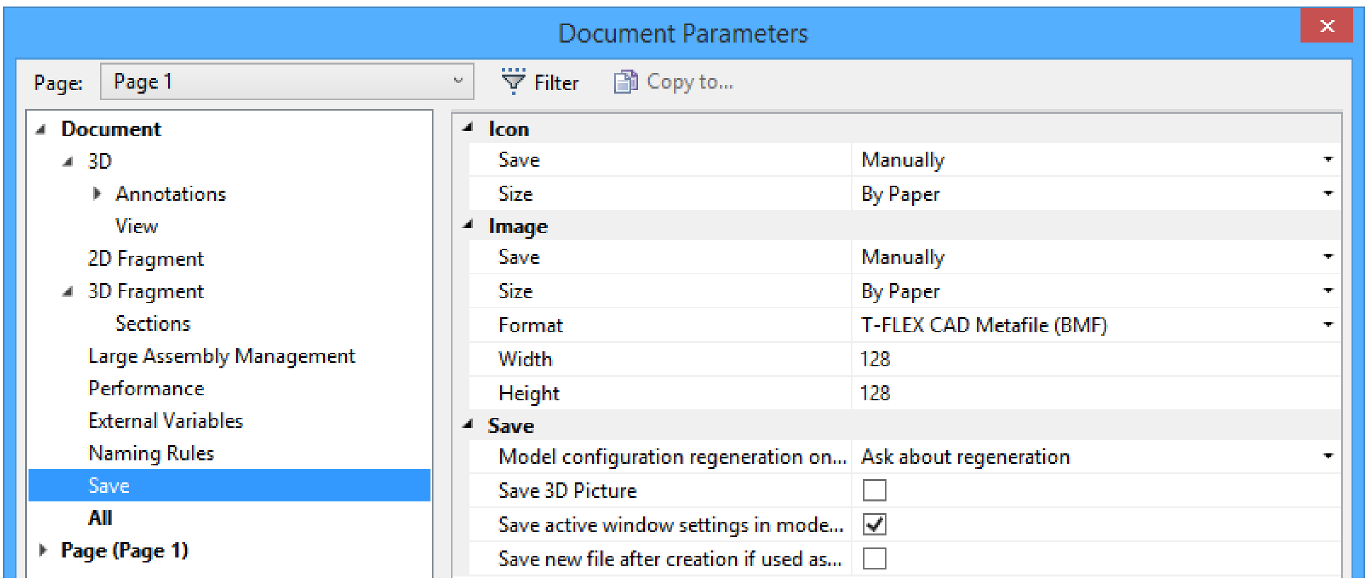


Consecutive numbering of all pages. Pages are numbered regardless to their types.



“Save” Tab

This tab defines automatic creation of a preview and an icon for the current drawing. The preview and the icon can both be created using this tab.



The **Icon** group defines parameters for creating the icons of the document. The document icons will appear in all places where the display of the document icon is foreseen (on the tab of the document upon its opening in the T-FLEX CAD, in the model menu, etc.).

Save. Defines mode of document icon creation. This parameter can assume the following values:

None. The icon is not saved. In this case, an earlier saved icon will remain with the file (if any).

To check existence of a icon in the file, see document properties on the “Preview” tab.

Auto – 2D. An icon with the 2D image is saved automatically on each document save.

Auto – 3D. This icon with the image of the 3D model will be saved automatically each time the document is saved.

Manually. In this mode, the icon can be created manually using the command **PV: Save Preview (Tools > Special Data > Preview)**. Unless using the latter command, this setting is equivalent to “None”.

Size. Defines the size of the icon. The size can be selected from the list, as follows: **Maximize Image**, **By paper**.

The **Image** group defines the parameters of the drawing preview image for its quick display in the document preview pane of the **File > Open…** command dialog box.

Save. This parameter defines the preview saving mode. The parameter can assume the following values: **none**, **auto**, **manually**.

None. Preview is not saved. In this case, an earlier saved preview image will remain with the file (if any).
To check existence of a preview image in the file, see document properties on the “Preview” tab.

Auto. Preview is saved automatically on each document save.

Manually. In this mode, the preview image can be created manually using the command **PV: Save Preview (Tools > Special Data > Preview)**. Unless using the latter command, this setting is equivalent to “None”.

Size. Defines the size of the preview image. The size can be selected from the list as follows:

Maximize Image. With this value, the created preview is restricted to the actual drawing image limits.

By paper. With this value, the created preview covers the whole page according to the paper size defined on the **“General”** tab of the same dialog box.

Format. Defines the preview file format. A format can be selected from the list: **T-FLEX CAD Metafile (BMF)**, **Windows Bitmap (BMP) – 2D**, **Windows Bitmap (BMP) – 3D**.

Width and Height. These parameters define the sizes, in pixels, of the bitmap image.

Save:

The **Model configuration regeneration on save** defines the system behavior on saving a document with configurations. The following modes are possible when document saved:

- **Ask about regeneration** – if the document contains outdated configurations, then the user will be asked at the time of saving the document, whether or not to regenerate configurations.
- **Regenerate all model configurations** – on saving the document, all configurations will be regenerated and saved automatically.
- **Don’t regenerate model configurations**.

Save 3D Picture data in model file. Sets the mode of saving the 3D image in the document file. This saves the regeneration time on opening the file while taking more disk space for the file. Besides, a file with the saved 3D image can be used as a 3D picture (see the command **Operation** > **3D Picture**).

Save active window settings in model file. This parameter allows retaining and saving the current set and position of drawing windows in a document file. For example, if you divided the window into two panes, one of which contains the 3D model, then after loading the file, the sizes and locations will be restored.

Save new file after creation if used as a template. This flag is useful if the current document is intended to be used as a prototype. If this flag is enabled, upon creation of a new document on the basis of this prototype the window of saving the file will immediately appear. When saving is rejected, the new document won't be created.

"Page" Group

"Paper" Tab

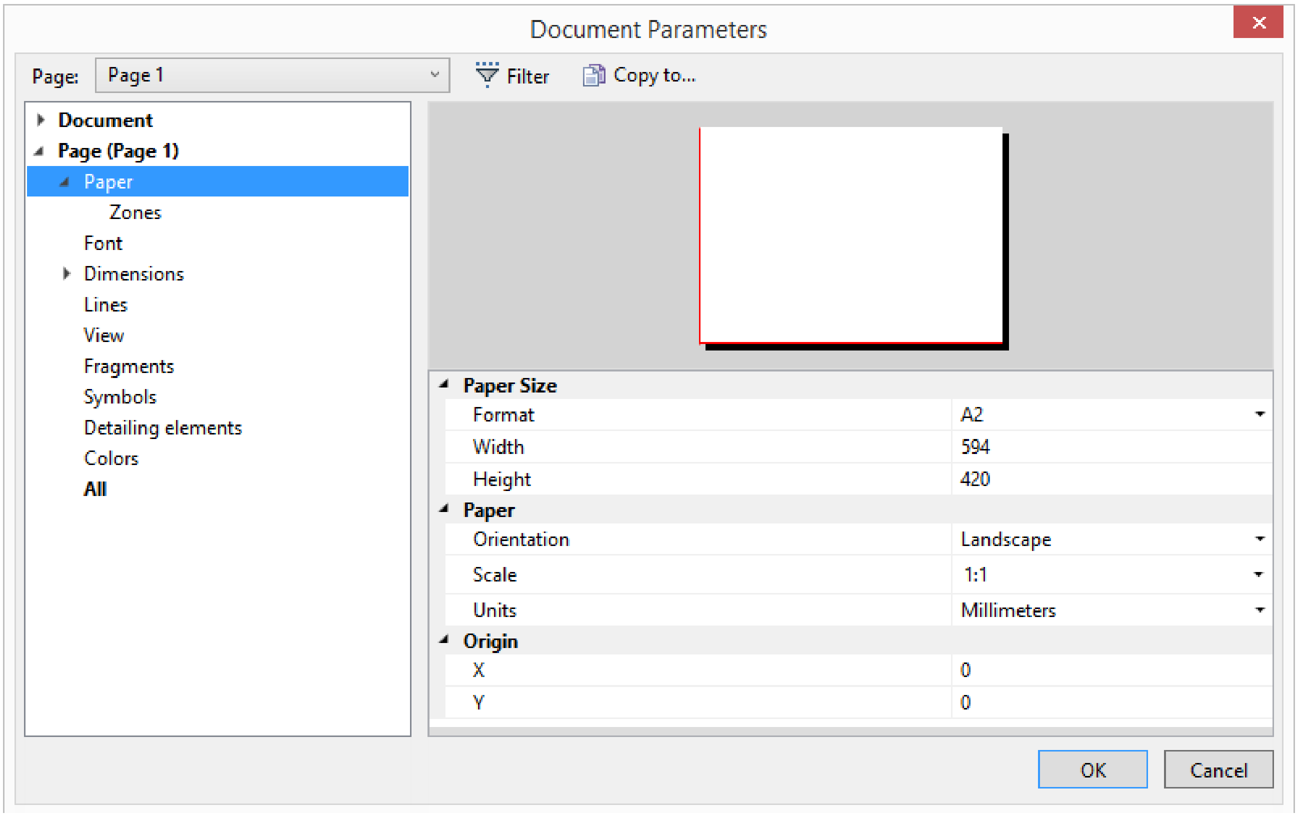
This tab defines main drawing parameters. The preview of the document layout with the specified parameters is available in the preview pane.

Paper size. This group of parameters defines the drawing boundaries.

Format. Provides selection from the list of main formats defined by common standards, such as ISO, ANSI, etc. If a standard format is selected then the **Width** and **Height** parameters are set automatically. If the **Custom** format is selected, then the sizes can be defined manually.

Width. Defines the paper format width. The value is set automatically when the format is selected from the list. If format is "Custom" the size can be defined manually.

Height. Defines the paper format height. The value is set automatically when the format is selected from the list. If format is "Custom" the size can be defined manually.



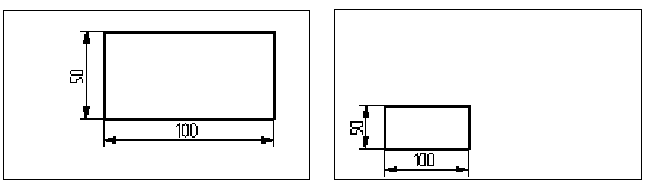
Paper group

Orientation. This parameter defines the orientation of the drawing format. The orientation can be **Portrait** or **Landscape**.

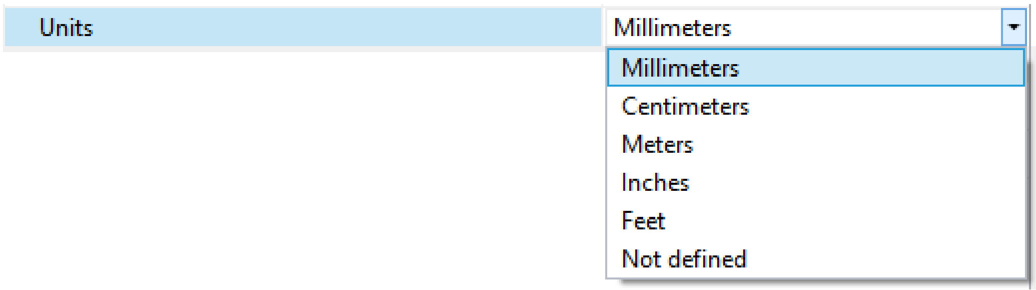
Scale. Defines the drawing scale. An arbitrary value of the scale can be defined, or otherwise be selected from the list. It is recommended to use scale only when actually necessary. In most cases, scaling is not necessary for the following reasons:

1. When outputting to a plotter or printer, the drawing size is not a concern.
2. Special tools are provided for scaling dimension notations of the whole drawing or a portion thereof, as described below.

If it is still necessary to set a scale, it is better be done before starting creating the drawing. Changing the scale setting on an existing drawing may require manual editing of some of its elements. This is because scaling does not merely a proportional modification of all drawing element sizes. The scale in T-FLEX defines only the drawing lines locations, while, for instance, the size of dimension arrows and text will stay unchanged.



Units. Defines the measurement units used by T-FLEX CAD. This parameter is selected from the list.



This parameter affects several issues, such as the following:

- Calculating dimension tolerances that is done differently in metric and inches systems;
- The menu settings when defining roughness parameters and GD&T symbols for surfaces;
- Output to a plotter;
- Exporting different formats;
- The way of converting dimension values, if a conversion system is assigned on the **Dimensions** tab in groups **Scale** or **Alternative scale** to the **Scale** parameter.

Origin. The following parameters define the location of the origin of the drawing coordinate system.

X. Defines the X coordinate of the drawing lower-left corner.

Y. Defines the Y coordinate of the drawing lower-left corner.

The coordinate values are defined in the same units as used for creating the drawing. These are defined under the **Units** item.

> When defining document parameters, one can use variables that will be stored in the drawing file. To do so, enter a variable name instead of a particular parameter value, whether numerical or string. The variable name must be entered in {braces}. If it is a string variable, its name must begin with "$" character. Pressing <F8> while in the intended input box allows selecting the variable from the list of already created ones. The variable values can be modified using the variable editor.

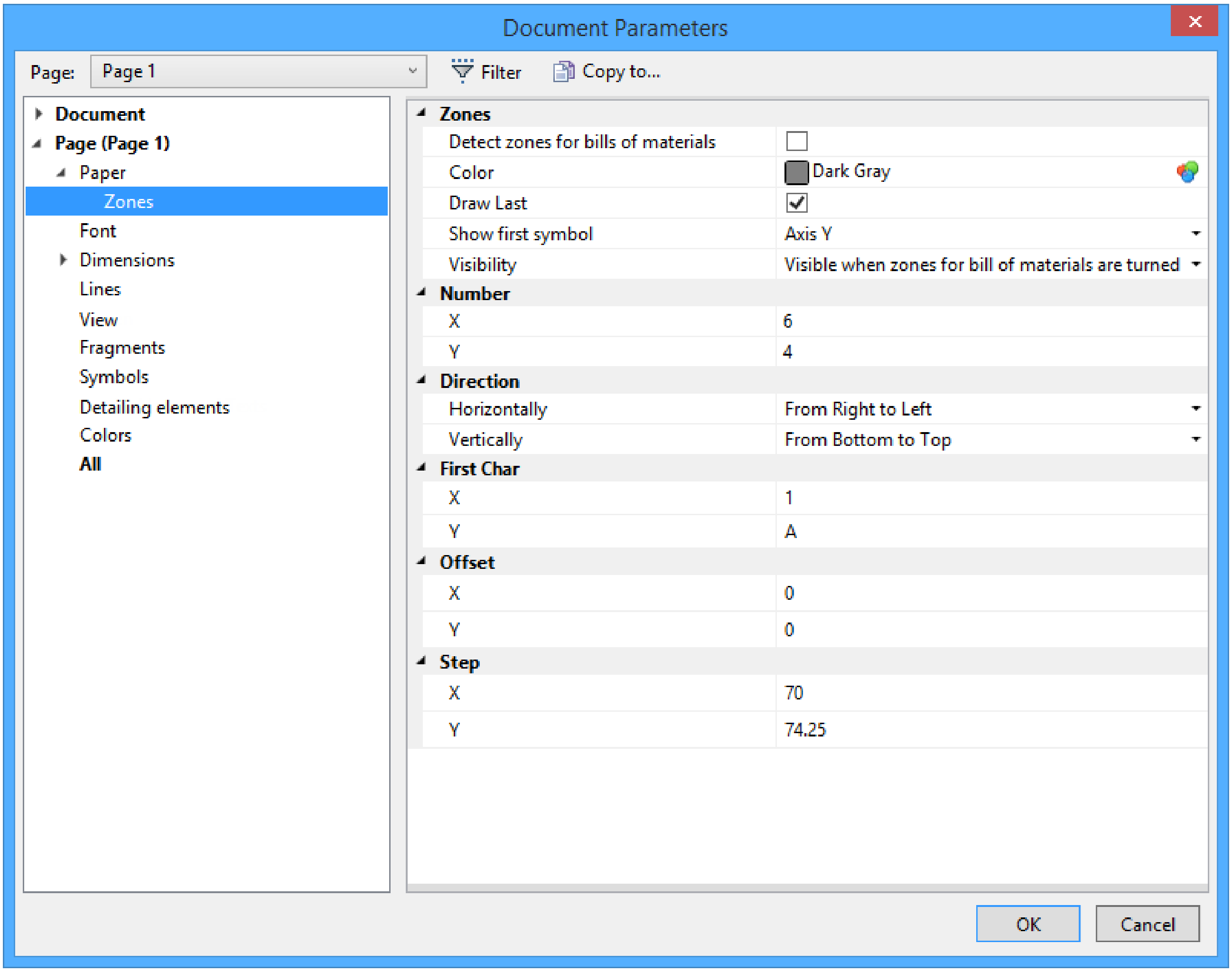
"Zones" Tab

The **Zones** tab is used for defining parameters of dividing a drawing page into zones. For standard formats, the default zone parameters are preset and can be accessed via the **ST: Set Document Parameters** command on the **Paper > Zones** tab. In the dialog box brought up by the command one can redefine these parameters for the current drawing.

Detect Zones for Bills of Materials. The set attribute commands relation between assembly zones and the "Zone" column of the standard BOM.

Color. Defines the color of the zone border lines.

Show first symbol. Specifies, which of the zone-defining symbols, (the one of the X or Y axis), will come first in its marking.



Visibility. Defines zones visibility on the drawing. This parameter has three values:

Yes or **No** – defines zone visibility on the drawing.

Visible when zones for bill of materials are turned on – provides zone visibility only when the parameter **Detect Zones for Bills of Materials** is set. Otherwise, the zones are not displayed on the drawing.

Number, X and Y. The number of zone columns and rows respectively.

Direction. Defines the itemization direction for zones: left to right or right to left, top down or bottom up.

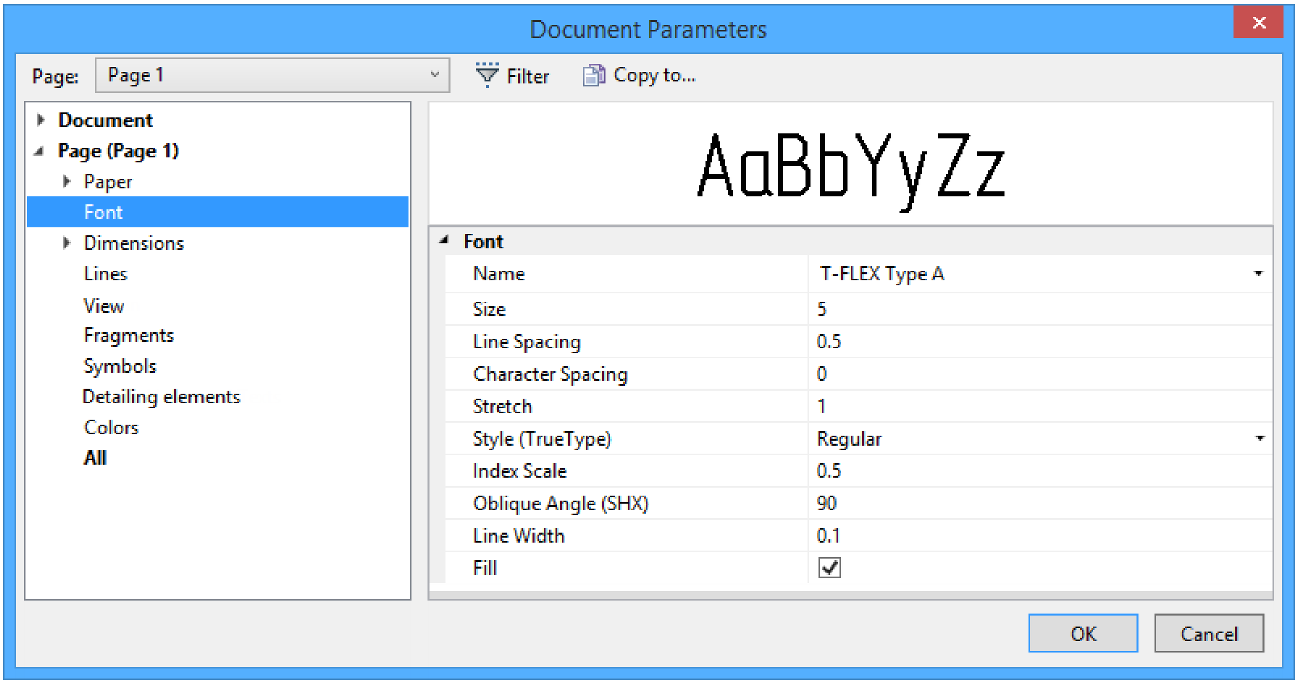
First char, X and Y. Define the characters to begin with when itemizing the zone columns and rows respectively.

> To keep in mind: the **Document parameters** command settings affect only one page. Therefore, for drawings spanning over multiple pages, the division into zones should be defined separately, taking into account the through notation of zones along the horizontal (X axis).

Offset. Defines the X and Y offsets of the area being divided into zones with respect to the point (0,0).

Step. Defines the X and Y dimensions of one zone.

"Font" Tab



This tab defines font parameters for all standard detailing elements unless already defined within the detailing elements themselves. These detailing elements are text, dimensions, roughnesses, leader notes and tolerances.

Name. Defines name and type of the default font that will be used on text objects creation. T-FLEX CAD supports two types of fonts: TrueType fonts (T), that are standard for Windows, and vector fonts in SHX (TF) format. The TrueType and SHX fonts are distinguished in the font menu by the respective icons before the names.

Size. Defines the vertical size of the font by capital letters, as, for instance, the height of the character "A". Any font size is allowed, except 0.

Line Spacing. This is the distance between two neighboring lines of a multi-line text. Line interval is set in relative units. To calculate the absolute value of the line interval, multiply this parameter by the font height.

Character Spacing. Defines the additional distance between two neighboring characters in a string. This parameter value is also relative. To calculate the absolute value of the additional interval, multiply this parameter by the font height.

Stretch. Defines the scale factor for the width of the font symbol. It is possible to specify any value for this factor except 0.

Style (TrueType). Defines the font style. This is a standard parameter of fonts and assumes the values supported by Windows, namely normal, semi-bold, italic, semi-bold italic.

Index scale. Defines the scale factor for calculating the size of the subscript and superscript fonts from the normal font size. Subscripts and superscripts are used, for example, for dimension tolerances and other indexes.

Oblique Angle (SHX). Defines the font tilt angle. Any angle value is allowed, except 0, 180, 360, etc. The normal (upright) font has the tilt angle value of 90 degrees.

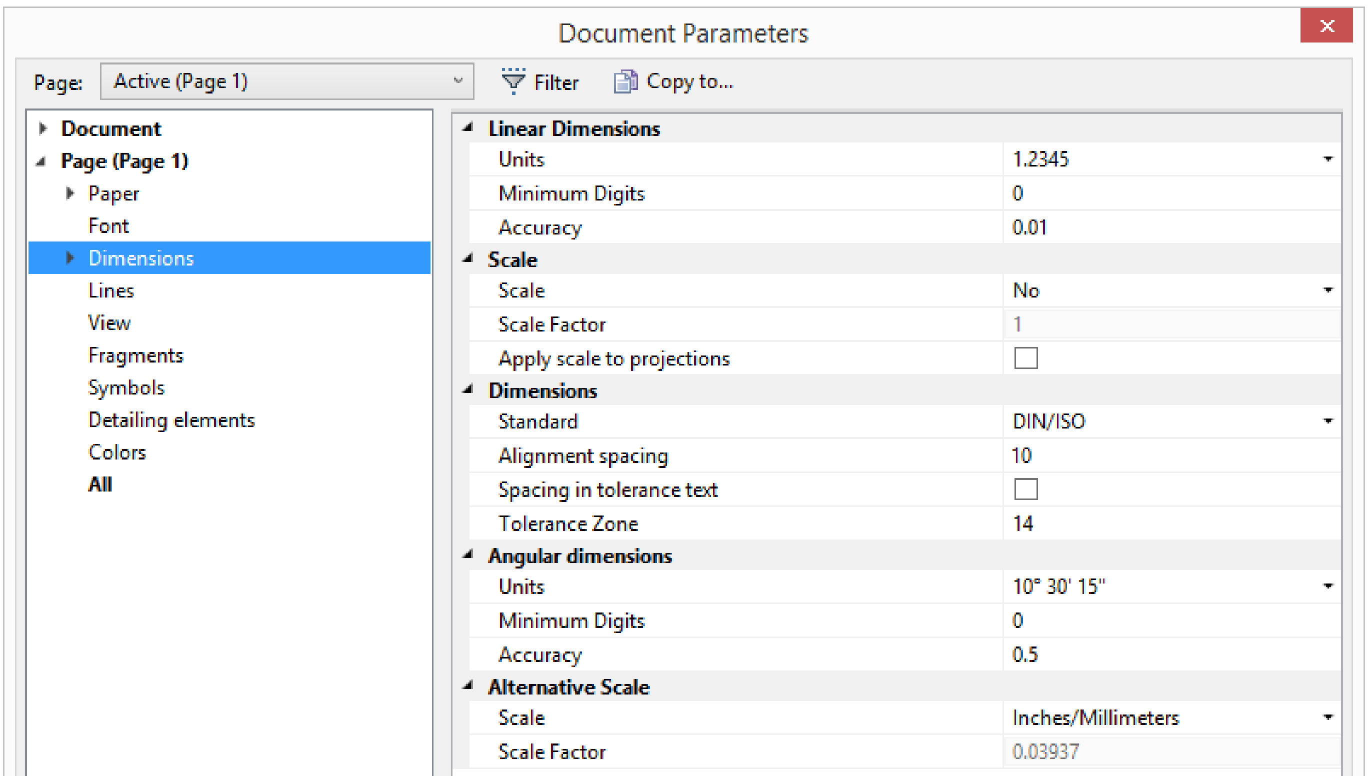
Line Width. This parameter defines thickness of character contour lines for the SHX fonts.

Fill. If set, then the characters are filled with color when applicable. Otherwise, they are drawn by contours only.

> The **Oblique Angle, Line width** and **Fill** parameters affect only vector fonts of the SHX format.

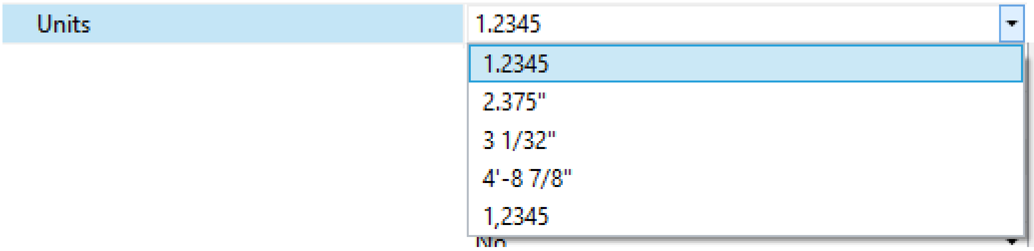
The settings on this tab affect the size of all detailing elements of the drawing (dimensions, text, roughnesses, GD&T symbols, leader notes), whose **Font** tab settings are displayed in the square brackets, that is taken from the document parameters**.** The elements will not be modified whose **Font** tab settings are explicitly defined.

"Dimensions" Tab



Linear Dimensions. This group contains the following parameters:

Units. Defines the linear dimension value representation. This is mostly relevant to dimension values in inches.



Accuracy. Defines rounding accuracy of the linear dimension values. For example, the accuracy "*0.01*" means the dimension values will be rounded to the second decimal digit. The "*0*" accuracy means the dimension values will not be rounded.

Minimum digits. Sometimes, a certain number of decimal digits are required to be displayed on a dimension, including trailing zeros. This can be insured with the “Minimum digits” setting. For example, setting the value “3”, then the dimension value 28.5 will be shown on the drawing as 28.500.

Scale:

Scale factor. Reflects the scale factor of dimension value conversion per the dimension scale parameter. All dimension values will be multiplied by this scale factor. If **Dimension scale** is **Custom**, one can enter your own scale factor for converting dimension values of linear dimensions.

Scale. Defines the way of converting dimension values of the linear dimensions. The following conversion systems can be selected from the list:

No. No dimension conversion is done.

Inch/Metric. In this case, conversion of the dimension values will be done per the parameter **Units** on the **General** tab. If the inches system is set, then the linear dimension values will be converted to millimeters, that is, multiplied by the scale factor of 25.4. If the metric system is set, then the dimension values will be converted into inches, that is, multiplied by the scale factor of 1/25.4.
Thus, one can work with a drawing in metric system while applying dimensions with values in inches, and vice versa.

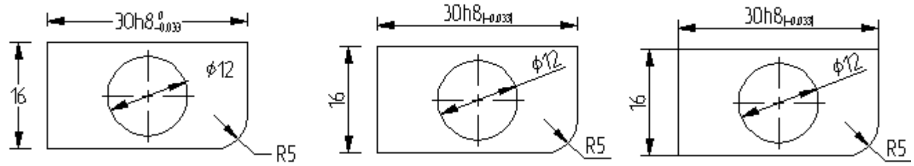
Custom. With this setting, an arbitrary dimension value conversion factor can be specified.

Apply scale to projections. This parameter is used in the 3D version of the system only. It has an effect on dimension creation within 2D projections in the case when a dimension scale is defined in the command **ST: Document Parameters** for the 2D page that contains the projection. When the flag is set, then the specified scale is applied to the dimensions on a 2D projection with a scale; if the flag is cleared, the scale is ignored.

Dimensions:

Standard. Defines the appearance of dimensions on a drawing. A dimension standard can be selected from the list. Three standards are provided in the list – ANSI, BS-308 and DIN/ISO.

BS-308 stands for ANSI standard for architects or British standard. Upon changing the dimension standard, the dimensions automatically redraw to comply with the new standard.



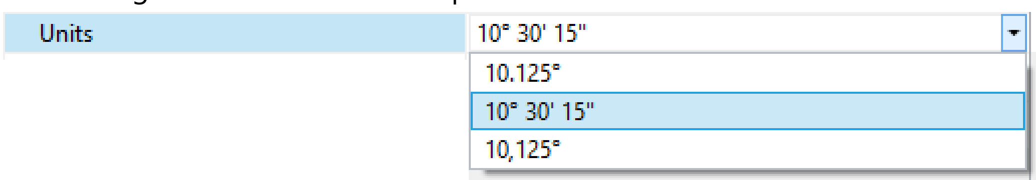
Standard ANSI   Standard BS-308   Standard DIN/ISO

Alignment spacing. Specifies spacing when aligning dimensions with offset.

Tolerance grade. Defines the tolerance grade for dimensions on a drawing. If the accuracy parameter of the dimension is set equal to this parameter then the tolerance string and limits will not be displayed as part of the dimension.

Angular Dimensions. This group contains the following parameters:

Units. Defines the angular dimension value representation.



Minimum digits. Defines a certain number of decimal digits, which are required to be displayed on a dimension, including trailing zeros.

Accuracy. Defines rounding accuracy of the angular dimension values. Accuracy “0” means that dimension values will not be rounded. Value “0.5” = 30 minutes, «1/60» = 1 minute, «1/360» = 1 second.

Spacing in tolerance text. When this option is enabled, tolerances of dimensions are separated with spaces for easy reading.

Alternative scale.

Scale. Defines the way of converting dimension values of the alternative dimensions. The following conversion systems can be selected from the list:

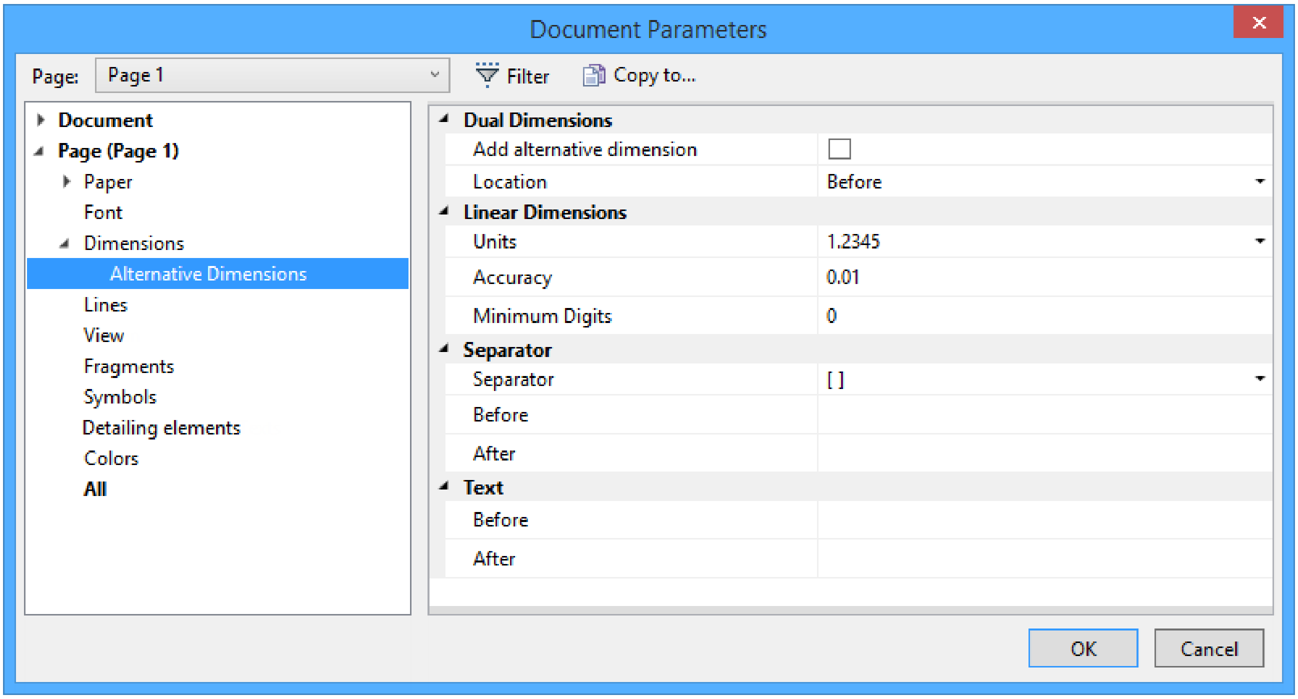
No. No alternative dimension conversion is done.

Inch/Metric. In this case, conversion of the dimension values will be done per the parameter **Units** on the **General** tab. If the inches system is set, then the alternative dimension values will be converted to millimeters, that is, multiplied by the scale factor of 25.4. If the metric system is set, then the alternative dimension values will be converted into inches, that is, multiplied by the scale factor of 1/25.4. This allows setting both metric and inches value using dual dimensions.

Custom. With this setting, an arbitrary scale factor can be specified for converting an alternative dimension value.

Scale factor. Reflects the scale factor for converting alternative dimension values per the alternative scale parameter.

"Alternative Dimensions" Tab



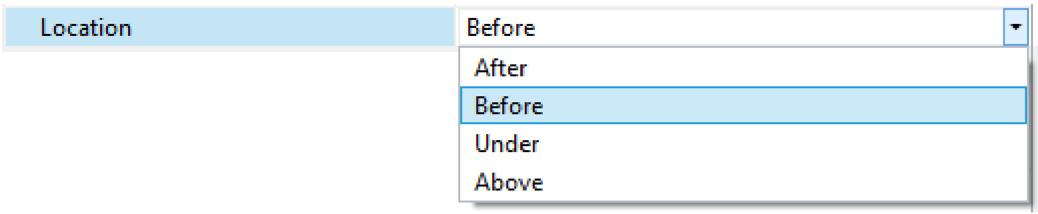
Parameters defined on this tab will only affect those alternative dimensions whose parameters are marked **Set as default.**

The group of parameters **Dual Dimensions** defines the presence and location of alternative dimensions on the drawing.

Add Alternative Dimension. This attribute sets the presence of alternative dimensions on the drawing. If alternative dimensions are not required, clear the check in the attribute field.

Location. Defines the location of alternative dimensions on the drawing with respect to the primary dimension value.

According to the selection choice, the alternative dimension value may be displayed **After**, **Before**, **Under** or **Above** the primary dimension value on the drawing.



Linear Dimensions. This group includes:

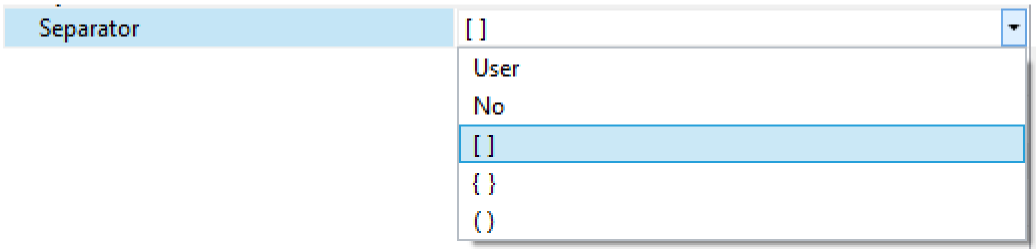
Accuracy. Defines rounding accuracy of the alternative linear dimension values.

Units. Defines the alternative linear dimension value representation.

Minimum digits. Defines the minimum number of decimal digits.

Separator:

Separator. Parameters of the given group define separating characters that separate the value of the alternative dimension from the primary one. In the drop-down list the following variants are available:



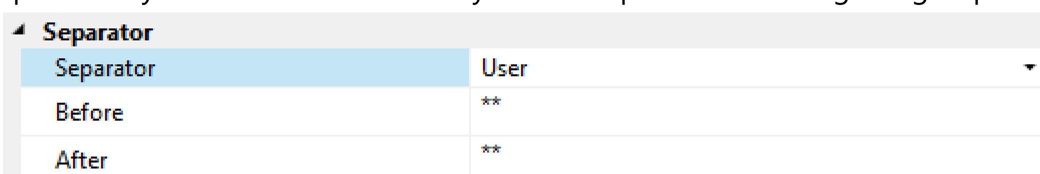
None. The alternative dimension value will be displayed without separators.

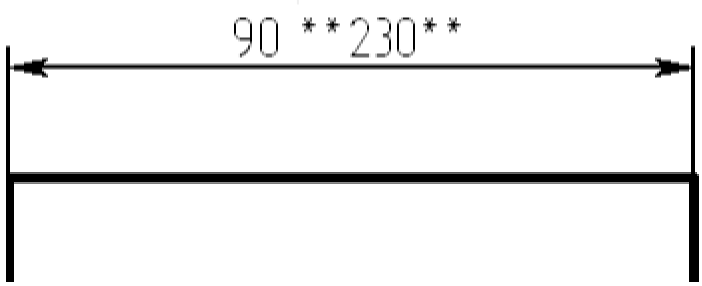
[]. The alternative dimension value will be displayed in brackets.

{}. The alternative dimension value will be displayed in braces.

( ). The value of the alternative dimension will be shown in circular brackets.

User. . The value of the alternative dimension will be displayed with the help of arbitrary separators that are specified by the user in the identically-named input fields of the given group.





Before. If “Separator” is "User", it is possible to enter any character as separator, for example, ‘*’.

After. If "Separator" is "User", it is possible to enter any symbol as separator, for example, '*'.

Text group:

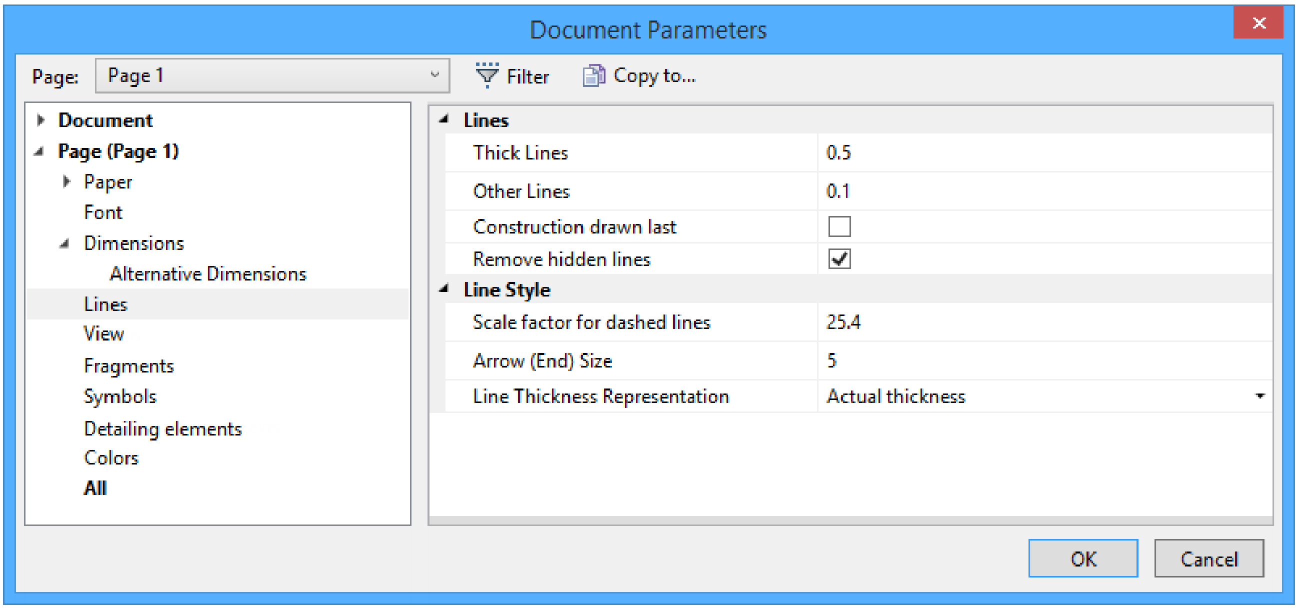
Before and **After**. Defines the text strings to precede and follow the value string of an alternative dimension. The strings can be entered explicitly or substituted by numerical or string variables.

The variables for the strings must be entered by their names in braces. For example, suppose, a variable *"A"* is introduced in the drawing, and we want its value to be displayed as part of an alternative dimension value string. Enter the variable name in braces, *{A}*, as one of the strings, *"Before"* or *"After"*. If this is a string variable with the value being a character string, then the first character in its name must be the dollar sign, for example, *{$Text}*.

"Lines" Tab

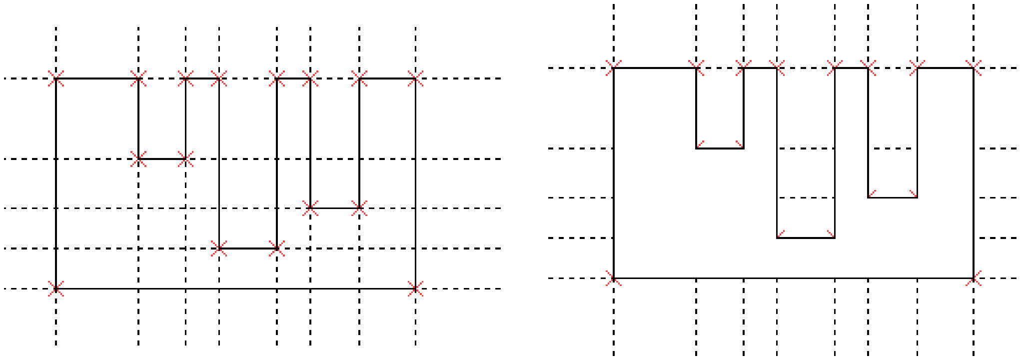
Lines:

Thick lines. Defines the thickness of the main solid graphic line (CONTINUOUS).

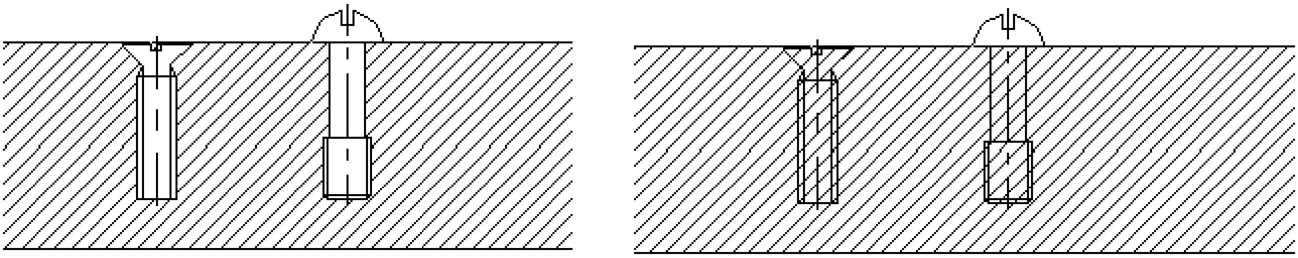


Other lines. Defines the thickness of all the rest of graphic lines and detailing elements (the lines of tolerances, roughnesses, leader notes and dimensions).

Construction drawn last. If set, the construction elements will be drawn last upon redraws (see the diagram on the left). Otherwise, they are drawn first (right) and may be hidden by other system elements (fills, graphic lines, etc.).



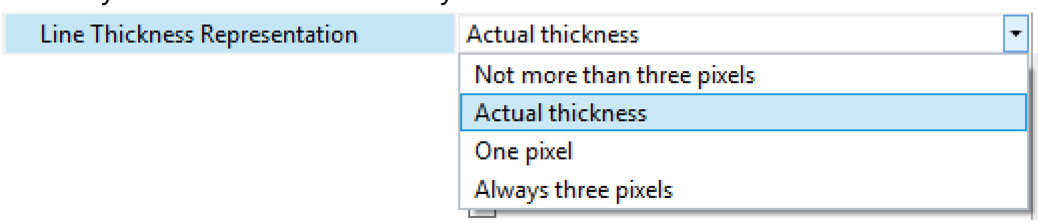
Remove hidden lines. If set, the hidden lines will be removed from display. These are the lines behind the fragments with hidden line removal contours. The lines will be removed according to the fragment priorities.



Line style:

Scale factor for dashed lines. Defines the scale factor for dashed lines with respect to the dash size described in the file of line types (TCAD.LIN). Does not affect the display of solid lines. The file format is the same as the format of the AutoCAD line description file.

Arrow (end) size. Defines the size of arrows (ends, tips)of the dimension leader and witness lines and graphic lines. Any desired size can be set by the user.



Line thickness representation. This parameter defines the graphic line appearance on the screen. The parameter can be selected from the list:

Not more than three pixels. All graphic lines will be displayed no more than three pixels thick. This parameter is relevant to the lines whose line thickness is greater than three pixels.

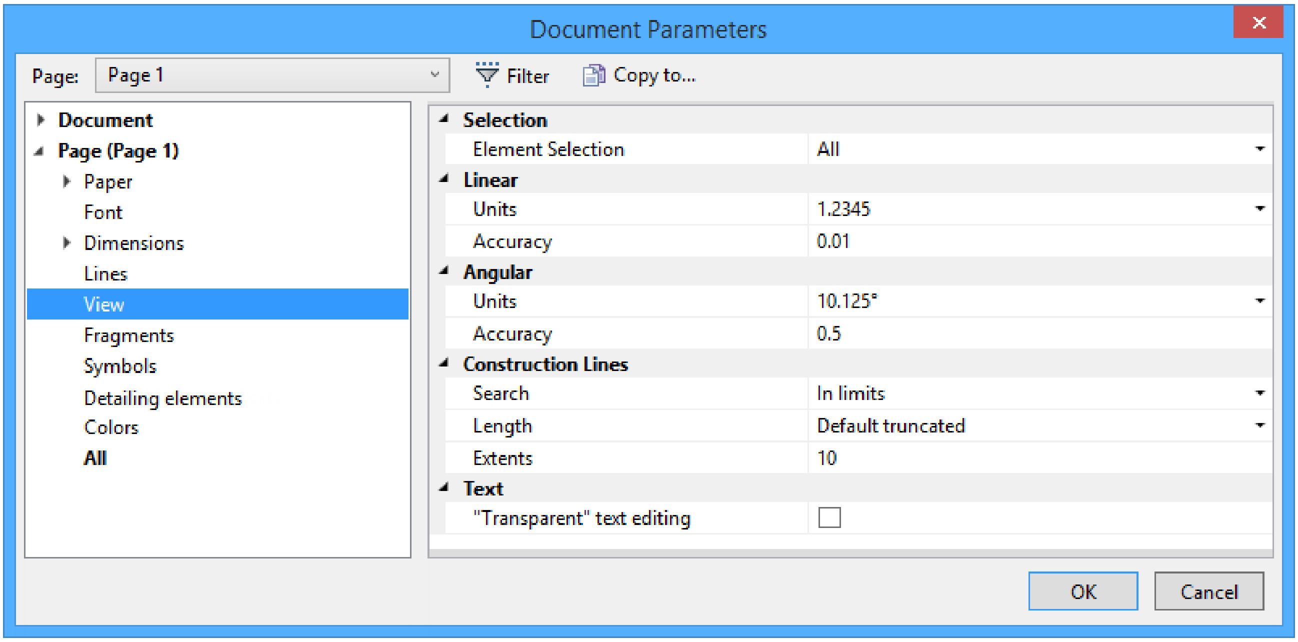
Actual thickness. All graphic lines will be displayed as thick as actually are. The actual line thickness is defined by the **Line thickness**” item on this tab.

Always three pixels. All primary graphic lines will be drawn with a thickness of 3 pixels, and all thin graphic lines – with a thickness of 1 pixel regardless of the scale of the drawing.

One pixel. All graphic lines will be displayed one pixel thick.

" View " Tab

The **View** group of parameters defines the modes of displaying linear and angular values and the means of searching and selecting elements. These parameters do not modify the drawing graphic elements. Rather, these are system settings specific to the particular drawing.



Selection:

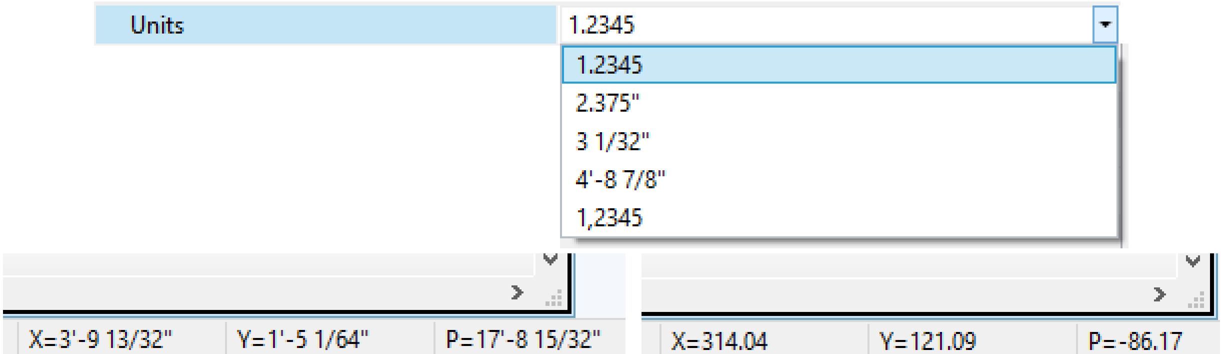
Element selection. Defines element selection modes while in drawing and editing commands.. Select one of the two modes:

All. When creating and editing elements, all existing elements will be allowed for selection.

Visible only. When creating and editing elements, only the visible elements will be allowed for selection. The element visibility is determined based on element levels and visibility intervals defined in the command **SH: Set Levels (Customize > Levels…)**, as well as layer configurations defined in the command **QL: Configure Layers (Customize > Layers…)**.

Linear:

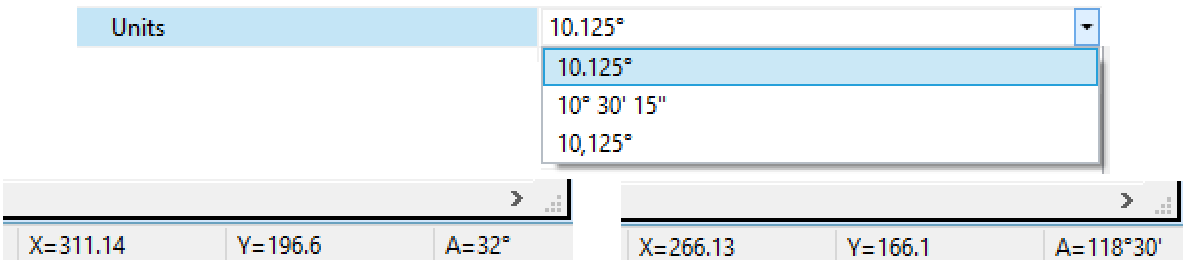
Units. Defines the way of displaying linear coordinates in the information fields of the application, as, for example, *X* and *Y* coordinates in the status bar. The parameter does not affect the display of dimensions and other graphic elements.



Accuracy. Defines the accuracy of displaying linear coordinates in the information fields of the application, as, for example, *X* and *Y* coordinates in the status bar. The parameter does not affect the display of dimensions and other graphic elements.

Angular:

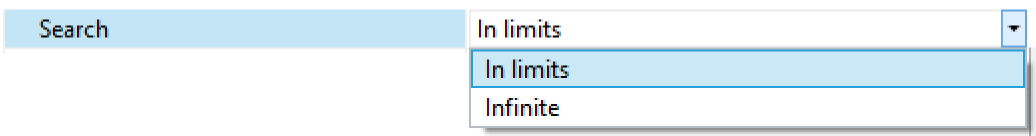
Units. Defines the way of displaying angular coordinates in the information fields of the application. The parameter does not affect the display of dimensions and other graphic elements.



Accuracy. Defines the accuracy of displaying angular coordinates in the information fields of the application. The parameter does not affect the display of dimensions and other graphic elements.

Construction Lines:

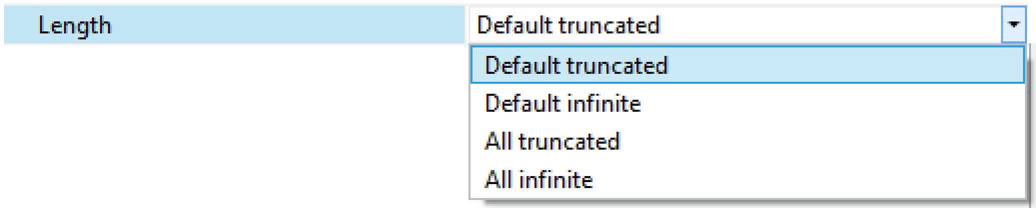
Search. Defines the mode of selecting straight construction lines. One of two modes can be selected, as follows:



In limits. The lines will be selected according to their length defined by the **Length** parameter on this tab. If construction lines are displayed as finite line segments, then the nearest segment will be selected.

Infinite. The lines will be selected as infinite lines, regardless of the **Length** parameter value on this tab and the way of displaying the lines.

Length. Defines the way of displaying the straight construction lines. Construction lines are displayed as either infinite lines, or finite segments bounded by their end nodes. To refresh the lines display per the new settings, use the command **EC: Edit Construction** under the icon (use the option "Update all Line extents" under the icon  in the automenu). One can set one of the following construction line display modes:

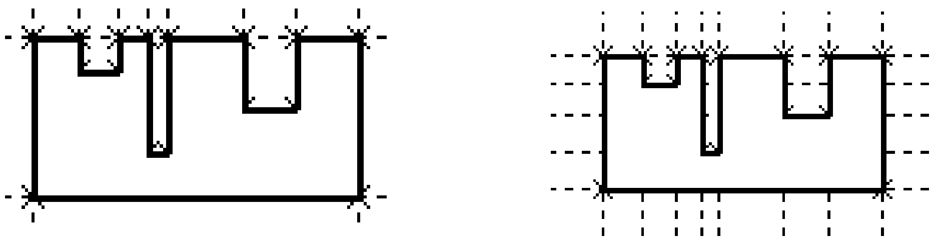


Default truncated. Affects the construction lines whose **Length** property is **Set as default.** Such lines will be displayed as segments bounded by two end nodes.

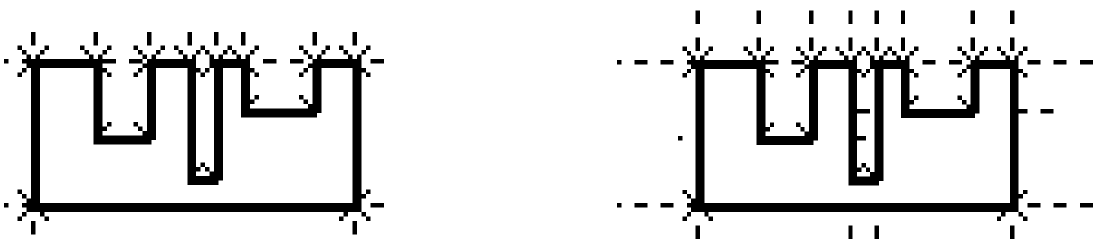
Default infinite. Affects the construction lines whose **Length** property is **Set as default.** Such lines will be displayed as infinite lines.

All truncated. With this value, all construction lines will be displayed as segments bounded by two end nodes.

All infinite. With this value, all construction lines will be displayed as infinite lines.



Extents. Defines extension of construction line overhangs beyond the end nodes when displaying as a finite segment.

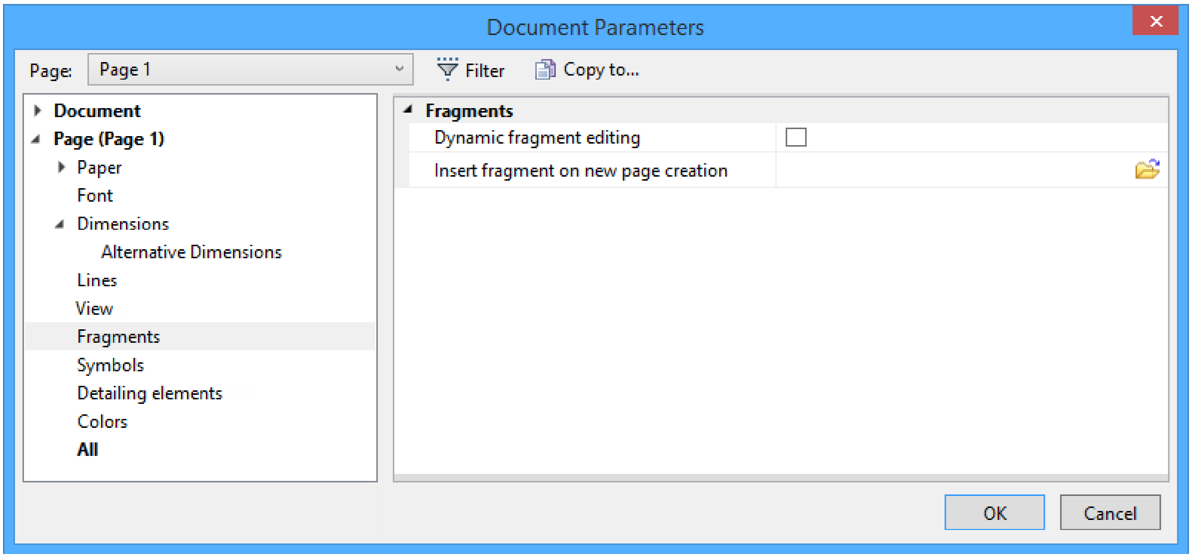


Text:

"**Transparent**" **text editing** allows to edit paragraph text right after clicking the element with left mouse button. If the attribute is not set, then only the variables inserted in the paragraph text can be edited in this way.

"Fragments" Tab

Dynamic Fragment Editing. When this parameter is set, dynamic regeneration of the model and redrawing of the image occur upon inserting and editing 2D fragments. This makes the work with fragments clearer. However, this is not suitable for drawings with large number of elements since it slows down the work.

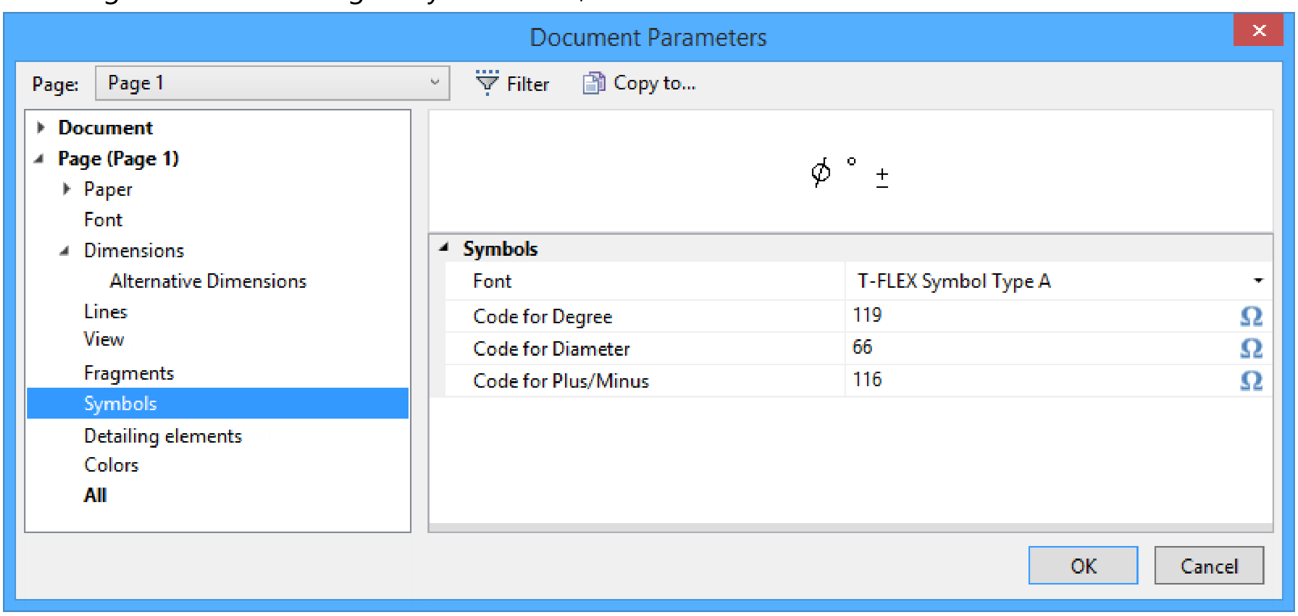


Insert Fragment on new Page creation. In the provided input box, enter the path to the fragment to be automatically inserted upon new page creation. If the fragment is a multi-page document, then the first page will be inserted. This parameter is defined automatically when creating BOM.

"Symbols" Tab

Font. Defines the font for special symbols used on the current page.

Code for Degree. Defines the symbol to substitute instead of the "*%%d*" key in text strings on the drawing. Default is the degree symbol code, which is 119.



Code for Diameter. Defines the symbol to substitute instead of the "*%%c*" key in text strings on the drawing. Default is the diameter symbol code, which is 066.

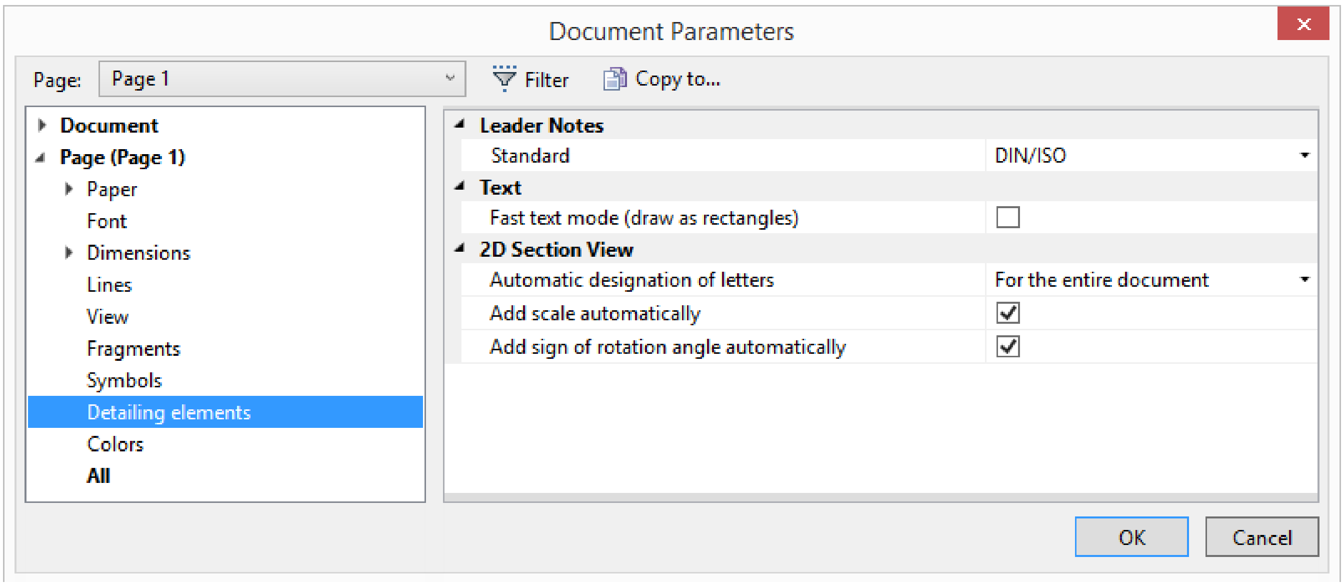
Code for Plus/minus. Defines the symbol to substitute instead of the "*%%p*" key in text strings on the drawing. Default is the plus/minus symbol code, which is 116.

When changing the font, a new symbol code can be defined manually by typing the appropriate value or by selecting a symbol from the menu "Insert Symbol" after pressing the button **[Select]**. When selecting a symbol from the table, the symbol code is entered automatically.

"Detailing elements" Tab

Leader Notes:

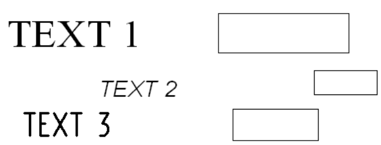
Standard. Defines the appearance of leader notes on a drawing. A leader note standard can be selected from the list. Two standards are provided – ANSI and BS/DIN/ISO.



Upon changing the leader note standard, the leader notes automatically redraw to comply with the new standard.

Text:

Fast text mode (draw as rectangles). If set, all text are displayed as rectangles of the text size. Meanwhile, the text itself will not be displayed. This setting helps speed up display of large drawings.



2D Section View:

Automatic designation of letters. Automatic assignment of letters sequence for detail, section and auxiliary views.

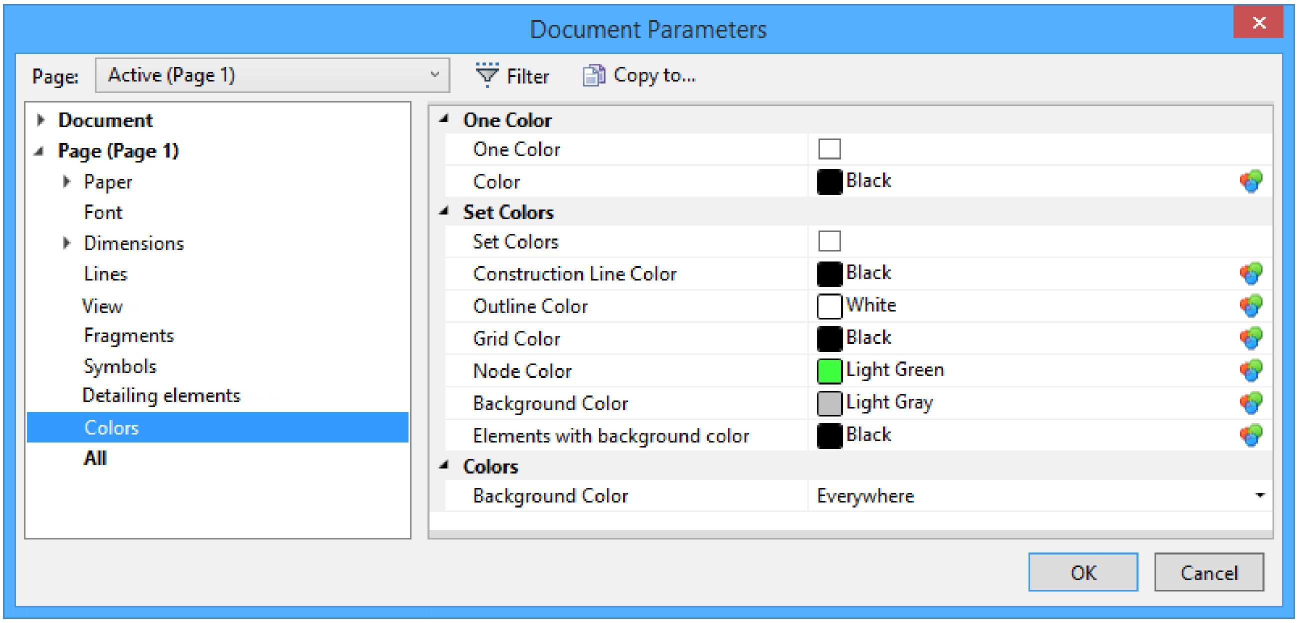
Do not designate. You can assign the same letters for different section views.

For page only. Section views on the page will always have different letters.

For the entire document. Letters that are used for section views will differ in the entire document.

Add scale automatically. When this flag is set, scale will be drawn on the projections automatically, if scale was changed.

Add sign of rotation angle automatically. When this flag is set, sign of rotation angle will be drawn on the projections automatically, if rotation angle was changed."Colors" Tab



Defines the color palette of the drawing and allows saving it with the drawing. These settings override the system settings of the respective parameters done by the command **Customize** > **Options** > **Colors**.

One color group defines color in the case of using one color throughout the drawing.

One color. Sets the drawing mode of using one color for all drawing elements. This color overrides element own settings.

Color. Defines the color for all drawing elements when one color mode is set. The color can be selected from the menu of colors.

Set colors group allows to define a color scheme that shall be stored together with the particular drawing.

Set colors. Setting this attribute allows to define colors for the following drawing elements:

Construction Line color. Defines the color of construction lines for the current page.

Outline color. Defines the color of the border of the drawing area (also called "format").

Grid color. Defines the color of the grid display.

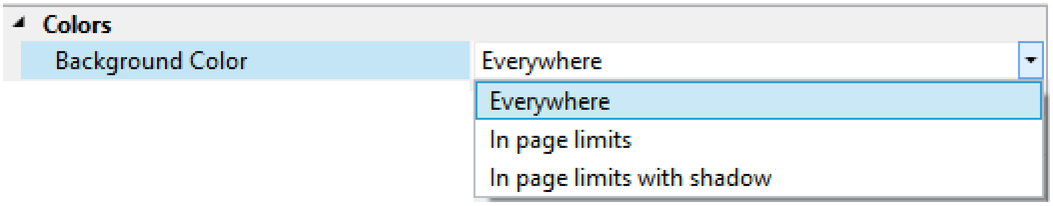
Node color. Defines the color of nodes.

The colors can be selected from the color menus for each item separately.

> Note that "**One color**" setting overrides the individual color settings for the elements with the background color, construction lines and nodes.

Background color. Defines the color of the current drawing background.

Elements with background color. Defines the color of elements whose color is the same as the current drawing window background. Important to keep in mind that each color in the system has an Id. Altogether there are 256 standard colors (Ids 0-255). It is possible that two colors visual perception is same while the Ids differ. For example, suppose, the background color is set to black with Id equal 0, while the element color is gray 100% that is also black, however, with Id 226. In this case, the element will not be identified as an element with the background color.



Colors:

Background color. Defines the color of the current drawing background:

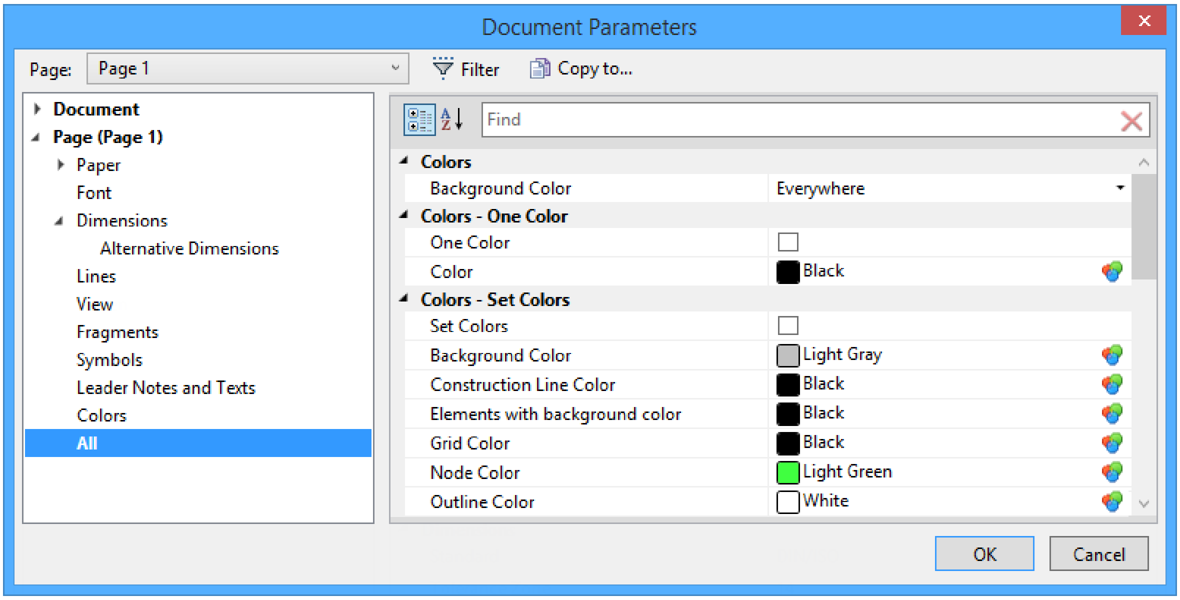
Everywhere. Background color is displayed inside and outside drawing border.

In Page Limits. Upon turning on this parameter, only the region *inside the drawing border* will be painted with the background color of the drawing.

In Page Limits with shadow activates the mode of displaying shades around the drawing border.

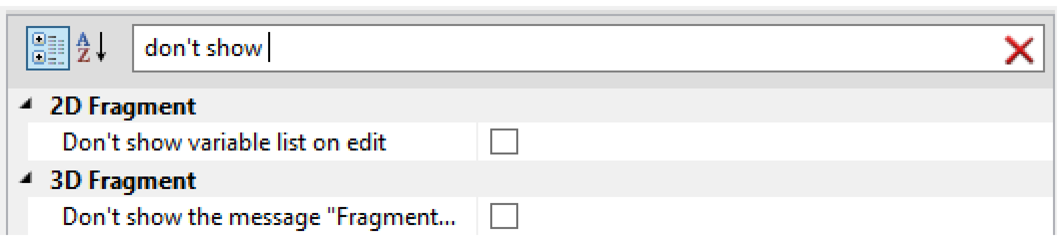
"ALL" TABS

All existing in the current group parameters are displayed on the **All** tab.



The parameters can be sorted by categories or alphabet .

You can enter parameter name or part of the name in the search bar to find it.

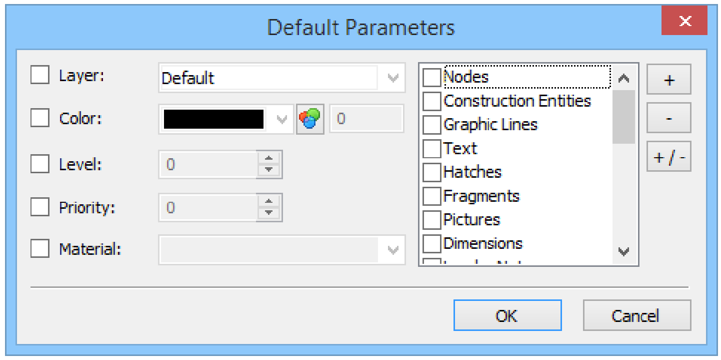


DEFAULT PARAMETERS

For convenience of drawing creation, common parameters can be defined for 2D and 3D drawing elements, such as color, layer, level and priority, and, additionally for 3D operations, the material. The selected parameters are assigned to the newly created elements by default, but only for those whose types were selected from the list.

The following command is used for setting the default parameters: **PD: Set Default Parameters:**

Icon	Ribbon
	Edit→ Document → Defaults
Keyboard	Textual Menu
<PD>	Customize > Defaults



The command brings up the "Default Parameters" dialog box.

The pane on the right-hand side of the dialog box contains the list of all drawing element types – construction, graphic, detailing elements, etc. Checkmark the elements intended for setting new parameters. To check all elements at once, use the **[+]** button. Vise versa, to clear all checks, press the **[-]** button. The **[+/-]** button inverts element checkmarking.

To change a parameter, set the checkmark before the parameter name. Otherwise, this parameter will not be accessible for editing. Each parameter affects its specific group of elements. For example, a material can be defined only for 3D operations, while the "Color" parameter does not affect construction entities.

Should different default parameters be assigned to different elements, call the "Default Parameters" dialog box over again.

For example, suppose, we need to set the default color blue for graphic lines. Besides, let the newly created detailing elements be moved on a new special layer. To do so, call the command **Customize > Defaults…** twice. On the first time, checkmark only the graphic lines in the list, check the "Color" parameter and define the desired setting which is the blue color. Then complete the first round by pressing **[OK]**. On the second time, checkmark the detailing elements – "GD&T Symbols", "Roughness Symbols", "Leader Note", "Dimensions", and clear the check on the "Graphic lines" element. Then check the "Layer" parameter and set as desired. Remember to first uncheck the "Color" parameter. To complete the second round, press **[OK]**. Now, when creating graphic lines, call the **Graphic Line Parameters** dialog box by typing <P> key and see that the "Color" parameter is set to blue color. Similarly, watch the new default parameters of the detailing elements.

It is possible to define the complete set of default parameters for a particular type of elements. To do so, call the element creation command. Then immediately call the "Parameters …" dialog box of this element and do the desired settings. Keep in mind that the "Parameters …" dialog box shall be called before creating a new element. In this way, the element parameters will assume the new default settings upon creating new elements of the given type. Otherwise, the ☐ Set as default item needs to be checked in the lower-left corner of the "Parameters …" dialog box.

While creating elements, some parameters can be modified on the system toolbar. If the parameters on the system toolbar are modified while within a creation or construction command (check the automenu), then these parameter settings are set as default for the current element type. In the case when the automenu is empty (no command is active), then after modifying a parameter the “Default Parameters” dialog box appears (see above) in order to define what types of elements are to assume the modified parameters as the default.

LIBRARIES

When working with assemblies, standard elements are often used as fragments or pictures. Libraries provide an orderly way of managing sets of standard element files.

A T-FLEX CAD library keeps the path to the folder with the standard element files. When selecting a file, a document preview and document properties preview are available in a separate window.

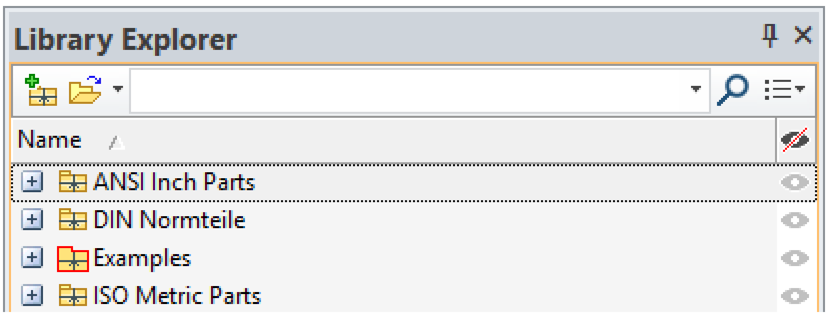
Sometimes, simultaneous access to several libraries is needed. T-FLEX CAD allows composing library configurations. Each configuration can include an unlimited number of libraries groups, each group containing a set of libraries.

LIBRARY CONFIGURATIONS AND LIBRARY EXPLORER

A library configuration is a convenient means of managing numerous documents and libraries of parametric elements that can be used as fragments.

Library configurations are stored in ".tws" files. A library configuration can consist of libraries or groups of libraries. Groups can include other groups or libraries. Therefore, library configurations can be structured hierarchically. A library contains data about paths to a folder on the disk, containing the document files. Therefore, for documents files to be included in the library simply place the files in the appropriate folder on the disk.

The actual management of libraries and included there document files is done via the library explorer.



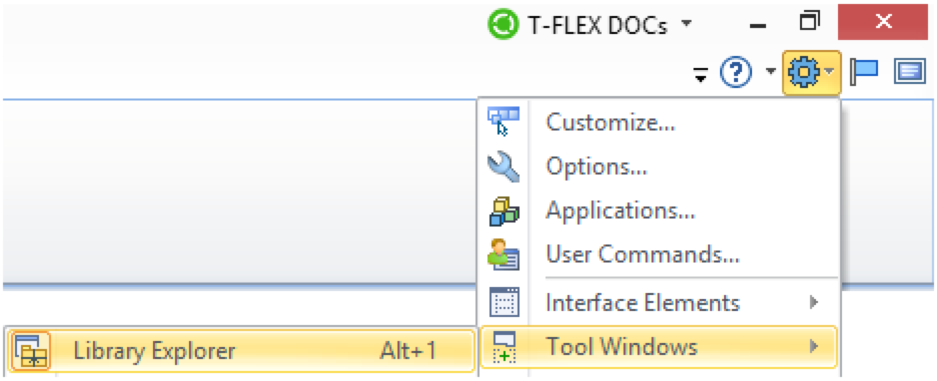
The library explorer is a window displaying the open library configurations with all its files. The user has access to files for opening and for inserting fragments and pictures in the commands **FR: Create Fragment**, **IP: Insert Picture**, **3F: Insert 3D Fragment**, **3MO: Insert External Model**.

Icons of library explorer elements:

Configuration	Library
Active configuration	Not found library
Libraries group	Duplicated library

Icon created in command **IC: Create/Edit icon** is displayed for files.

Library explorer window can be activated via > **Tool Windows** > **Library explorer**.



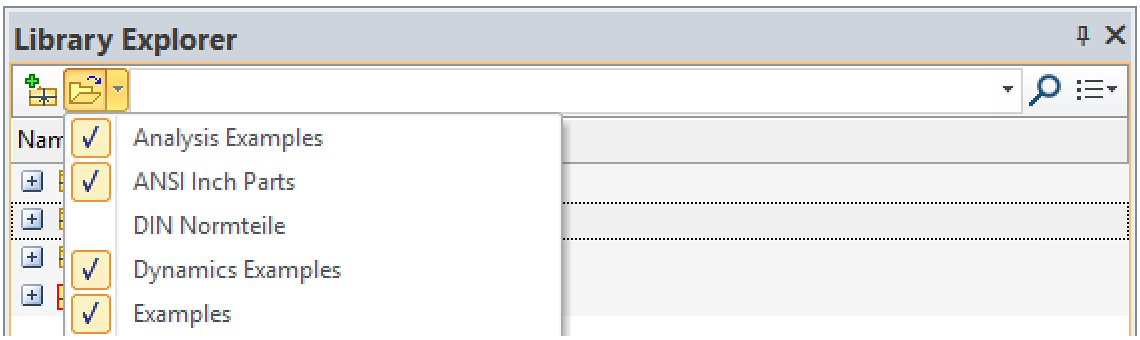
TOOLBAR OPTIONS

There is a special toolbar for **Library explorer** window.

The toolbar allows to:

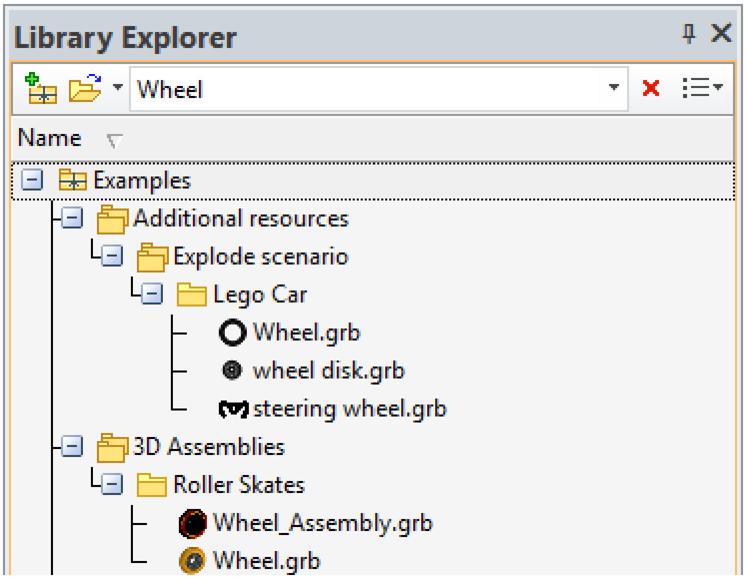
1. Create a new configuration .
2. Open a configuration .

Drop-down list of all installed library configurations appears after pressing down arrow next to **Open Configuration** command icon . It is helpful for opening and closing existing library configurations.



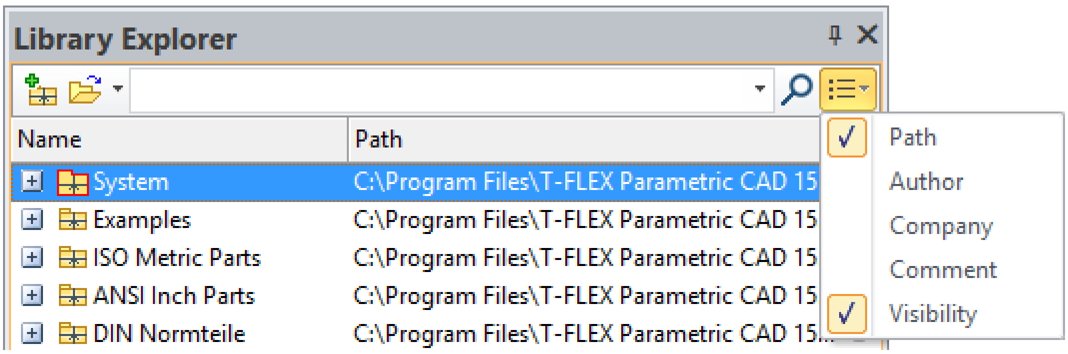
3. Use search bar .

When you start search, all documents, which names satisfy the search request are shown in the Library Explorer.

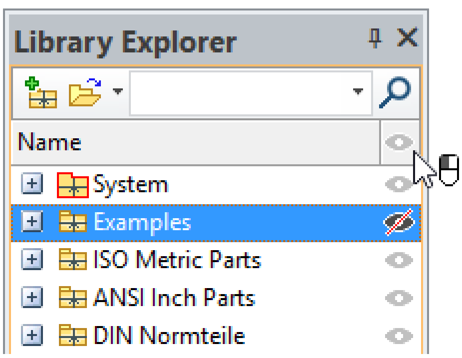


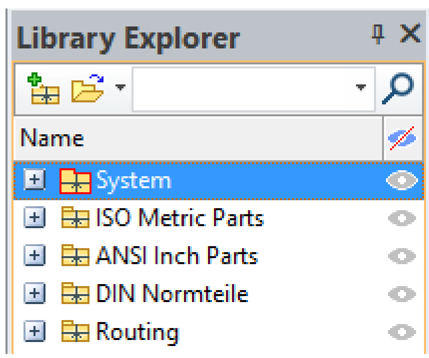
4. Show/hide elements information columns .

Columns contain information about library explorer elements. The information can be edited. Records can be sorted by columns.



"Visibility" column allows to hide configurations, libraries and files in the window. You need to click icon next to the element for this purpose. You may control visibility of the hidden elements by clicking icon in the column header.





"Path" column displays path to the library and allows to change it manually. There is an option ... in the column, which allows to select a new directory for the library.

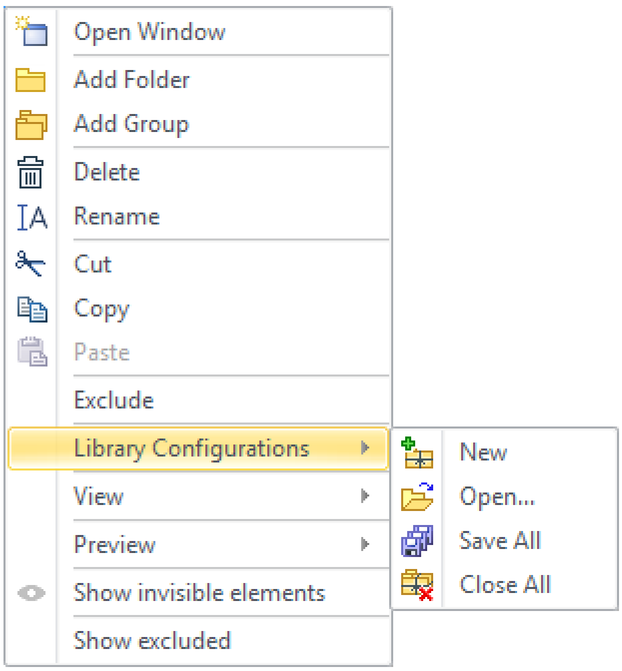
You can insert information for configurations, libraries and libraries groups in "Author", "Company" and "Comment" columns. The information will be stored in a configuration file.

CONTEXT MENU COMMANDS

Library configurations are filled via the library explorer. The library explorer commands are called by mouse right-clicking in the "Library explorer" window. The menu of the currently available commands pops up on the screen. The context menu contents depend on where the mouse cursor is pointing to at the moment (the window title, the name of a library configuration, a group name, a library name, filename, etc.).

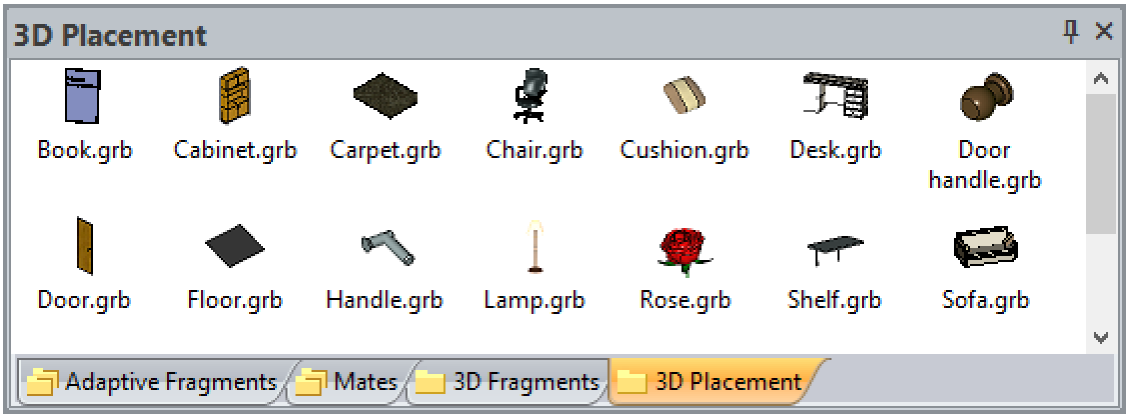
Common Commands of Context Menu

Context menu commands that can be used for different type of library explorer elements are described in the section.



Open Window . Opens a separate library window. This command is available in the context menu of libraries and libraries groups. You can also simply drag and drop the icon of the desired library from the library explorer into the document window.

Individual windows can be opened for several libraries. Like any service windows, windows with libraries can be combined into a common window with tabs.



Add Library /**Add group** . Allows to add a new library/libraries group. The commands are available in the context menu of configurations or libraries groups.

Open Folder . The command is available in context menu of configurations, libraries and files. A Windows folder with the selected configuration, library or document will be opened after the command activation.

Delete . The command deletes groups, libraries or library elements.

If you delete a group, all libraries and groups included in it will not be deleted from the configuration.

If you delete a library, its folder will not be deleted from the disk.

If you delete a file, it will be deleted from the library explorer and from the disk.

Rename . Use this command to rename library, group of libraries or file.

Library Configuration. Set of commands that allow to work with libraries configurations.

New. Creates a new library configuration. The command brings up a dialog box for inputting the name of the newly created configuration file and defining saving location on the disk.

Open. Opens the library configuration file. Several configurations can be opened in the **Library explorer** window.

Save All. Saves all open library configurations.

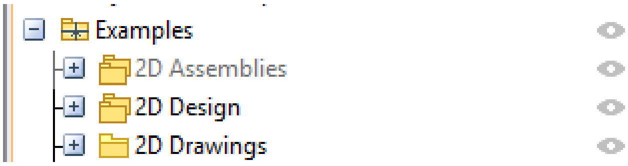
Close All. Closes all open library configurations. If changes were made to some library configuration, a query about saving changes will be made before closing.

The commands for managing configurations and libraries can be called from the main menu **File** > **Libraries**.

> Note that in absence of the **Library explorer** window in display, library configuration creation and loading operations will not be noticeable until the user opens the "Library explorer" window on the screen.

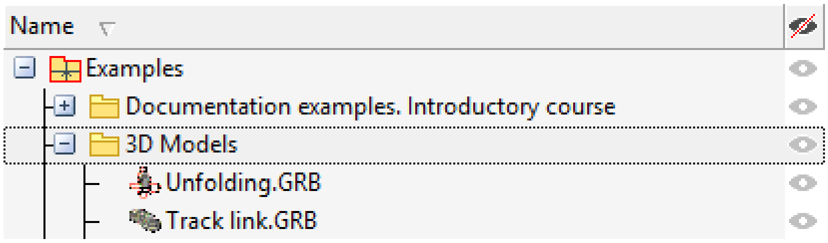
Exclude/Include. The option **Exclude** allows to hide selected files and folders. The option **Include** allows to show the hidden elements in the library explorer,

Show excluded/Hide excluded. When the option **Show excluded** is enabled, all hidden elements are shown in the library explorer with grey. Use the **Hide excluded** to hide them again.

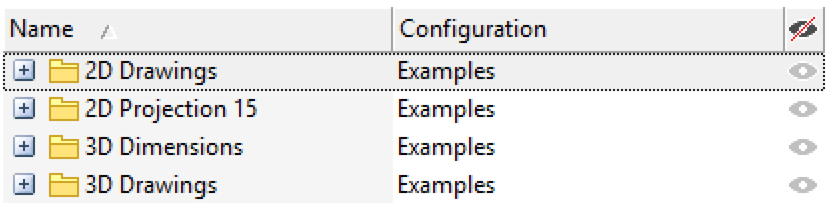


View:

Tree. Preset hierarchy of configurations, libraries groups, libraries and files is displayed in this mode.

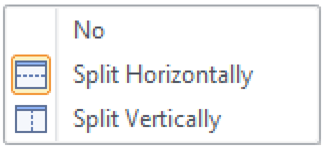


Libraries Only. Only libraries and their content are displayed in this mode.



Preview. The library explorer window allows opening an additional preview pane. The preview pane displays the file image or content of libraries/libraries groups.

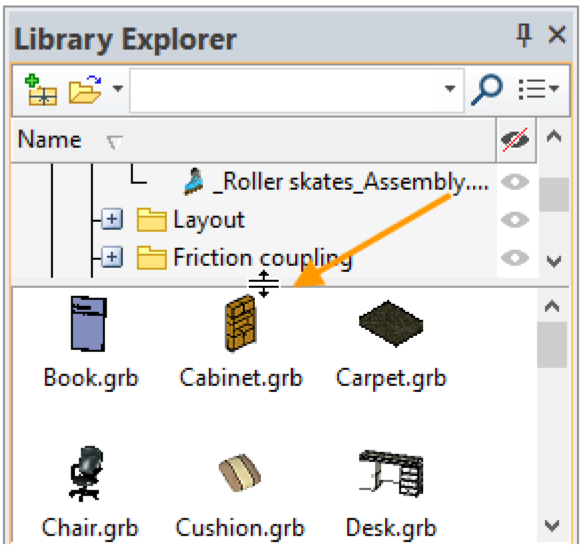
Position of the preview pane can be controlled by choosing the way the windows are split: **vertical** or **horizontal**.

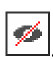


Note that for the preview to appear in the preview pane, it needs to be created first. The image can be saved either as a bitmap, or as vector graphics. Different way of saving preview may be preferable in different drawing situations. Parameters for saving the drawing preview can be specified in the command **ST: Set Document Parameters** on the tab **Save.**

For detailed information on saving a preview or icon image refer to the chapter "Preview/Slide".

You can change preview window size or hide it by moving separation line.



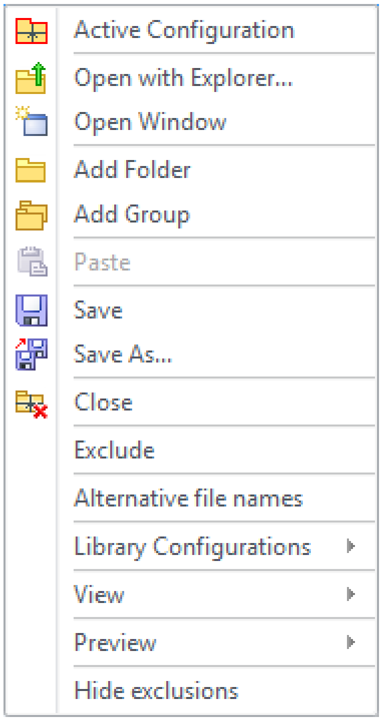
Show invisible elements / **Hide invisible elements** .The option appears in the automenu if there are any invisible elements in library explorer. The option allows show/hide the elements in the list.

Library Configuration Management Commands

Active Configuration . Marks selected configuration as active.

The search for a fragment file when opening or regenerating the document begins in the active library configuration. If the element is not found, the search continues in all the rest of open configurations.

If several library configurations are open then one of those is the active one, with the icon outlined in red.

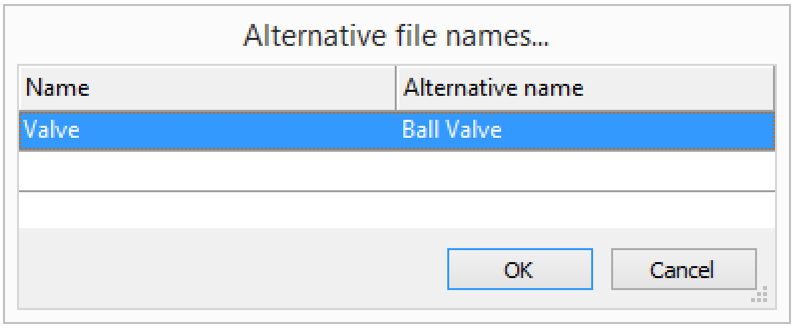


Save. Saves the changes in the current library configuration.

Save As. Saves the current library configuration in a new file.

Close. Closes the current library configuration.

Alternative file names. The command is used when you want to rename a file in the library and save links of the file with already existing assemblies. You should specify new and alternative names of the file in the command dialog.

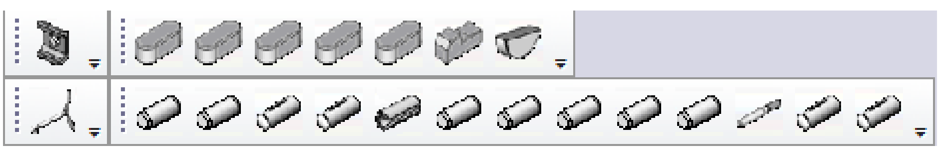


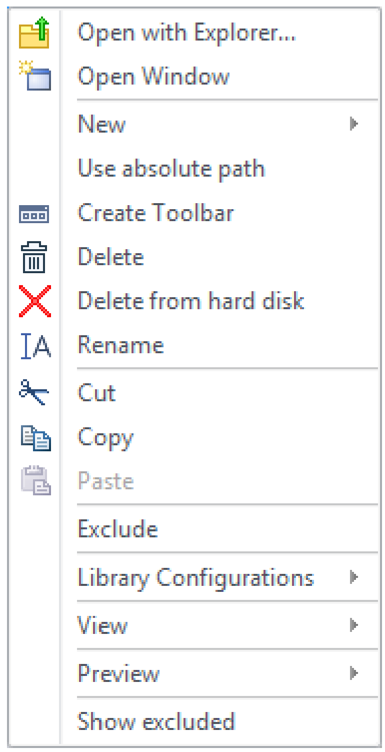
Assemblies will use the alternative name to search for the file, so it should match the old name.

Library Context Menu

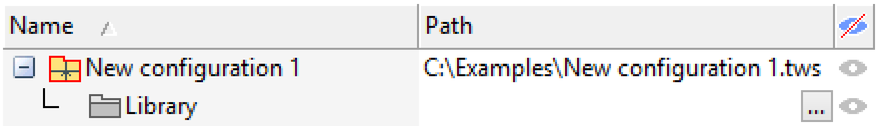
New command allows to create a new file in the library. You can select prototype from the drop-down list.

Create Toolbar . This command adds a new toolbar to the application interface. A separate button corresponds to each file from the selected library. If the document has an icon it will be displayed on the button. Name of the file is displayed on the button if there is no icon. Pressing the button will insert appropriate library element into the document as a fragment. This feature allows you to create customized toolbars based on standard and custom libraries.

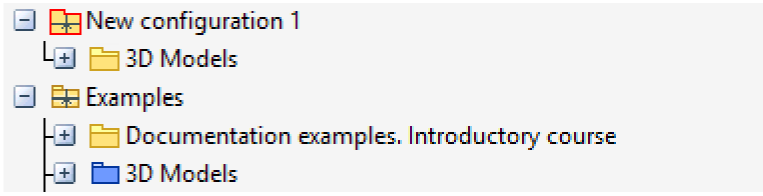




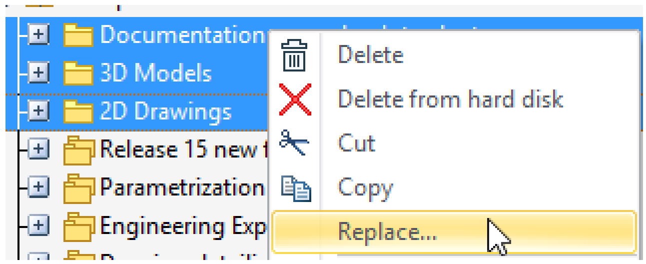
Delete from hard disk ☒. The option allows to delete libraries folders from the disk. An empty library will exist in library explorer. Its folder is colored grey. This library can be linked with folder on the disk again. For this purpose you can manually insert new path to the folder or use option [...].



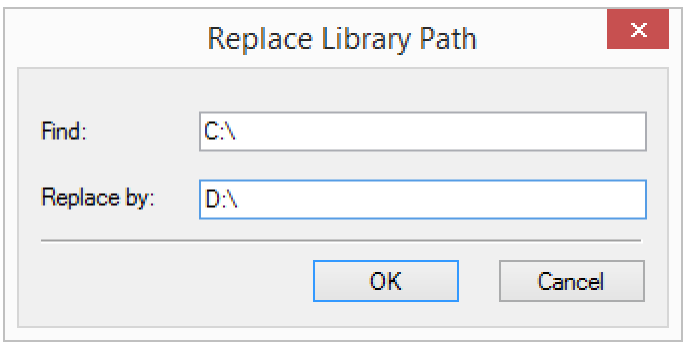
If there are libraries with duplicated names in library explorer, they are pointed with blue folder icons.



Replace command appears in context menu when you select several libraries. It allows to change path or its part for all the selected libraries.

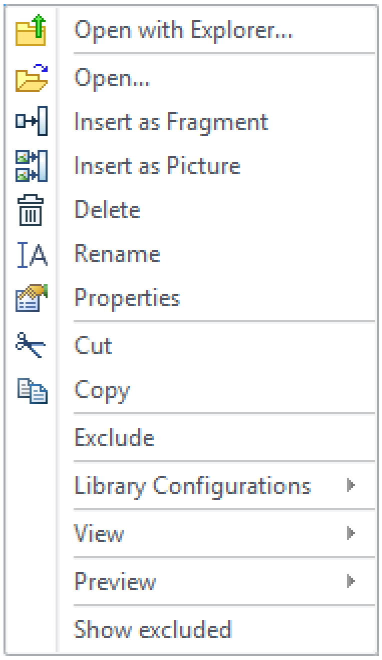


Replace Library Path dialog apears after its activation.



File Context Menu

Open command allows to open the selected file in T-FLEX CAD.

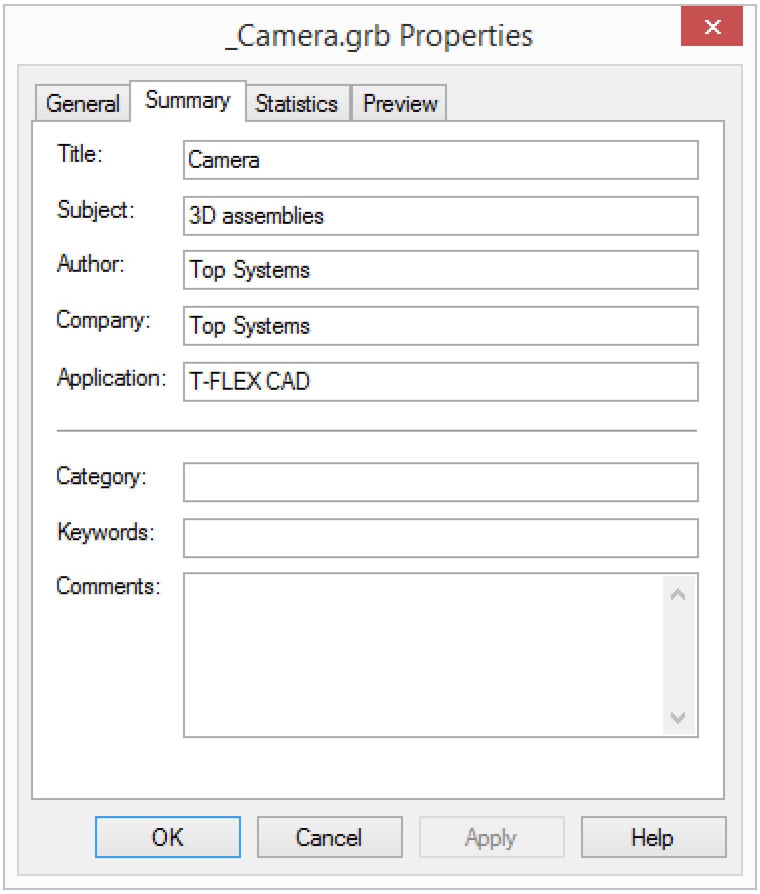


Insert as Picture activates **IP: Create Picture** command and inserts the selected file as a picture into the current open document.

Insert as Fragment allows to insert the selected file as a fragment into the current open document. **3F Insert Fragment** or **FR: Create Fragment** command is activated according to the active - 2D or 3D - view in the document.

> A library name can be used in the commands for inserting external files into a T-FLEX document, such as **FR: Create Fragment, IP: Insert Picture, 3F: Insert 3D Fragment**. For example, if a fragment drawing fragment.grb is located on the hard disk in the catalog `C:\TFW32\LIB\FRAG`, and there is a library named “Parts” referencing this path, then the following name can be used for inserting the fragment: “<Parts> fragment”. This way is convenient not only in that it replaces a long and possibly kludgy path with a slick one, but also helps to save paths to fragment files upon transferring documents to another disc or computer. You just need to open the library configuration.

Properties. Calls the property dialog box of the selected file.



To have complete information displayed about the drawing, fill the informational fields **PS: Show Model Properties**. To make a preview appear in the preview pane, create it by the command **PV: Save Preview**.

Creation of Configurations, Groups of Libraries and Libraries

You can use option **New configuration**  on the toolbar to create a new library configuration. You can also use **Library configuration > New** item from the context menu.

Command **Add Library** / **Add Group**  adds a new library/libraries group. The commands are available in context menu of configuration or libraries group. The element is created on the current library configuration hierarchy level.

When you create library it is necessary to select folder with necessary files.

> An example of library explorer elements creation can be found in "Example of configuration creation" section.

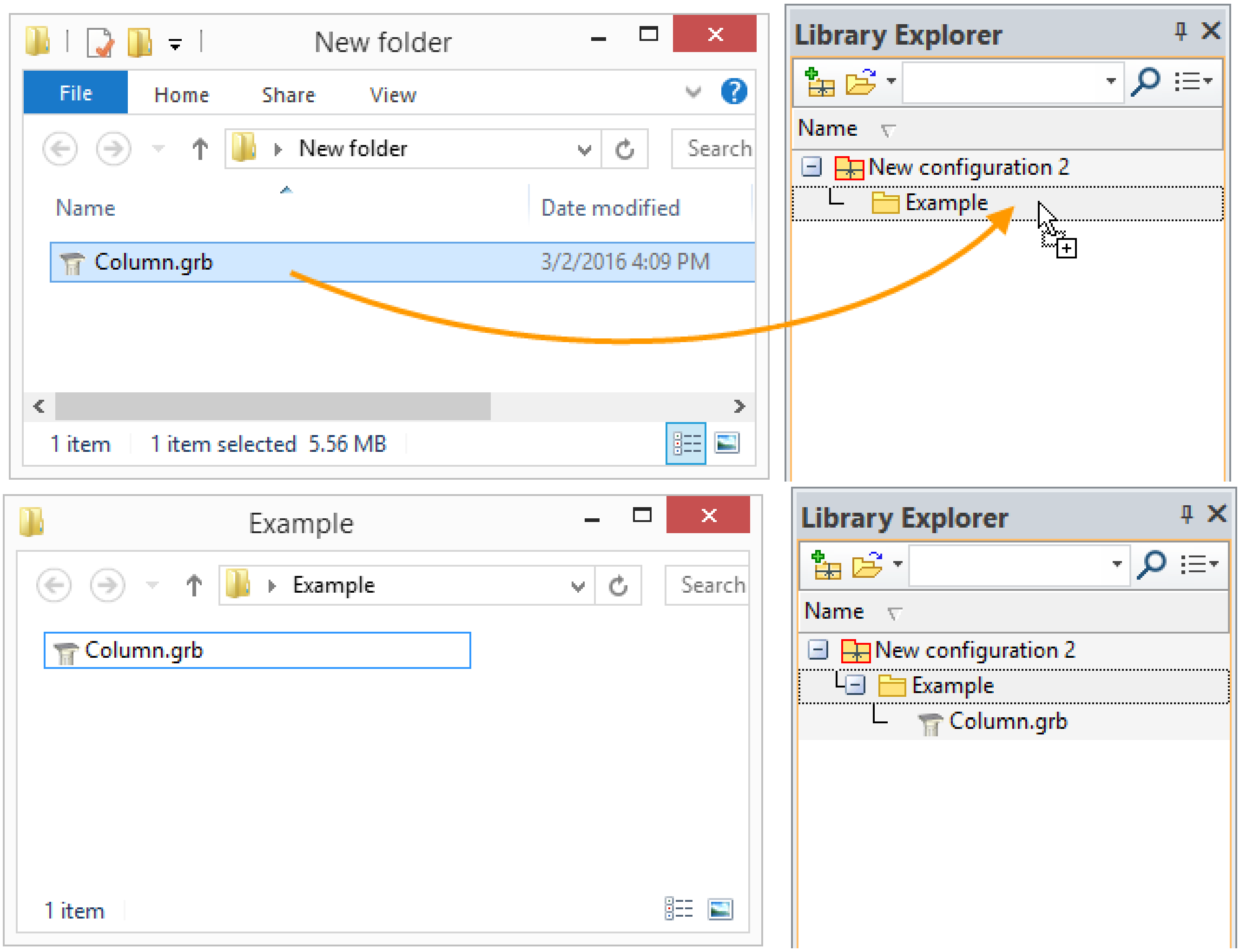
ADD FILES TO LIBRARIES

There are several ways to add a new GRB file into the library:

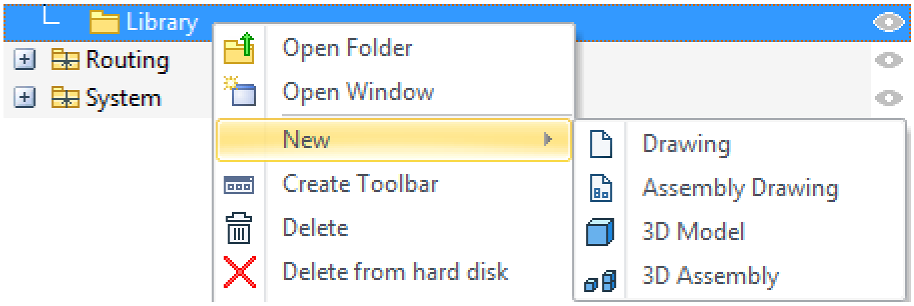
1. Add file to the catalog, which is linked with the library.

Library Explorer window is automatically updated if document files included into library were deleted, added or renamed. Thus, files added to the folder in Windows Explorer are displayed in the Library Explorer window.

2. Transfer files from Windows explorer to Library explorer window using Drag'n'drop or Copy/Paste options.



3. Create a new file using one from the prototypes using option **Create** from the context menu.



Moving of Library Explorer Elements

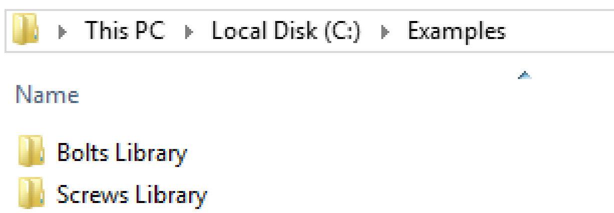
You can move libraries groups, libraries and files using Drag'n'Drop mechanism. For example, to move several files into another library, first select the group of files (hold down **<Shift>** or **<Ctrl>** key as appropriate). Then, move the cursor over the highlighted files and press . Next, hold the left mouse button down and drag the group of files into another library. This amounts to performing first **Cut**, and then **Paste** command.

Cut , **copy** , **paste** . These commands are available in context menus to move/copy libraries, libraries groups and files.

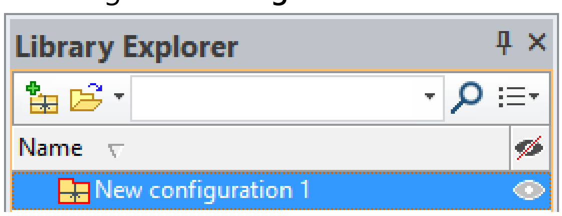
> If a file is dragged into the drawing window, it will be inserted as a fragment.
> Dragging a file icon into the menu or toolbar area opens it for editing.

Example of configuration creation

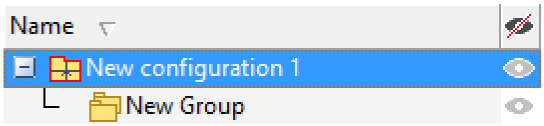
We need to create a new configuration and add "Bolts Library" and "Screws Library" into it. The folders include all necessary files and are stored on the disk.



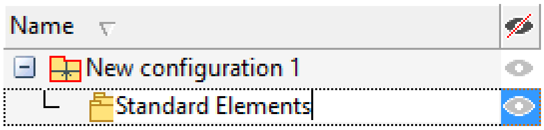
1. Create a new configuration using **New Configuration** command from the toolbar.



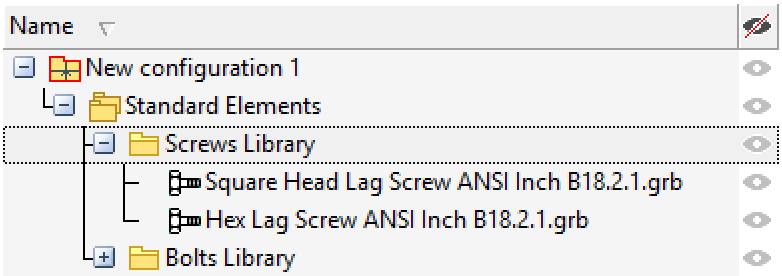
2. Create libraries group using **Add group** from the configuration context menu.



3. Rename the created libraries group using **Rename** command from the context menu.



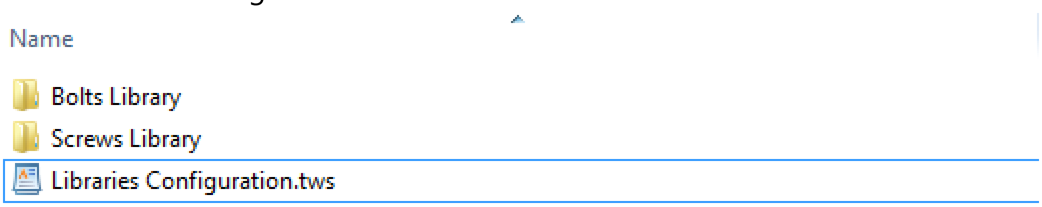
4. Select **Add Library** item from context menu of the libraries group. **Select Library Folder** dialog will appear. Select folder and press [Select folder]. The selected folder with all its files will be added to library explorer.



All the files that will be subsequently added to the selected catalogs will also be displayed in library explorer.

5. Save the configuration after adding all necessary libraries. Configurations are stored in textual TWS file, which contains information about included libraries. To save the configuration, use **Save as** command from the context menu of the configuration and select folder where it will be stored. You can also change name of the configuration upon saving.

It is recommended to store configuration files and their libraries folders on the same level.



SPECIFYING RELATIVE PATHS TO THE CATALOG

A relative path to the library or libraries group catalog can be specified with respect to the installation home of the T-FLEX CAD application, or with respect to the configuration file that includes this library. A relative path allows use of symbols “*”, “..” and “.”.

The symbol “*” in the beginning of the path stands for the path to the T-FLEX CAD home.

The symbol “.” denotes the path to the folder of the library configuration file.

The symbol ".." means ascending one level up the folder.

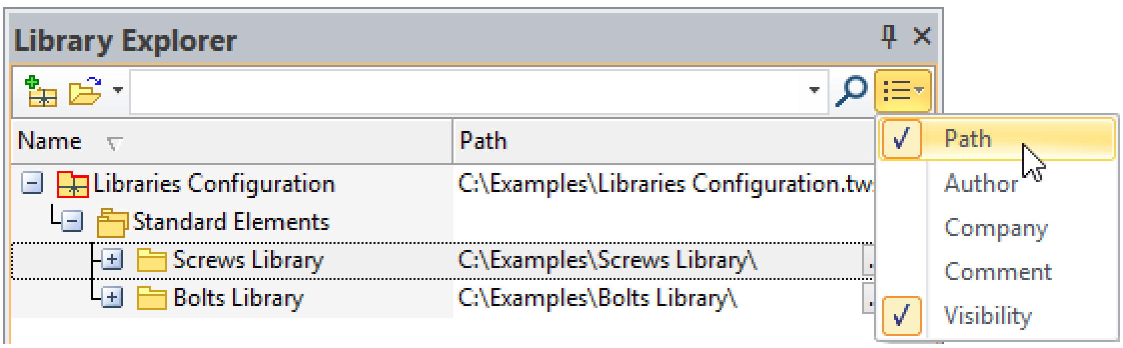
Suppose, for example, that the T-FLEX CAD application is installed in the `C:\Program Files\T-FLEX Parametric CAD 3D` folder, and a library configuration is stored in the file `C:\Library\A library1.tws`. In this case, the relative paths to libraries and groups of libraries are written out as follows:

Path relative to T-FLEX CAD home folder	**Path relative to the library configuration file**	**Absolute path**
`*\Library\Bolts`	`.\..\ Program Files\T-FLEX Parametric CAD 3D\Library\Bolts`	`C:\Program Files\T-FLEX Parametric CAD 3D\Library\Bolts`
`*\..\..\Screws`	`.\..\Screws`	`C:\Screws`
`*\..\..\Library\Rods`	`\Rods`	`C:\Library\Rods`
`*\..\..\Library\Bolts\Normal`	`\Bolts\Normal`	`C:\Library\Bolts\Normal`

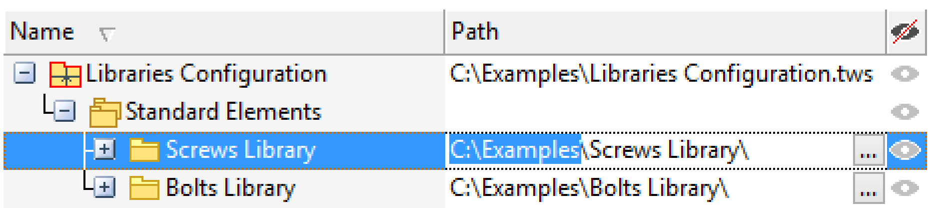
In the case of inputting a path to a non-existent folder, the system changes library icon to grey folder.

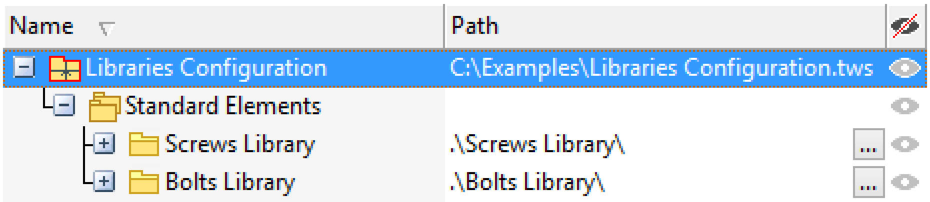
You can change paths using one of the following ways:

Activate "Path" column displaying in library explorer. Here you can change path to the folder.



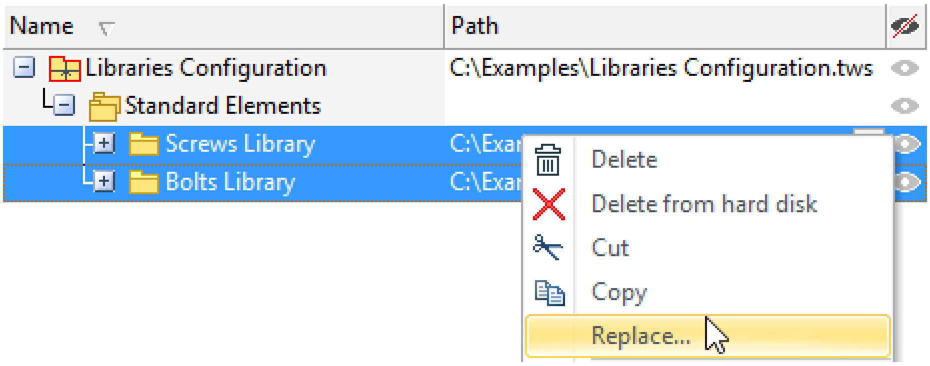
Configuration file is located on the disk on the same level with its libraries. So that you can change path from "C:\Examples" to ".".



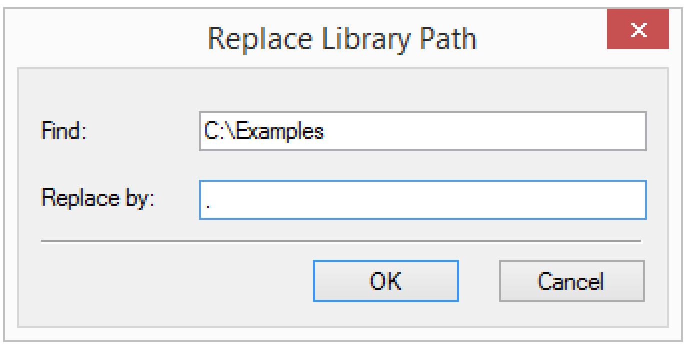


Or

Change paths for several libraries at once. You need to select the libraries with hold down <CTRL> button and select **Replace...** item in their context menu.



Replace Library Path dialog appears. Here you can input an old path or its part and replace it with a new one.



After that select **Save** option in the context menu of the configuration to save configuration changes. Relative paths will be saved in the configuration file.

PAGES

For working convenience, T-FLEX CAD system provides a capability of creating multi-page documents. For example, it is handy to have in one file, yet on separate pages, auxiliary 2D constructions used in the main 3D model, as well as projections and sections of the 3D model with dimensions, or the BOM, etc. The elements can meanwhile interact with each other via the created relations, variables, databases, etc.

GENERAL INFORMATION

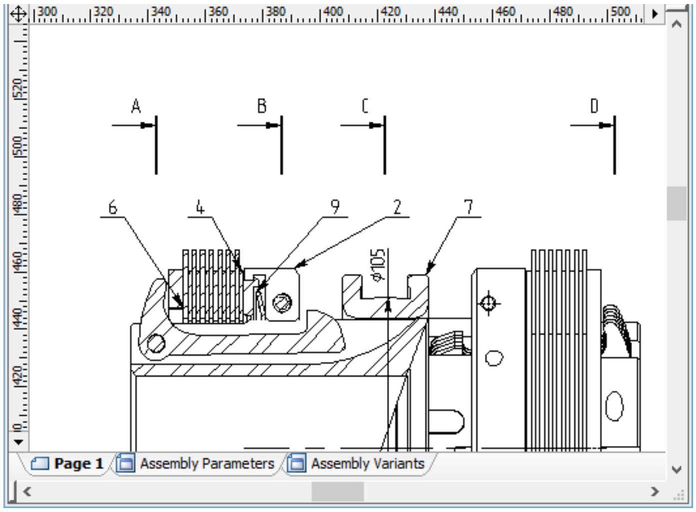
A T-FLEX CAD document can have an arbitrary number of pages. Each created document has at least one page.

Pages in T-FLEX CAD are divided into six types, depending on their purpose and the way of creation: “Normal”, “Auxiliary”, “Controls”, “Workplane” , “Text”, “Bill of Materials”. Such division is not strict: the type of a page in most cases can be changed by the user. Types help controlling page display. Depending on the drawing settings, the 2D window will display all pages in the document, or only the pages of certain types. Therefore, the user can manage page visibility while working with multi-page documents, hiding from display those unused at the moment.

Each page uses its own drawing settings defined in the command **ST: Set Document Parameters**, such as the paper format, drawing scale, font parameters, detailing elements, colors, properties of element display, etc. The settings made in the command **Customize** > **Levels…** also affect only the page that was current when the settings were made. When calling **File** > **Export** command, the system processes the current page of the drawing (except exporting to the AutoCad format, when all pages of the documents are getting converted).

> Modifications of “default” parameters affect all pages (see “Default parameters” section of the “Customizing Drawing” chapter). The same is true for managing layers in the command **QL: Configure Layers**.

If a T-FLEX CAD document contains several pages, the tabs with names of visible pages are displayed in the bottom of the drawing window. Visibility of the tabs can be controlled with the help of the flag **Customize** > **Tool Windows** > **Page Tabs** found in the textual menu.

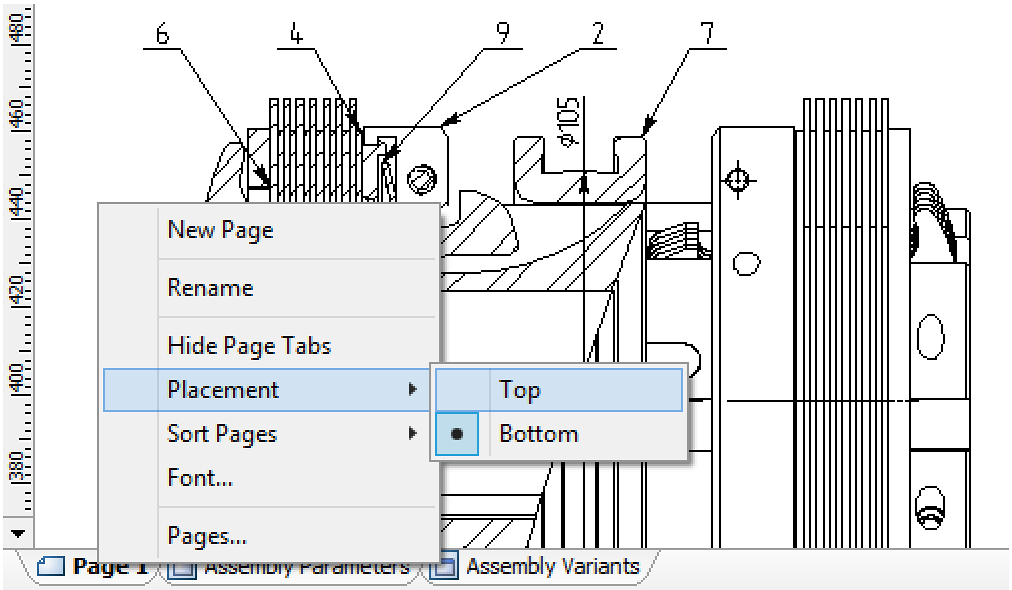


Switching pages is done without leaving the current command. This help creating, say, a copy or a detail view by selecting elements located on different pages.

Managing Document Pages

Working with Page Tabs. Tabs Control

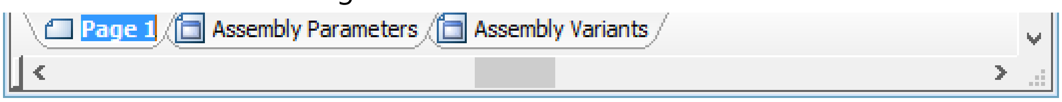
Page tabs, by default, are located along the lower border of the T-FLEX CAD document window. If necessary, their location can be changed by moving the page tabs upwards, as it was done in the previous versions of the T-FLEX CAD. To do it, it is enough to point with the cursor at the page tab, call the context menu with the help of and change the value of the toggle **Tabs placement** to the required one (**Top** or **Bottom**).



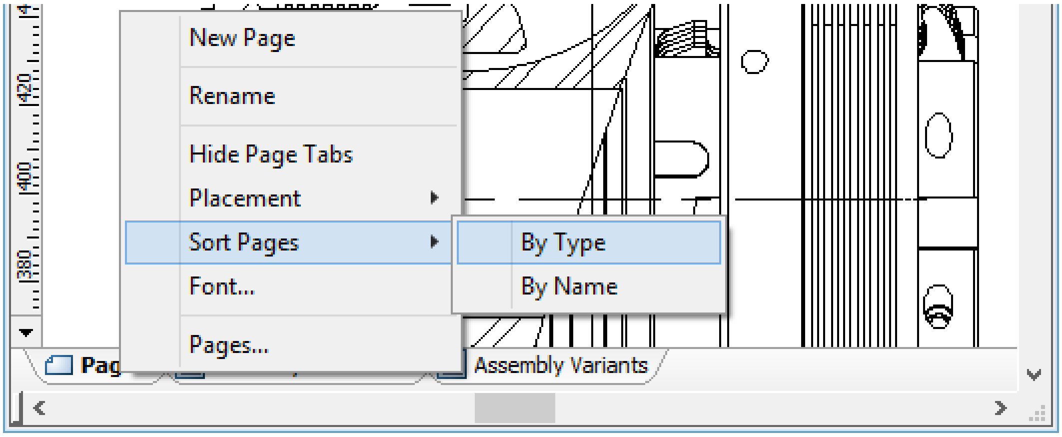
Page tabs are used for quick shifting between pages of the current document. For shifting to the required page of the document, it is enough to point with the cursor at its tab and press . Moreover, for shifting between pages the keys <Page Up>, <Page Down> can be used as well.

If the <Page Down> is pressed while being on the last page of the current document, the system will offer to create a new page. The created page will be placed at the end of the list of pages of the given document. It is also possible to create a new page by using the command **New Page** found in the context menu for the tab of any page.

On each page tab, the page name as well as the symbol showing its type are displayed. By default, the system gives to the pages the names “Page 1”, “Page 2”, etc. , however, afterwards these names can be changed to more meaningful ones to make the work more convenient. To rename the page, the command **Rename** can be called from the context menu of the page tab. After calling this command, the system will shift to the mode of editing the text shown on the tab.



The order in which the pages of a document are arranged can be changed. It is possible to quickly sort the pages by names or by types with the help of the commands **Sort Pages** > **Name** and **Sort Pages** > **Type** found in the same context menu called for any tab of the current document.



An arbitrary change in the page arrangement can be carried out just by dragging the page tab to another place. To do it, bring the cursor to the required tab, press and without releasing the mouse button, drag the cursor to the required position in the list of tabs.

Let's talk more about the context menu for the page tabs. With the help of the command **Font…** it is possible to modify font parameters used for displaying the text on the page tabs. Upon calling this command the standard window of font parameters opens up.

The command **Delete** allows removing the page on the tab of which the context menu has been called. If the page being removed contains some drawings, the system will inform about that and ask a user to confirm the necessity of page removal and all its contents.

The command **Pages…** invokes the dialog for controlling pages of the document. This dialog allows performing all possible operations with pages: create and delete pages, rename, change the type of page and its location in the list of pages. We will tell more about working with this dialog in the section "Working with Dialog «Pages»" below.

The last command of the context menu – **Hide Page Tabs** – turns off the display of the page tabs for the document. After the tabs were turned off, it is possible to turn the display of the page tabs back on with the help of the flag **Customize > Tool Windows > Page Tabs**.

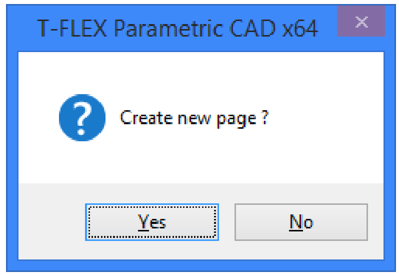
Creating New Pages

Additional pages can be created in a T-FLEX CAD document by the following means:

1. Using the <Page Down> key. When created, such pages are automatically assigned the type "Normal".
2. Via the help of the command **New Page** in the context menu of the page tabs;
3. *Via the command* **PG: Pages** (see the section "Working with Dialog «Pages»");
4. Automatically while working in the certain commands. The type of a thus created page depends on the actual command. For example, the **TR: Create Control** command allows creating a page of the type "Controls", while the command **SD: Create Darwing View**– a page of "Auxiliary" type. The command **3W: Construct Workplane** creates pages of the type

"Workplane", while "Normal" type pages are created under **BC: Create Report/Bill of Materials** and **BM: Reports/Bills of Materials**.

The quickest way of creating a new page is use of the <Page Down> key. To create a new page, go to the last page of the drawing and press the <Page Down> key. After that, confirm the new page creation query. The new page is created as a result. It is automatically assigned a new name "Page N", where N is the page number.



At creation, the page settings are copied from the last active page. These settings can later be modified by activating the page and calling the command **ST: Set Document Parameters**.

Creation of pages as per the item 3 is described in the respective chapters of the mentioned commands.

Working with Dialog «Pages»

The dialog "Pages" allows performing practically all possible operations with the pages of the current document. Several options are available to call this dialog. First of all, it can be called with the help of the command:

Icon	Ribbon
	Edit → Document → Pages
Keyboard	Textual Menu
<PG>	Customize > Pages

Moreover, the dialog "Pages" can be called via the toolbar "View" and from the context menu for the document tabs.

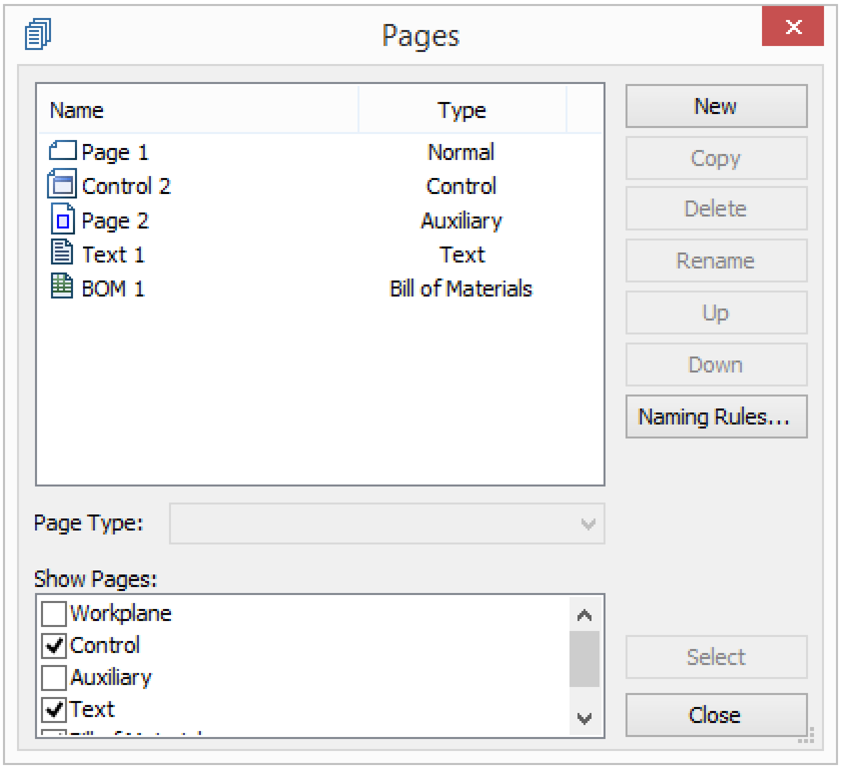
The main pane of the dialog contains the list of pages displayed in the 2D drawing window: names and types of pages.

In the bottom portion of the dialog box is the "Show Pages" group of flags for managing the list contents (and, therefore, the 2D window display) regarding the pages of a certain type, as follows: "Workplanes", "Controls", "Auxiliary", "Text", "Bill of Materials".

Pages of the type "Normal" are always present in the list and in display in the 2D document window.

Having selected a page in the list by clicking, you can change its type by choosing the required value from the drop down list "Page Type".

Pages of the type "Workplane" are created only with a workplane creation by the appropriate commands. Their type can't be changed.



The graphic buttons at the right-hand side of the dialog box allow the following actions over the selected page in the list:

[New]. Creates a new page of the drawing. Upon pressing the button, a new page is added in the document, launching the name editing mode. The page type is “Normal” by default. If necessary, you can assign the desired type via the parameter “Page Type”.

[Copy] Creates copy of the selected page with its parameters.

[Delete]. Deletes the drawing page selected in the list. If the page is not empty (i.e. contains some elements), the system queries the user whether to delete objects on the page or associated with the page. Positive answer results in deleting the page and all associated objects, the negative one cancels deletion.

[Rename]. Changes the name of the page. The page name is displayed on the tab. Clicking on the name of the selected page entry in the list highlights the name and allows its editing. When pages of the type “Workplane” are renamed, the “Name” parameter of the workplane changes according to the new name of the page.

[Up]. Moves the selected page entry in the list one line up.

[Down]. Moves the selected page entry in the list one line down.

Note that the order of pages in the document, as reflected by the tabs, corresponds with their positions in the list being described.

[Select]. Activates the selected page. Opens the selected page in the 2D drawing window.

[Close]. Closes the “Pages” dialog box. Completes the command.

Modifying Page Parameters

Each page has its own settings defined via the command **ST: Set Document Parameters.** The page size can also be modified directly in the 2D window by the command **PZ: Set Paper Size**:

Icon	Ribbon
	Workplane → Modes → Page size
Keyboard	Textual Menu
<PZ >	Customize > Page Size

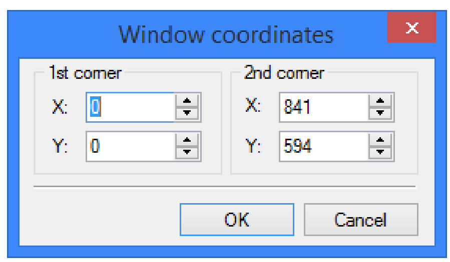
The command is used for modifying the size of a page and its position. Modifying the page size parameters affects the **Paper size** parameters on the **Paper** tab of the command **ST: Set Document Parameters**.

Upon calling the command, the borders of the current page highlight, outlined by a box with graphic controls of a square shape along the perimeter for resizing. If pointed to one of these squares, the mouse cursor changes to . It can be dragged now with the left mouse button depressed, modifying the size of the box diagonal. If pointed to a box mid-side square, the mouse cursor changes to and can be dragged now with the left mouse button depressed, modifying the vertical or horizontal size of the box, respectively. If within the box, the mouse cursor changes to and can be dragged now with the left mouse button depressed, modifying the box position.

The following options are available in the automenu in this mode:

	<Ctrl+ Enter>	Finish input
	<P>	Set page size
	<Esc>	Exit command

The option allows entering numerical values of the page corners in global coordinates. Upon modifying the size and/or position of the page, confirm the input with the option .



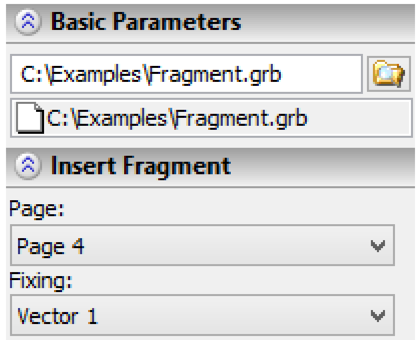
The grid can be brought up on the active page, and grid snapping turned on, by using the command “QG: Change Grid settings”.

Special Handling of Multi-Page Documents

As mentioned above, each page of a multi-page document in T-FLEX CAD can be used for creating various elements: nodes, construction and graphic entities, hatches, text, etc. if necessary, elements can be moved/copied from one page to another.

Elements located on different pages of the same document can be totally independent of each other, or, vise versa, related by various means: by the copy operation, by projective relations, via variables and databases. Besides, T-FLEX CAD allows creating elements located on several pages simultaneously. When creating text or BOM elements, one can define transition to the new page if the element being created is getting out of current bounds. As a result, each page will display the respective part of the multi-page element. Editing and parameter modification commands work on the whole element, regardless of the portion selected in the command.

A multi-page document, just as any other T-FLEX CAD document, can be inserted into another document as a fragment or picture, exported in other graphic formats or printed. Each of these situations will be reviewed separately.



When using a multi-page document as a fragment in an assembly, only one page of the fragment document is inserted. Therefore, when inserting, the user can specify the particular page of the fragment drawing to assemble, along with specifying fixing points or vector and defining the fragment variables

When inserting a multi-page document in another drawing as a picture only one page of the selected document is inserted as well. The required page is determined by the user upon inserting a picture.

When exporting a multi-page document into another graphic format, the output graphic file will contain the document page that was active when the export command was called (except exporting to the AutoCAD format).

When printing a document, the user can specify the pages to print in the print parameters dialog box.

Drawing Creation

Construction Entities

LINES

By "line", we mean infinite lines that belong to construction entities and serve as the parametric framework of a drawing. Lines are displayed as thin dashed lines.

Constructing Lines

To construct a line, call the command **L: Construct Line**:

Icon	Ribbon
	Draw → Construct → Line
Keyboard	Textual Menu
<L>	Construct > Line

The following options become available:

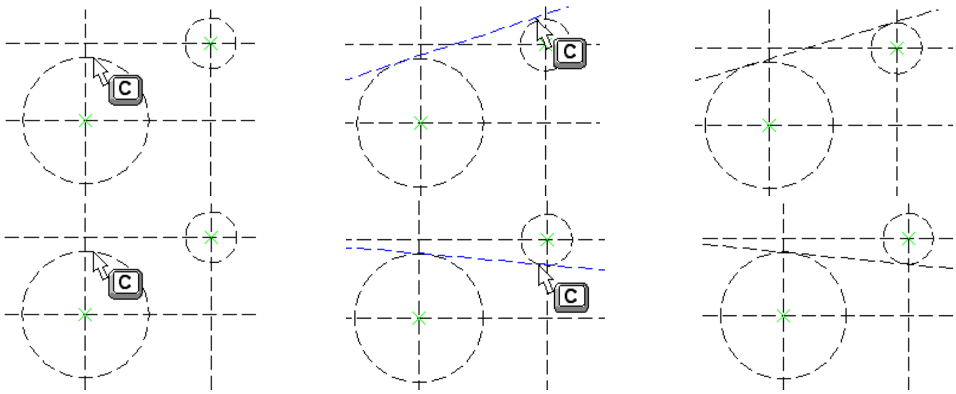
	<P>	Set Line Parameters
	<X>	Creates two crossing Lines and Node
	<H>	Creates horizontal Line
	<V>	Creates vertical Line
	<L>	Selects related Line
	<N>	Selects Node
	<C>	Selects tangent Circle
	<E>	Selects ellipse to create Line
	<S>	Selects spline
	<O>	Create orthogonal Line
	<T>	Creates Proportional Line
	<U>	Create Axis of Symmetry
	<A>	Selects Axis of Symmetry (straight Line)
	<W>	Select 2D projection
	<O>	Create two Lines and Node in (0,0)

	<Space>	Creates a node at the nearest intersection of construction lines
	<F> or <Ctrl><1>	Create insertion Point for Fragment (xn, yn)
	<F4>	Executes edit Construction command

Some of these options become available only after selecting certain construction entities.

There are different techniques of creating lines. Some lines are independent of other construction entities. These could be a standalone horizontal or vertical line. Usually, these are the very first lines on a drawing. By creating a vertical and a horizontal line, you define the base lines, to which all the rest will be related.

Other lines require the related elements to be selected at the time of creation. For example, a line tangent to two circles requires the circles to be selected and the tangency condition defined.



A number of line creation techniques require a certain geometric parameter to be defined. For example, consider constructing a line parallel to another line and passing at a certain distance from the other line. In this case, it is necessary, besides selecting the original line, to define the distance between the lines.

Exact values of the numerical geometric parameters can be entered in the property window in transparent mode. Besides, one can use the line parameters dialog box under the option , that, besides the geometric parameters, also allows defining general system ones, as level, layer, etc. If exact value is not required, one can simply point at the desired location on the drawing and click . In this case, the numerical parameter value is defined by the cursor position.

When constructing lines, one should keep in mind that after creating one line the command sticks in the selected line type creation mode. For example, once a pair of crossing lines was created under the <X> option, another pair of crossing lines can be created again without re-selecting the option. This feature helps speed up constructing same-type lines. To quit such a mode, right-click .

The line creation command allows making a variety of construction line configurations by combining the limited set of options, as follows:

<X>,<P>	Crossing (vertical and horizontal) lines with a node at the intersection and exactly defined placement coordinates
<H>,<P>	Horizontal line with exactly defined coordinates
<V>,<P>	Vertical line with exactly defined coordinates
<L>,<P>	Parallel to a line, the specified distance away
<N>,<P>	Line at a specified angle with respect to X axis
<N>,<L>,<P>	Through a node, at a specified angle with respect to a line
<N>,<L>,<O>	Through a node, orthogonal to a line
<N>,<N>	Through a pair of nodes
<H>,<N> or <N>,<H>	Horizontal line through a node
<V>,<N> or <N>,<V>	Vertical line through a node
<L>,<N>	Parallel to a line, through a node <*>
<C>,<C>	Tangent to two circles
<N>,<C> or <C>,<N>	Through a node, tangent to a circle
<A>,<L>	Symmetrical to another line <L> with respect to a specified axis <A>
<C>,<L>,<P>	Tangent to a circle, at a specified angle with respect to a line
<U>,<L>,<L>	Symmetry axis for a pair of lines
<L>,<C>	Parallel to a line, tangent to a circle <*>
<T>,<N>,<N>,<P>	Line orthogonal to the segment spanning two nodes, dividing the segment in specified proportion
<E>,<C>	Line tangent to an ellipse and a circle
<E>,<E>	Line tangent to two ellipses
<E>,<S>	Line tangent to an ellipse and a spline
<S>,<S>	Line tangent to two splines
<L>,<E>	Line parallel to another line and tangent to an ellipse
<E>,<P>	Line tangent to an ellipse, at a specified angle with respect to another line

<*> - Use of <L> equivalent to <Enter> or .

> Note: Whenever the property window or the parameters dialog box is used for defining a numerical parameter of the line being constructed, variables or expressions can be entered as well as fixed values.

Whenever the <P> option is not present, the line does not have numerical parameters. For example, this would be a line trough a pair of nodes.

Let’s review in details each of line creation techniques mentioned above. In following examples, we will show how to define these relations using the keyboard. Alternatively, one can work with the options via the automenu icons. Additionally, if object snapping is turned on, then the described actions of the command can be performed without use of icons or keyboard.

Line Construction Techniques

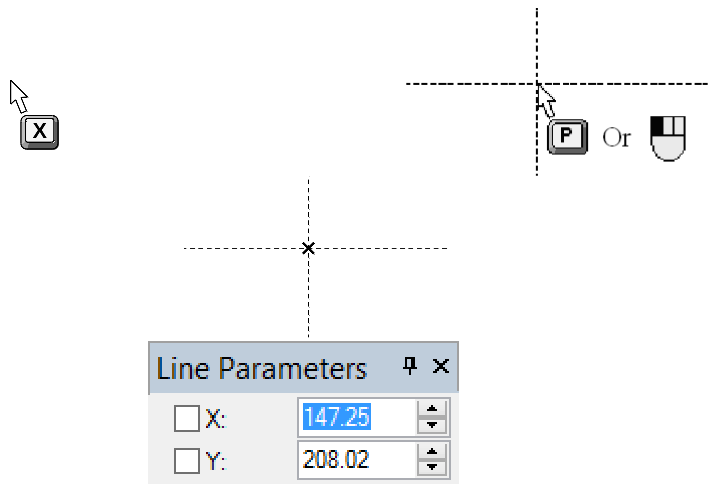
Follows is the description of various line construction techniques. Each technique implies using a sequence of certain options that include keyboard input and automenu icon picks.

The <P> option notation in the descriptions of line constructions means a numerical parameter is to be input. In this case, instead of calling the parameters dialog, one can use the property window or simply click within the drawing.

<X>, <P>

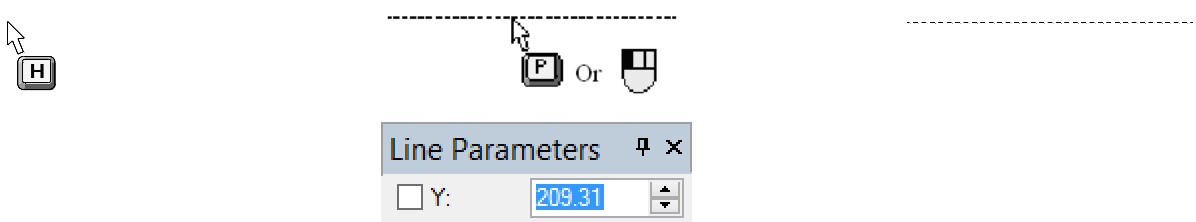
These options are used for creating a set of construction entities in one action, namely, a horizontal line, a vertical line and a node at their intersection.

First type <X>, then <P>.



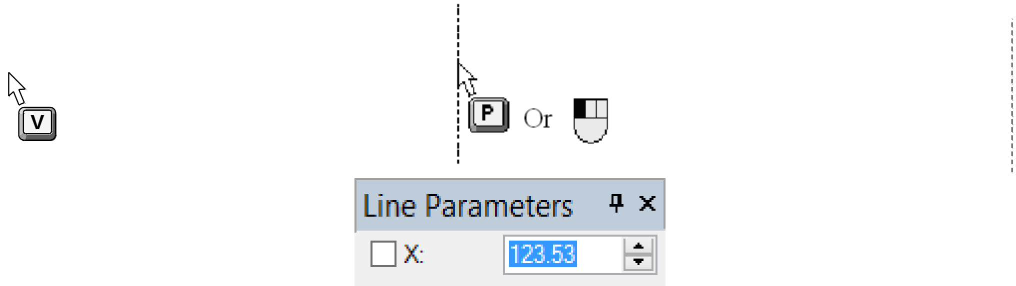
<H>, <P>

This option sequence creates horizontal lines at a specified distance from the X-axis. Type <H>, and then <P>.



<V>, <P>

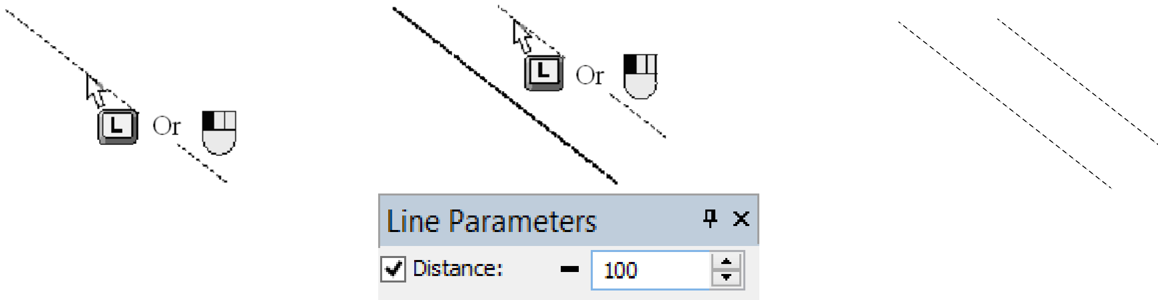
This key sequence creates vertical lines at a specified distance from the Y-axis. Type <V>, and then <P>.



<L>,<P>

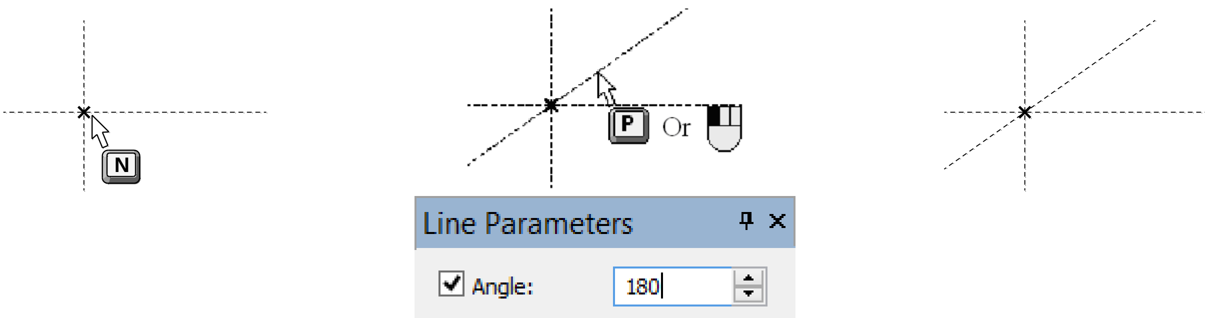
This key sequence creates a line, parallel to the selected line at a specified distance. Type <L>, and then <P> for defining the distance from the selected line.

> As a rule, this line type is used in drawings most often. This is because the lines on a drawing usually make parallel pairs, with the distance between them being a design parameter.



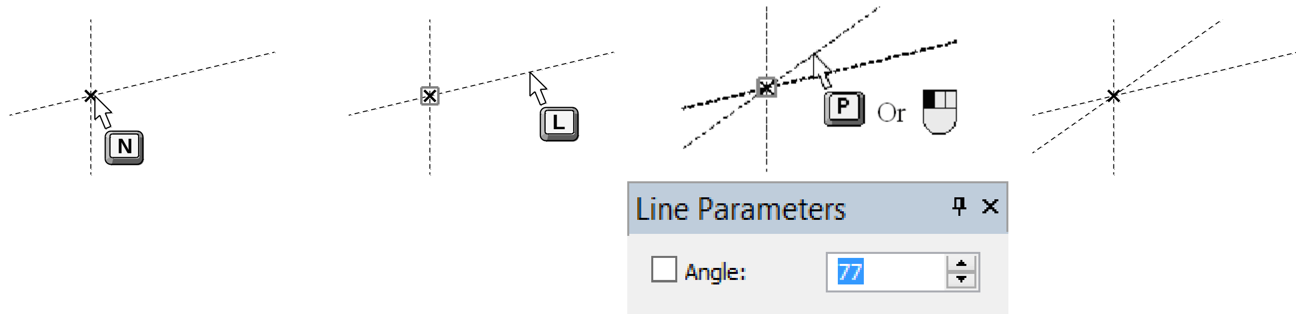
<N>, <P>

This key sequence creates a line at a specified angle to horizontal. Type <N>, and then <P>. The angle is entered in degrees.



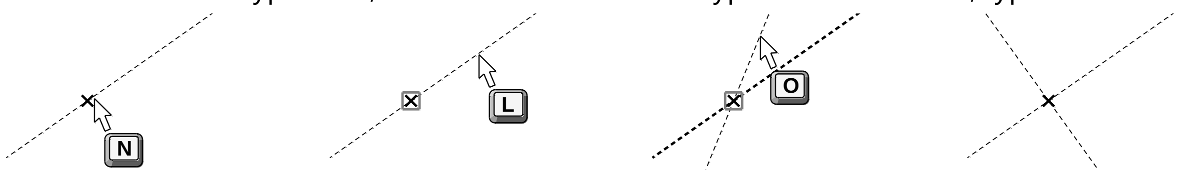
<N>, <L>, <P>

These options define a line passing through a node at a specified angle to a selected line. Move the cursor to the node and type <N>, and then to the line and type <L>. Thereafter, specify the angle between the lines (in degrees).



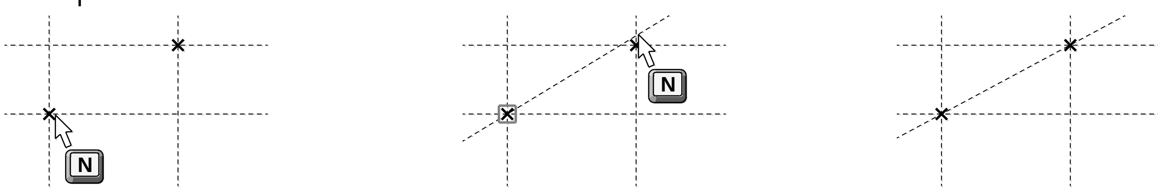
<N>, <L>, <O>

These options define a line passing through a node and orthogonal angle to a selected line. Move the cursor to the node and type <N>, and then to the line and type <L>. Thereafter, type <O>.



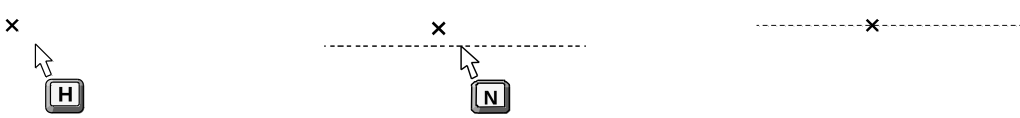
<N>, <N>

This combination defines a line passing through a pair of nodes. Move the cursor to the first node and type <N>. Repeat the same for the second node.



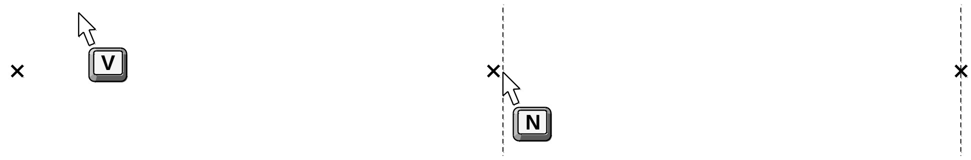
<N>, <H> or <H>, <N>

These option sequences create a horizontal line passing through a node. Move the cursor to the node and type <N>. Next, type <H>. These actions can be reversed.



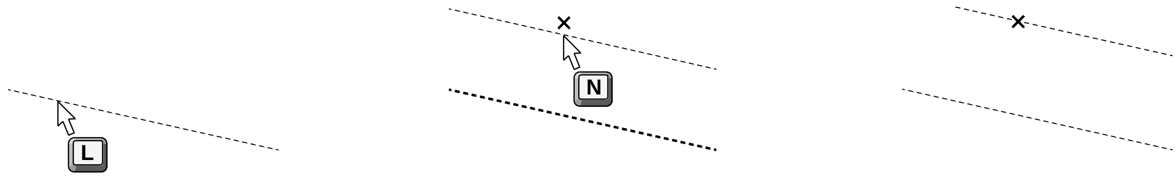
<N>, <V> or <V>, <N>

These option sequences create a vertical line passing through a node. Move the cursor to the node and type <N>. Next, type <V>. The actions sequence can be reversed.



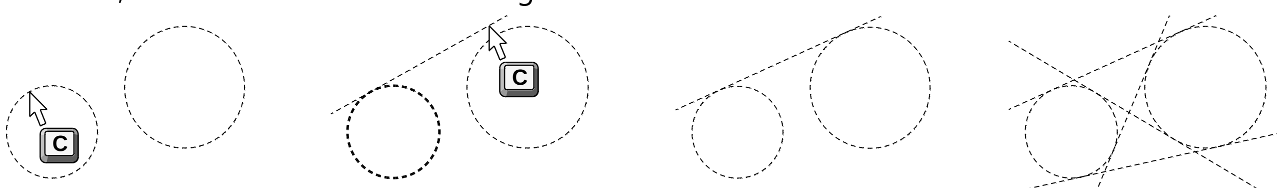
<L>, <N>

This option sequence creates a line parallel to a selected line and passing through a node. Move the cursor to the line and type <L>. Next, move over the node and type <N>.



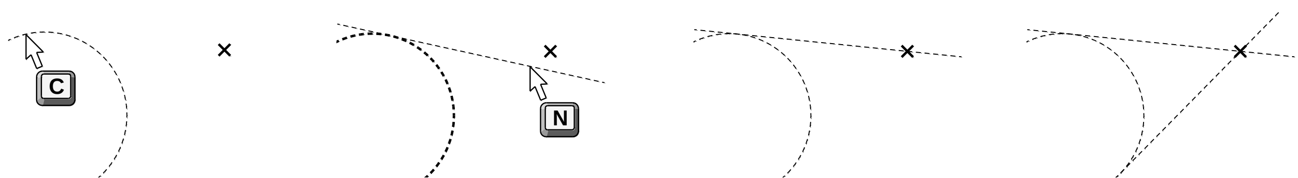
<C>, <C>

This combination defines a line tangent to two circles. Move the cursor to the first one and type <C>, then move to the second and type <C> again. Generally speaking, four distinct lines can be created in this situation, all of which would be two-tangent to the circles.



<C>, <N> or <N>, <C>

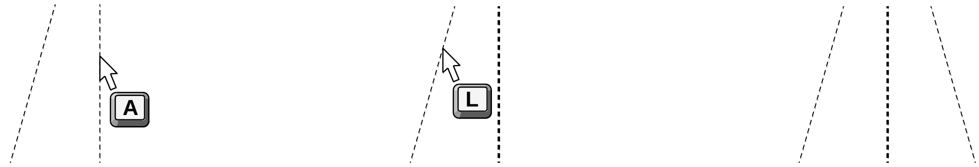
Using these combinations creates a line tangent to a circle and passing through a node. Move the cursor to the circle and type <C>, then move over the node and type <N>. Two distinct possibilities exist in this situation. One can do selections in the reversed order as well.



<A>, <L>

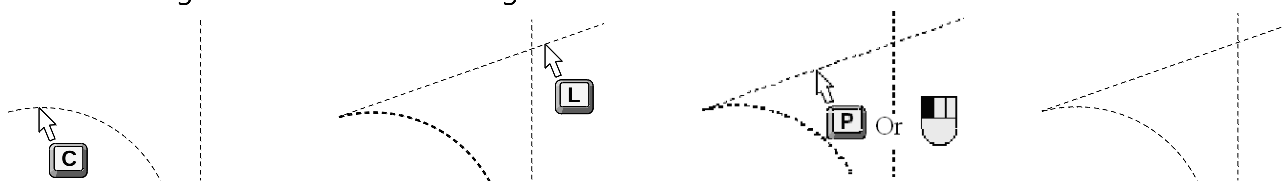
Use of such options creates a line (or lines) symmetrical to another one with respect to the selected axis. The symmetry axis is selected by typing the first key, <A>. Then, one can create a single line or multiple lines symmetrical with respect to the axis. A line to be mirrored is selected by typing <L>. Note that after typing <L> key once, the symmetry axis first stays selected, ready for mirroring more lines.

This mode stays active until canceled via <Esc> or .



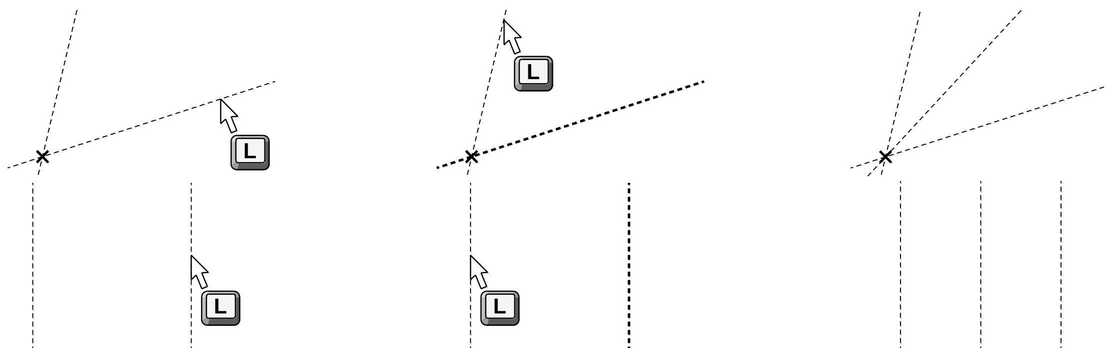
<C>, <L>, <P>

This combination creates a line tangent to a circle and passing at a specified angle to the selected line. Move the cursor to a circle and type <C>, then move over the line and type <L>. You will then see the angle value displayed in the coordinate field of the status bar. This is the angle between the selected and the rubberbanded lines. Now you have a choice of setting the parameter according to the cursor position or edit the parameter by entering a value, variable or expression. In the first case, simply click , and in the second – use the property window or the parameters dialog box for editing the angle parameter. Note that the angle value is entered in degrees.



<U>, <L>, <L>

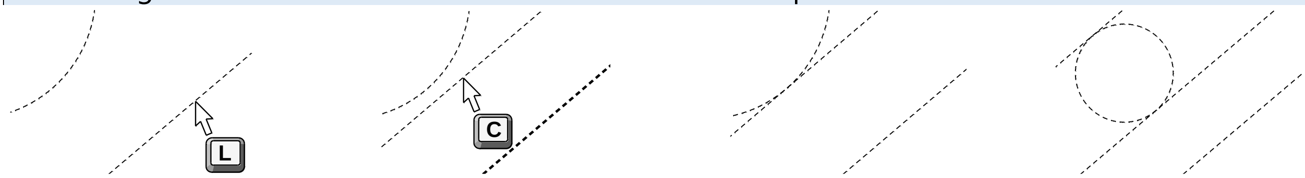
Use these options to create a line that is the symmetry axis for the two selected lines. In the case of intersecting lines, turn on the option <U>, move the cursor over one of them and type <L>. Then move over the other line and type <L> once more. A newly created line will bisect the angle between the two lines, acting as the symmetry axis. The same command works on parallel lines as well. After activating the option <U> move the cursor over one line and type <L>, then move over the other and again type <L>. A third parallel line will be created bisecting the distance between the two selected.



<L>, <C>

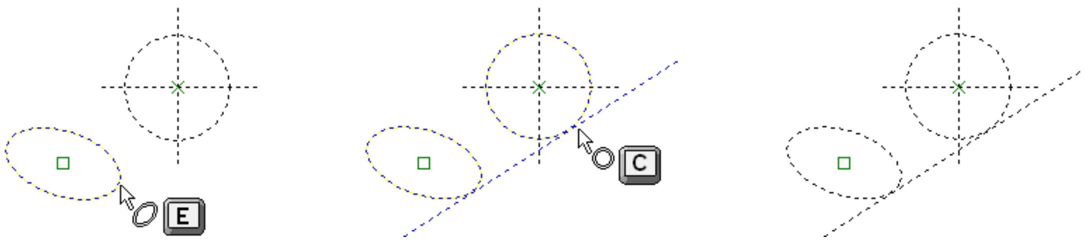
These options help define a lane parallel to a selected line and tangent to a circle. Move the cursor over a line and type <L>. Then move over a circle and type <C>. A new line will be created, parallel to the selected line and tangent to the circle.

Note: In this situation, as in some other cases, there are two possibilities for the line placement with respect to the circle. The system distinguishes them and at the creation instance settles with the configuration in which the line is closer to the selection point on the circle.



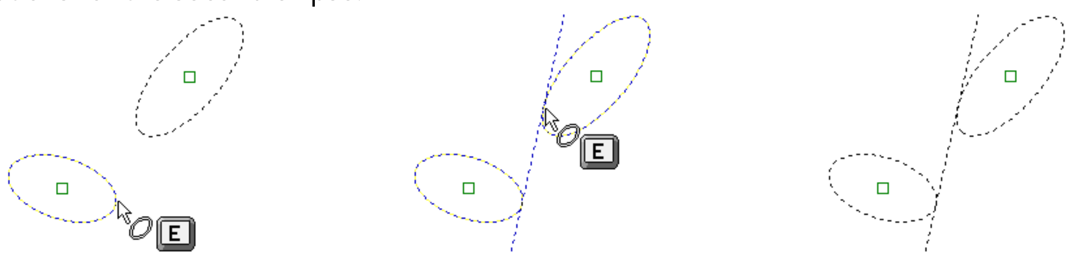
<E>, <C>

The options allow creating the line tangent to an ellipse and a circle at the same time. For constructing the line, bring the cursor to the given ellipse and press <E> (if the object snap mode is on, then it is sufficient just to press pointing at the ellipse). After that, dynamic image of the created line will end up being tangent to the ellipse for any cursor translations. Then point at the circle and press <C>. Tangency points with an ellipse and a circle (in case of non-uniqueness of the solution) will be selected by the system from the condition of maximum closeness to the cursor location upon selecting the ellipse and the circle.



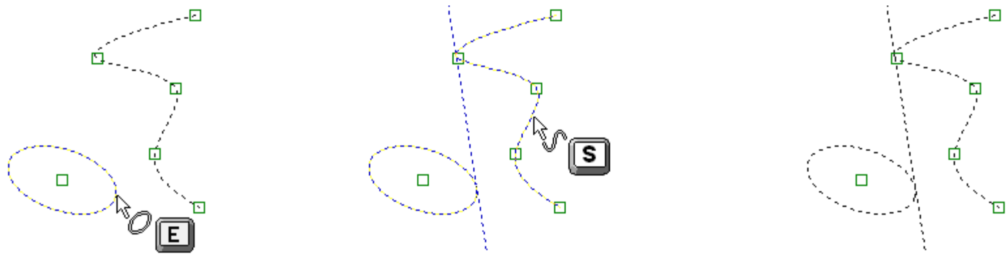
<E>,<E>

With the help of these options the line tangent to two ellipses can be constructed. For constructing the line, point at two ellipses successively: bring the cursor to the first ellipse and press <E>, then repeat the same operations for the second ellipse.



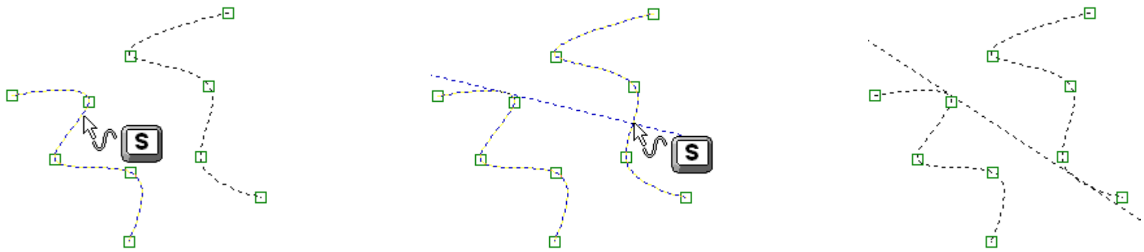
<E>,<S>

This option sequence allows creating a line tangent simultaneously to an ellipse and a spline. Bring the cursor to the required ellipse and press <E>. Dynamic image of the cursor will become tangent to the selected ellipse. After that, move the cursor to the spline, which a created line should be tangent to, and press <S>.



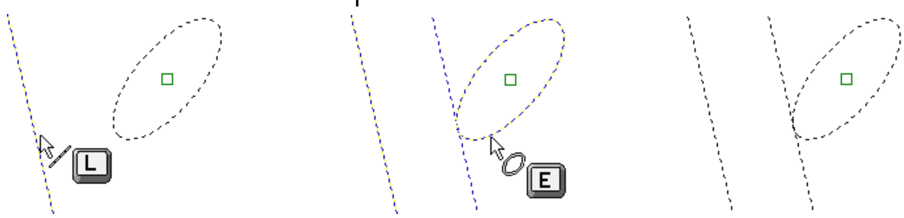
<S>,<S>

By using the option <S> twice, the line tangent to two splines will be created. For constructing the line, bring the cursor to the first spline and press <S>. Then repeat the same for the second spline. Similar to two previous cases, if different solutions for touching the splines and a created line can be found, tangency points will be selected by the system from the condition of maximum closeness to the cursor location upon selecting the splines.



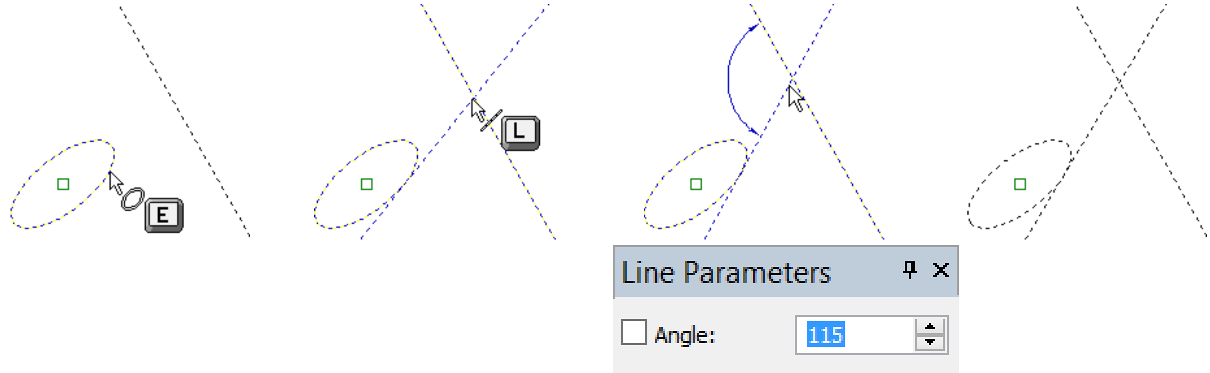
<L>,<E>

This option sequence allows creating the line parallel to another line and tangent to an ellipse. Bring the cursor to the line, which has to be parallel to the new line, and press <L>. Then move the cursor to the ellipse which a created line has to touch and press <E>.



<E>,<L>

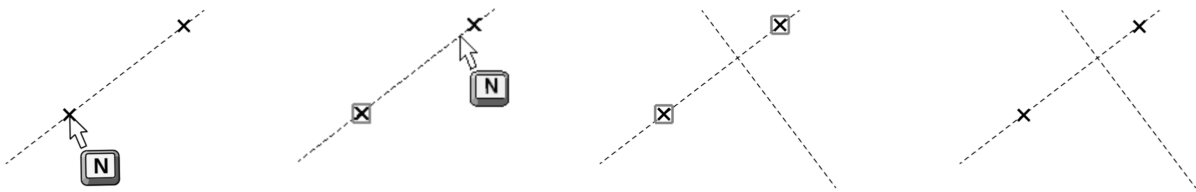
For constructing a line tangent to an ellipse and angular to another line, bring the cursor to the ellipse which the created line has to touch and press <E>. Then point with the cursor at the line at an angle to which the created line has to be drawn, and press <L>. After that, it is necessary to specify the angle between two lines (in degrees) directly in the drawing window or in the properties window.



<T>, <N>, <N>, <P>

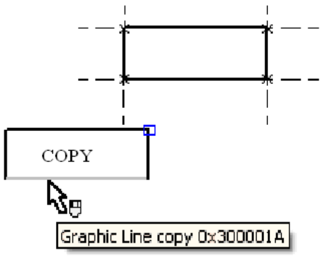
This sequence defines a line orthogonal to the imaginary segment spanning two selected nodes. The line is constructed so as to divide the segment in specified proportion. This proportion value is the parameter of the line being created, defined in dimensionless units. For example, with the parameter equal 0, the line will be passing through the first node, with the value 1 – through the second one, while the value 0.5 sets the line through the midpoint between the nodes. Should the nodes change location, the line will adjust, keeping the defined proportion. This kind of construction is used, for example, for drawing a spring with a fixed number of coils and variable length. The distance between the coils of such a spring will vary in a fixed proportion to the total length.

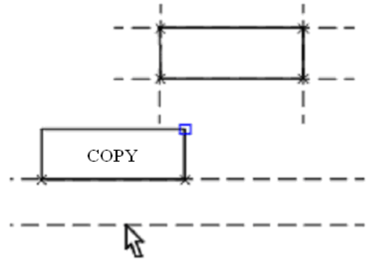
To create this kind of line, in the automenu turn on the option <T>, move the cursor to the first node and type <N>, then over the second node and type <N>. A line starts rubberbanding after the cursor orthogonal to the segment spanning the two selected nodes. The coordinate field will be displaying the current proportion value. Use the property window or the parameters dialog box to specify this value, or simply click , if satisfied.



Lines Created from 2D Projection, 2D Fragment, or Copy

Such lines can be created in object-snapping mode, when the respective flag is set on the **Snap** tab of the **SO: Set System Options** command. Move the cursor to a graphic line that is a part of a 2D projection, 2D fragment or a copy. The line will get pre-highlighted. Click . A construction line will be created on top of the selected graphic line. Besides, nodes will appear by the end points of the graphic line. Meanwhile, the system will assume the parallel line creation mode.





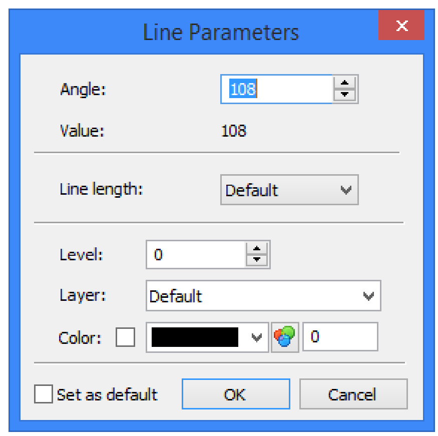
If the object-snapping mode is off, such lines can be created only based on 2D projection lines. To do so, select a desired projection on a drawing using the option. The selected projection will be highlighted, and the cursor will gain a glyph . To create a line, simply point the cursor at a line on the projection and click .

In this case, the following options will be accessible in the command automenu:

	<P>	Set Line Parameters
	<Esc>	Cancel selection

Line Parameters

When creating and editing lines, it is often required to define various line parameters. The geometrical parameters, such as coordinates, the distance or the angle to the related line, can be entered in transparent mode in the property window. However, in order to define the general system parameters of the line, one has to use the option to bring up the dialog box of all line parameters.



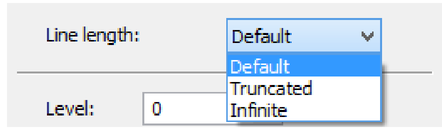
Distance. This is the distance between the newly constructed line and the line selected as the reference for the construction.

Level. Places the line being created on the particular visibility level. Levels help hiding certain elements from display. The level parameter can be assigned a variable.

Layer. This parameter allows placing the line being created on a certain layer.

Line length. Defines the way of construction line representation in display. The detailed description of this parameter follows below. The available values of this parameter, provided in the list, are:

Default (From Document);



Truncated;

Infinite.

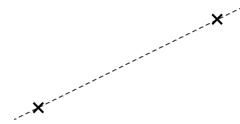
Color. This parameter defines the color used for displaying the line.

Set as default. Setting this flag means, the current parameter settings in this dialog box will be used from now on in construction line creation, with the exception of the “Distance” parameter.

Truncated Lines

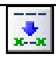
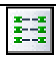
Normally, construction line entities appear on a drawing as infinite lines. However, as a drawing grows crowded, managing it becomes difficult. A means is provided for setting shortened representation of construction lines that allows working with lines as segments of limited length.

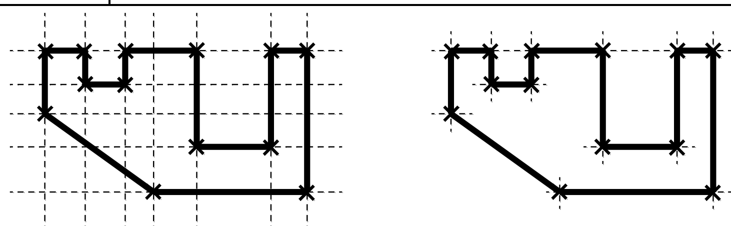
A trimmed, or truncated, construction line is bound by its two end nodes. The extents of trimmed construction line overhangs can be defined in the command **ST: Set Document Parameters** the parameter **View > Extents.**



If a construction line does not have nodes, then it will always appear as an infinite line. If a construction line has only one node then the “Extents” parameter should better be set greater than zero, as otherwise the line will disappear from display.

The line display gets up to date after executing the following options of the **EC: Edit Construction** command:

	<T>	Update selected Line(s) extents
	<Q>	Update all Line extents



The function of the “Line length” parameter significantly depends on the settings under the **ST: Set Document Parameters** command parameter **View** > **Length**.

Four ways of line display are supported by the **ST: Set Document Parameters** command:

Default truncated. If a particular line has the parameter “Line length” set to “Default” value, then this line is displayed as a segment.

Default infinite. If a particular line has the parameter “Line length” set to “Default” value, then this line is displayed as an infinite line.

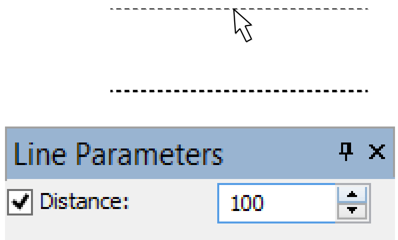
All truncated. Any and all lines will be displayed as segments, regardless of the “Line length” parameter settings.

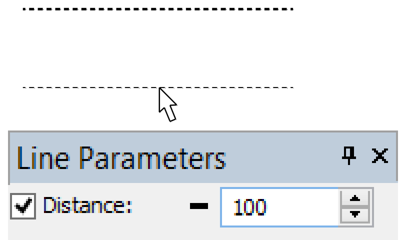
All infinite. Any and all lines will be displayed as infinite lines.

Another special parameter under the **ST: Set Document Parameters** command is **View** > **Search**. It defines the line selection mode. The lines are selected either within the displayed limits or as infinite lines, regardless of other parameter settings.

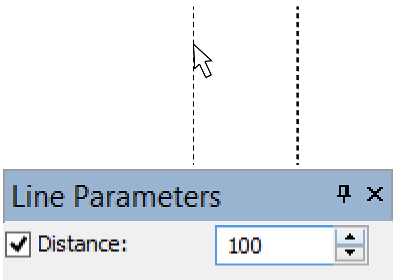
Using Numerical Parameters

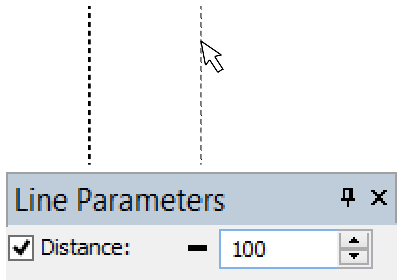
Entering the "Distance" parameter is a most common case of working with construction line parameters. The positive values of this parameter correspond to locations above the reference horizontal line, while the negative are below, respectively.





In the case of a vertical reference line, the positive values of this parameter are for the left hand-side locations, while the negative are for the right hand-side locations, accordingly.

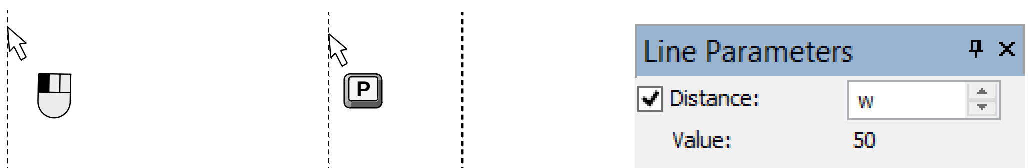




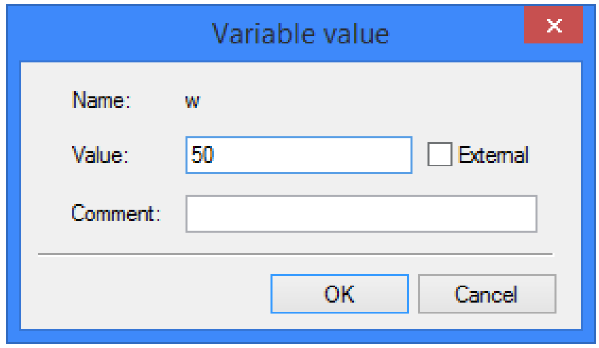
These rules result from the use of the coordinate system in T-FLEX CAD. That helps keeping the once set relations between construction entities under any modifications of the parameter values.

Note that the "-" sign may be preset by the system before the parameter input box in the property window. The system is monitoring the rubberbanded line position with respect to the reference parallel line. When the new line is rubberbanded in the area of negative offsets, the negative sign is automatically set, and the user needs to enter only the absolute distance.

As an alternative to entering a specific distance value, one can use variables. The name of a variable is defined as a US ASCII string of no more than 10 characters. The names are case-sensitive, therefore, for example, the two names *"Width"* and *"width"* are different. Let's assign the distance between the two lines a variable *"W"*.



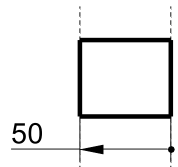
A dialog box will come up then, for defining the value of the variable. The variable being created can be flagged as “External”. The variable can have a positive or negative value, or assume the value of another variable or mathematical expression based on other variables. Let’s enter the value *“50”* for the variable.



If we flagged the variable as “External”, then in future it will be accessible for assigning values from outside the document.

Such situations include, for example, use of external applications, or assigning a value from an assembly document upon inserting the present drawing as a fragment. In our particular example, the variable is not required to be external.

Once the value has been assigned to the variable, a line is created, parallel to the reference one, at the distance of 50 units on the left-hand side of the reference.

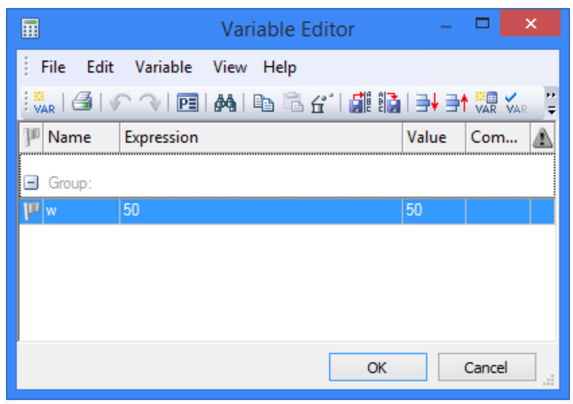


Now, one can verify the just defined relation between the lines. Enter the command **V: Edit Variables**:

Keyboard	Textual Menu	Icon
<V>	Parameters > Variables	

The Variable editor dialog box will come up on the screen. The only variable displayed in the dialog box will be the just created *“W”*, with the value *“50”*. The Variable editor has four fields (columns): “Name”, “Expression”, “Value” and “Comment”. Since we entered a numerical value for the variable, the “Value” and “Expression” readings are the same.

The quantity in the “Expression” field can be modified as necessary. After that, exit the editor by pressing **[OK]**. The drawing will instantly update per the new value of the *“W”* variable.



An expression can be used for the parameter “Distance” just in the same way. For example, suppose, the new line is constructed on the right-hand side of the vertical line. Then, to work around the “-” sign in the value of the variable, the variable can be assigned the expression *“-W”*. Generally speaking, one can use formulas with several variables in the expressions.

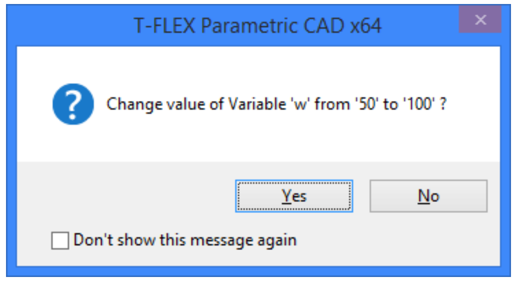
> For viewing and editing the values of the variables, one can use an additional window “Variables” which allows working with variables in transparent mode.

Editing Lines

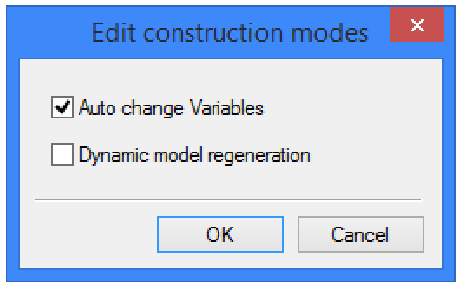
The **EC: Edit Construction** command is provided for editing construction lines. It is one of most often used commands. This is the command that supports creation of new drawing configurations by providing a dialog box for varying necessary construction parameters. The command allows editing all kinds of construction entities.

The command is called as:

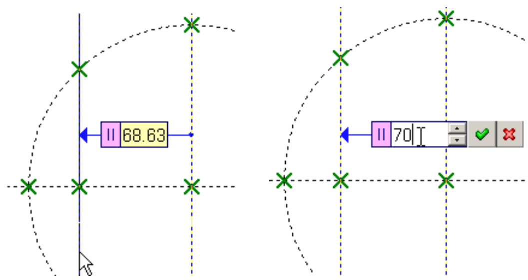
Keyboard	Textual Menu	Icon
<EC>	Edit > Construction > 2D Construction	



To modify location of some construction entity, simply select it using , move the cursor over the desired location, and click again. To specify the exact value of the placement parameter, use the property window or the parameters dialog box via the  option.

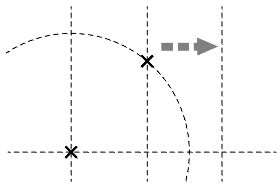


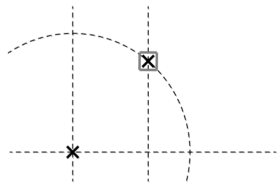
If the entity was driven by a variable, the system will output a warning. To avoid the system warnings, call the option  before selecting any entities. A dialog box will come up, in which the item **Auto change Variables** needs to be checked.

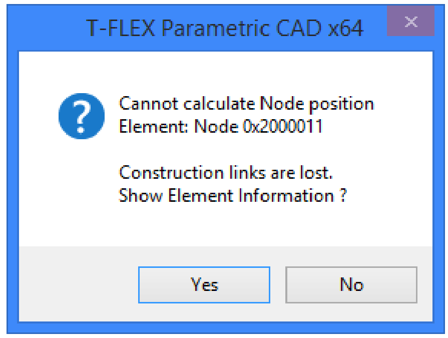


When modifying the values of construction parameters it is possible to use Relations that appear on element selection. These Relations are *temporary*. They will automatically disappear on editing finish. To modify the values of construction parameters with the help of Relations it is necessary to turn off “Dynamic recalculation” mode (option , see below).

It is possible that some construction entity can’t be recreated after modifying parameter values due to geometrical incompatibilities among the entities. In such a case, the system will output an error message and specify the particular failing relation.







The selected line is highlighted on the drawing. Besides, other construction entities are highlighted that were used as references for the line creation. The following options are available in the command **EC: Edit Construction**:

	<>	Dynamic model regeneration mode
	<P>	Set command options
	<O>	Create Name for selected Element
	<M>	Modify Construction Line relations
	<T>	Update selected Line(s) extents
	<Q>	Update all Line extents
	<K>	Break link with variable
	<I>	Select Other Element
	<R>	Select element from list
	<*>	Select All Elements
	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

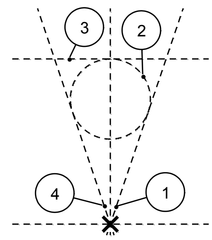
<Shift><Enter>		Add Construction Element to Selected for Editing

<Ctrl><Enter>		Exclude Construction Element from Selected List

The option (<K>) allows switching all parameters of the selected construction line from dependency on variables to the constant values.

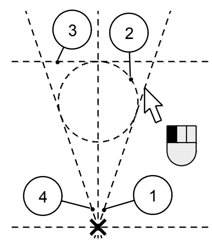
The option <O> allows specifying names for construction lines in order to define advanced parametric dependencies. Such a name will help exactly identify a construction line and, in particular, directly access certain proprietary data of the line in the variable editor via the command **V: Edit Variables**, using the function "get". The name is not required for common situations of parametric design.

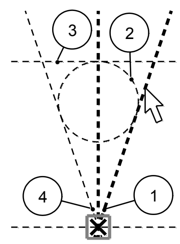
Modifying Relations between Construction Lines



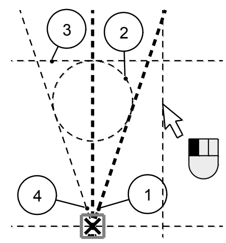
If for some reason you would like to modify the existing relations between the construction lines, this can be easily done using the  option. Let's review an example of using this option.

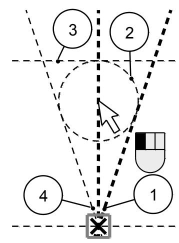
The line 1 is created at a given angle to a vertical line. Besides, it is passing through a node at the intersection of the vertical and a horizontal line. The circle 2 was constructed tangent to the lines 1 and 4, while the line 3 tangent to the circle 2.



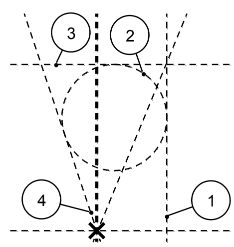


Suppose, you would like to make the line 1 parallel to the vertical line. Since other construction entities are created using the former line as a reference, the line may not be simply deleted and then differently created anew. The deletion of this line would require also removing the line 3, followed by removal of the circle 2. This is exactly the case when the option <M> is to be used. Enter the command **EC: Edit Construction** and select the line 1 for editing. This line will get highlighted along with another line and the node used as references for this line creation.





After selecting the line, type <M> for modifying the definition of the line 1 construction. Note that the system brought you into the construction line creation command, **L: Construct Line**. Now you can create this line as if anew. The difference from constructing a line without using the <M> option is that both the line being modified and the reference line are both highlighted on the screen. Select a line - the reference for the line 1 to be parallel to. Then rubberband the new line to the desired distance from the reference and fix that location by clicking .



Line 1 will appear in the new position, while all the rest of entities using the line as a reference will keep their relations with the line, as, for instance, the circle 2 will remain tangent to the line 1.

The only restriction on modifying relations between construction lines is a ban on recursive definition, that is, the line may not reference itself. Should this occur, a message will be displayed about recursion, and the modification will be cancelled.

With this exception, any relations between lines and circles may be modified at any time. This functionality is especially useful on importing drawings from other systems, such as, for example, *.DXF or *.DWG files of the AutoCAD system.

Deleting Construction Lines

To delete a construction line, simply select the line using and call the <Del> option. If the line is not referenced by any other drawing elements, it will be deleted. Should there be other elements defined based on the selected line, a warning will appear about deletion of all elements related with the line.

Circles

Circles in T-FLEX CAD are constructed similarly to lines, that is, by defining their geometrical relations with other construction entities. Examples of such relations include placement of the circle center at a node, tangency to a line, tangency to a circle, passing through a node, concentricity with another circle, symmetry to another circle.

The T-FLEX CAD circles can be divided into two main groups:

- circles, whose radius can be assigned a number (for example, a circle with the center placed at a node, or a circle tangent to two non-parallel lines);
- circles, whose placement and radius are defined by construction (for example, a circle passing through three given nodes).

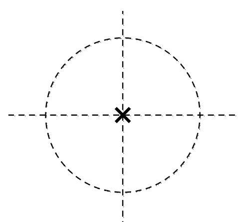
If a circle has a numerical parameter (the radius), then the parameter can be defined by a constant, a variable or an expression. To assign the numerical parameter, one can use the property window. The circles are created in the command **C: Construct Circle**. The relations defined at the time of circle construction can be modified in the command **EC: Edit Construction** in a similar to line way of handling.

Circle Construction Examples

Before discussing all capabilities of the command **C: Construct Circle**, let's consider examples of constructing most common types of circles. Additionally, the command **L: Construct Line** will be used in constructions. This command was described in the previous chapter.

Enter the command **L: Construct Line.** Select the option <X>, represented in the automenu by the icon . Move the cursor over the center of the graphic window and click. This creates two lines, a vertical and a horizontal one, and a node at the intersection. After that, enter the command **C: Construct Circle**.

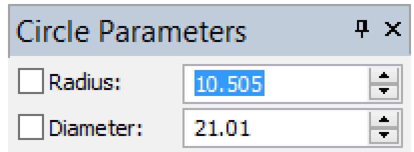
Icon	Ribbon
	Draw → Construct → Circle
Keyboard	Textual Menu
<C>	Construct > Circle



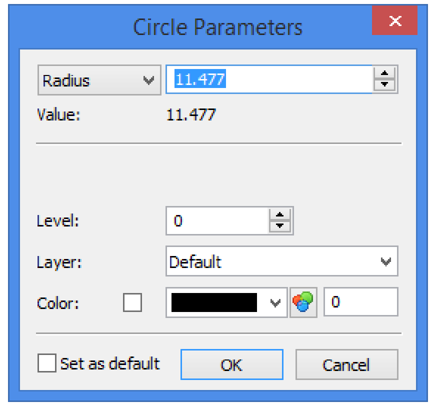
Move the cursor to the just created node and click. Thus you define the circle center. Internally, the system stores the relation between the circle center and the node. The node will be highlighted, and a circle will start rubberbanding after the cursor. Meanwhile, the radius of this circle will be dynamically updating on the status bar (*"R=..."*).

Clicking completes construction of the circle with the current radius.

For a simple way of defining an exact value of the circle radius while rubberbanding, use the property window in transparent mode. Besides, one can call a dialog box with the complete set of circle parameters via the option of the automenu.



The parameters dialog box allows defining, besides the radius, the circle's general system parameters, such as visibility level, layer and color. For instance, the default drawing settings of visibility are such that construction entities with the level defined in the range 0-127 are visible (see the command **SH: Set Levels**). This means, changing the level value of the circle being created to *"-1"*, will hide the circle from display, as the level value is not in the range 0-127.



For already created circles, in command waiting mode point the cursor at the circle and double-click . The circle parameters dialog box will appear on the screen. Change the level value to "-1" and press **[OK]** to confirm the input. Note that the circle disappeared from display. However, this does not mean that the circle is completely removed from the model Move the cursor to the location where the circle used to be, and once again click . The circle will be selected, in spite of being invisible. Call the parameters dialog box again and set a different value for the visibility level, for example, *"0"* (zero). Now, as you press **[OK]**, the circle becomes visible again.

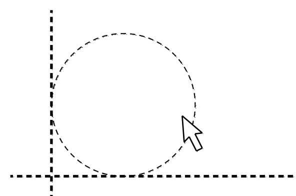
> Note: to actually make the hidden drawing elements unselectable, set the appropriate value for the parameter **View** > **Element selection** in the **Customize** > **Set Document Parameters** command.

Another way of making a circle invisible is by using layers. Place the circle on a certain layer, and then set the layer invisible in the command **QL: Configure Layers**. To place the circle on a layer, enter the layer name either in the **Layer** entry of the circle parameters dialog box, or in the respective field of the system toolbar.

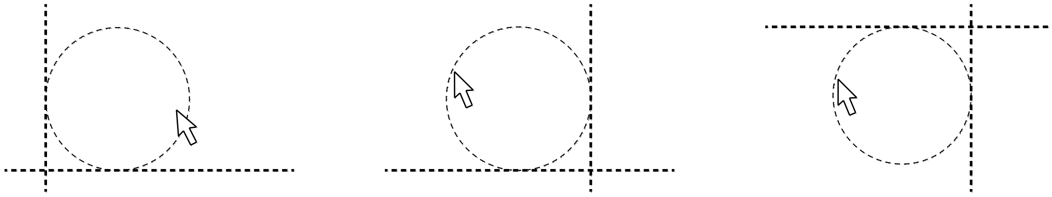
Another type of a circle often used in base geometry construction is a circle tangent to two lines. In order to try creating this type of a circle, begin again with the **C: Construct Circle** command.

First, move the cursor over the vertical line that already exists on the drawing, and type <L>. The line will get highlighted, and the cursor will start rubberbanding a circle locked tangent to the selected line.

Now move the cursor over the horizontal line and once again type <L>. The second line will be selected, and the rubberbanded circle will now be tangent to two lines.

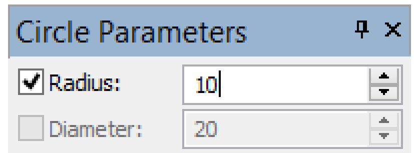


The dynamically changing circle radius will be displayed in the coordinate field of the status bar. Note that you can move the cursor to any of the four quadrants defined by the lines, and the rubberbanded circle will correctly follow the cursor.

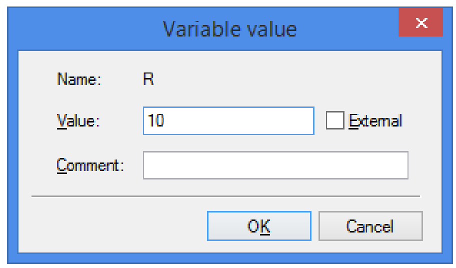


Now, you can set the circle radius by simply clicking, or by inputting in the property window or the circle parameters dialog box (<P>).

Once the circle is created tangent to two lines, this relation will always be maintained. To witness this, do, for example, the following. Exit the command **C: Construct Circle**. In command-waiting mode, move the cursor over the created circle and click. The system will enter the editing command (**EC: Edit Construction**). The circle will be highlighted. Now the user can easily modify its radius by moving the cursor. Meanwhile, the tangency to the lines will be maintained. Try doing this several times, moving the circle over other quadrants.



At any moment while rubberbanding the circle in the command **EC: Edit Construction**, one can modify the parameter value (the radius). This can be done in transparent mode in the property window, or in the parameters dialog (the option) Instead of a numerical value, a variable name or an expression can be used. For example, instead of the radius value, one can enter a variable name *"R"*.



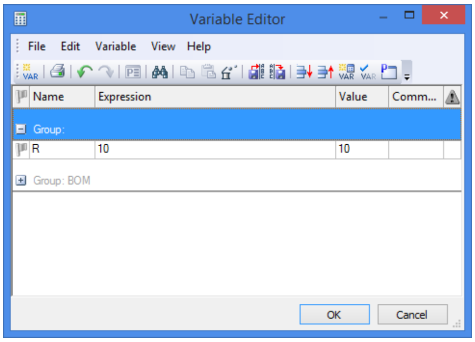
After pressing **[OK]** the system will request the value for the newly created variable *"R"*. One can accept the system default value or modify it as desired.

In future, this will allow using the variable for data exchange with external applications or for configuring the drawing when inserting into an assembly. Upon confirming the value of the new variable, the drawing will be regenerated per the entered radius value.

From now on, the circle radius is driven by the variable *"R"*. The value of this variable can be easily modified in the command **V: Edit Variables**. Call the command **V: Edit Variables**.

Icon	Ribbon
	Title Block → Additional → Variables
Keyboard	Textual Menu
<V>	Parameters > Variables

The variable editor window will come on the screen (see the chapter "Variables"), containing four columns: **Name, Expression, Value, Comment**. The **Name** entry contains the only existing variable, the *"R"*. The **Expression** entry displays the number defined at variable creation. The same number is displayed in the **Value** field. Let's change the value of this variable. Enter a new value, say, *"50"*. Upon confirming the input by <Enter>, the new value will be displayed in the third column, titled **Value**.



The **Comment** field may be used for entering information text about the current variable. This information is called “the variable comment”. The comment is not used in defining geometrical relations or regeneration per se, however, it may be quite helpful to the user at the time of modifying the drawing.
Modify the *“R”* variable several times, exiting the variable editor each time by pressing **[OK]**. The drawing will regenerate each time according to the new entered value of the variable.

Constructing Circles

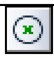
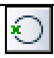
The command **C: Construct Circle** provides various options from the following list, depending on the current state:

	<Enter>	Select a node as the circle center
	<P>	Set circle parameters
	<L>	Select tangent line
	<N>	Select a node for the circle to pass through
	<C>	Select tangent circle
	<E>	Select tangent ellipse
	<S>	Select tangent spline
	<A>	Select symmetry axis (line) to mirror the circle.
	<O>	Select concentric circle
	<W>	Select 2D projection
	<Z>	Change construction tangency
	<Space>	Construct a node at the nearest intersection of two construction entities
	<F4>	Execute Edit Construction command
	<Esc>	Cancel selection
	<Esc>	Exit command

T-FLEX CAD supports the most common circle construction modes, which are:

- constructing a circle with the center at a selected node
- constructing a circle passing through a selected node

The following options initiate these modes:

	<T>	Select a node as the center of the circle
	<T>	Select a node for the circle to pass through

Upon calling the command, one of the modes is activated by default, as indicated by the pushed icon in the automenu.

Just as with the construction lines, various combinations of a small set of options with a specific circle construction mode yield a variety of geometrical dependency sets in the constructed circles:

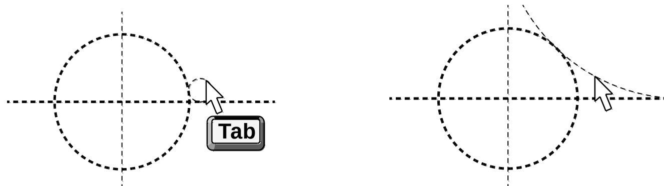
<Enter>, <P>	Circle with center at a node, with a specified radius <*>
<Enter>, <C>	Circle with center at a node, tangent to a circle
<Enter>, <L>	Circle with center at a node, tangent to a line
<Enter>, <N>	Circle with center in the nearest node, passing through a node
<L>,<L>,<P>	Circle tangent to two lines, with a specified radius <*>
<N>,<L>,<P>	Circle tangent to a line, passing through a node, with a specified radius <*> <**>
<N>,<C>,<P>	Circle tangent to a circle, passing through a node, with a specified radius <*> <**>
<N>,<N>,<P>	Circle passing through two nodes, with a specified radius <*> <**>
<C>,<L>,<P>	Circle tangent to a line and a circle, with a specified radius <*> <**>
<C>,<C>,<P>	Circle tangent to two circles, with a specified radius <*>
<N>,<N>,<N>	Circle passing through three nodes
<L>,<L>,<L>	Circle tangent to three lines
<N>,<L>,<L>	Circle tangent to two lines, passing through a node <**>
<C>,<L>,<L>	Circle tangent to two lines and a circle <**>
<C>,<C>,<N>	Circle tangent to two circles, passing through a node <**>
<C>,<L>,<N>	Circle tangent to a circle and a line, passing through a node <**>
<N>,<N>,<L>	Circle tangent to a line, passing through two nodes <**>
<N>,<N>,<C>	Circle tangent to a circle, passing through two nodes <**>
<L>,<S> <L>,<E>	Circle tangent to a line and a spline or ellipse <**>
<C>,<S> <C>,<E>	Circle tangent to a circle and a spline or ellipse <**>
<S>,<S>	Circle tangent to two splines
<E>,<E>	Circle tangent to two ellipses
<S>,<E>	Circle tangent to a spline and an ellipse <**>

<O>,<P>	Circle, concentric with another circle, with a specified radial offset
<A>,<C>	A circle mirrored about a symmetry axis

<*> - Use of <P> equivalent to <Enter> or .

<**> - The options <C>,<L>,<N>,<S>,<E> can be used in any order, besides the listed.

In both circle construction and editing, typing <Z> can be conveniently used for traversing configurations within the chosen type of circle construction. For example, two configurations are possible when constructing a circle tangent to a line and another circle, with the same cursor position. The <Z> key can be used for switching between these two configurations.



Similarly, during editing, a circle configuration can be flipped as well.

Let's review each way of constructing circles in details.

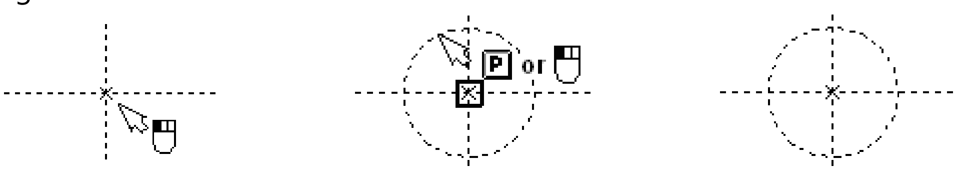
Various Ways of Constructing Circles

Begin with setting the option:

	<T>	Select a node as the circle center

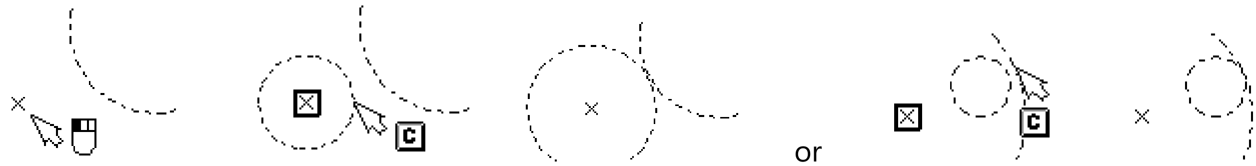
<Enter>, <P>

This combination creates a circle with the center in the selected node and the radius defined by either placing the cursor and clicking or entering an exact value in the property window or parameters dialog box (the option <P>). To create this type of a circle, point the cursor at the desired node and click . The node will be highlighted, and a circle with the center in this node will start rubberbanding. The circle can then be fixed manually, by clicking , or exactly, by specifying the radius value in the property window or parameters dialog box.



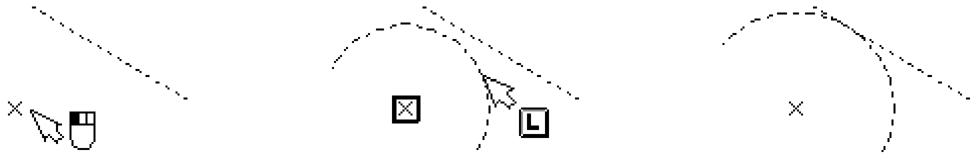
<Enter>, <C>

This combination creates a circle with the center at a specified node, and tangent to another circle. Move the cursor to an existing node or use the <Space> option for creating a node at the nearest intersection of construction entities. Click . This results in rubberbanding a circle with the center in the selected node. Move the cursor to a circle to become the tangency reference for the one being created, and type <C>. The required circle will be created. Two different configurations are possible, depending on where on the tangent circle the cursor was pointing at the time of entering the <C> option.



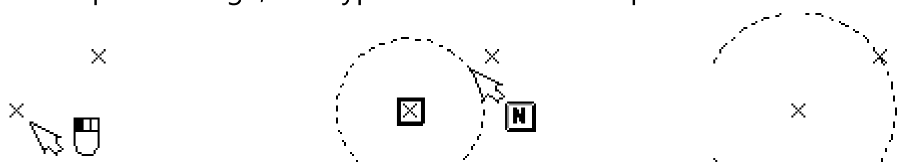
<Enter>, <L>

This combination creates a circle with the center at a node, tangent to a construction line. Move the cursor to an existing node or use the <Space> option for creating a node at the nearest intersection. Click . A circle will start rubberbanding with the center in the selected node. Move the cursor over the line to become the tangency reference for the circle, and type <L>. The required circle will be created.



<Enter>, <N>

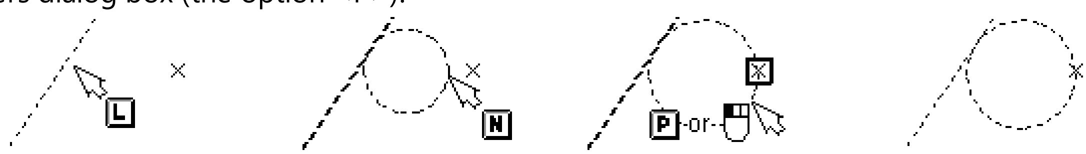
This combination creates a circle with the center at a node, passing through another node. Move the cursor to an existing node or use the <Space> option for creating a node at the nearest intersection. Click . This allows you to rubberband a circle with the center in the selected node. Move the cursor to the node for the circle to pass through, and type <N> or . The required circle will be created.



<L>, <N>, <P> for use in the mode of constructing a circle with the center at a node.

<N>, <L>, <P> for use in the mode of constructing a circle passing through a node.

Either of the two option combinations creates a circle of a specified radius, tangent to a line and passing through a node. Subsequently select a line and a node, using the options <L> and <N> respectively. Then specify the radius either by a click or by inputting the exact value in the property window or parameters dialog box (the option <P>).

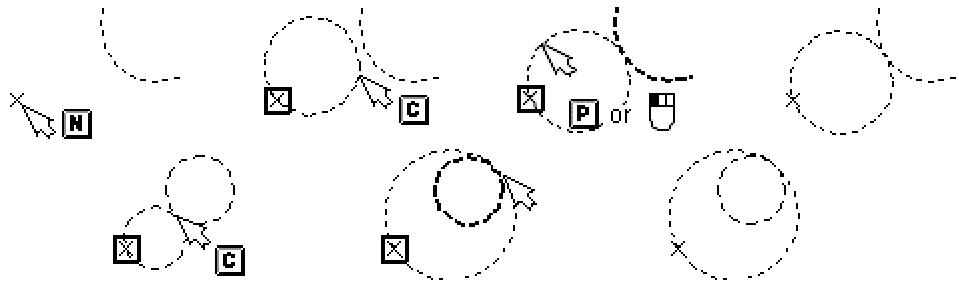


Please note that the node may belong to the line, which is often the case in drawings.

<C>, <N>, <P> for use in the mode of constructing a circle with the center at a node.

<N>, <C>, <P> for use in the mode of constructing a circle passing through a node.

Either of the two option combinations creates a circle of a specified radius, tangent to a circle and passing through a node. Subsequently select a circle and a node, using the options <C> and <N> respectively. Then specify the radius either by a click or by inputting the exact value in the property window or parameters dialog box. Please note that different configurations are possible, depending on where on the entities the cursor was pointing at the selection time.

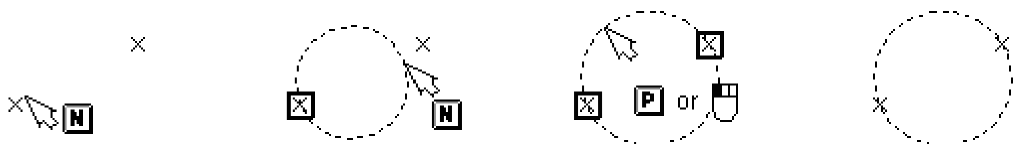


Now set the option:

	<T>	Select a node for the circle to pass through

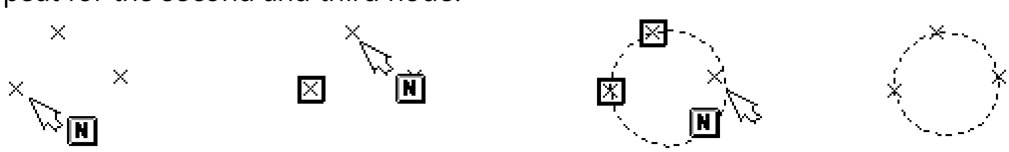
<N>, <N>, <P>

This combination creates a circle of a specified radius, passing through two nodes. Move the cursor over the first node and type <N>. Repeat for the second node. Then define the circle radius. Do this by specifying approximately by clicking or exactly in the property window or parameters dialog box (the option <P>).



<N>, <N>, <N>

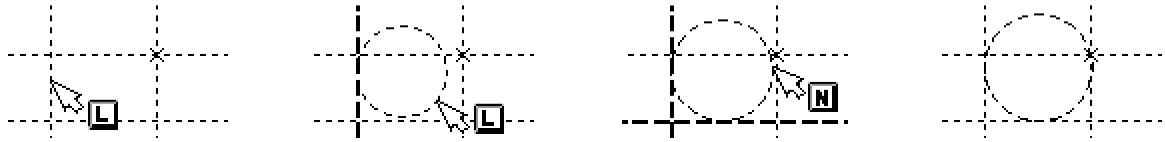
This combination creates a circle passing through three nodes. Move the cursor over the first node and type <N>. Repeat for the second and third node.



<L>, <L>, <N> for use in the mode of constructing a circle with the center at a node.

<N>, <L>, <L> for use in the mode of constructing a circle passing through a node.

This combination creates a circle tangent to two lines and passing through a node. Move the cursor over the first line and type <L>. Repeat for the second line. Then specify the node via <N>. Note that the order of typing the option keys may vary.



<L>, <N>, <N> for use in the mode of constructing a circle with the center at a node.

<N>, <L>, <N> for use in the mode of constructing a circle passing through a node.

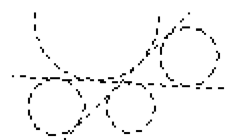
<N>, <N>, <L> for use in the mode of constructing a circle passing through a node.

This combination creates a circle tangent to a line and passing through two nodes. Move the cursor over the line and type <L>. Then use the option <N> twice for selecting the nodes. Note that the order of typing the option keys may vary.

Regardless of what construction mode the system is in, the circles can also be created in the following ways:

<L>, <L>, <P>

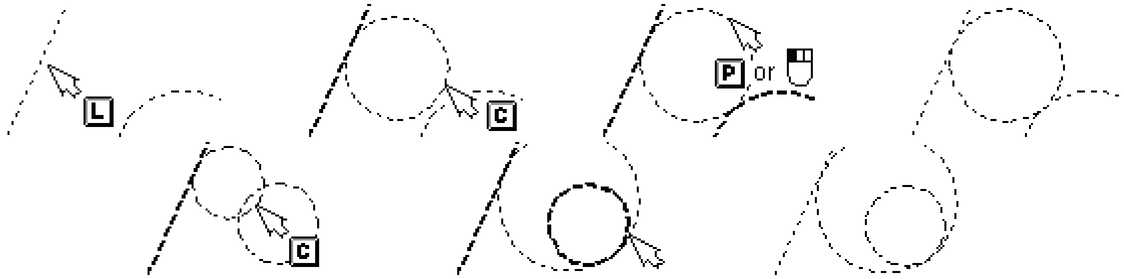
This combination creates a circle of a specified radius, tangent to two non-parallel lines. To create a circle, move the cursor over the first construction line and type <L>. Repeat for the second line. Then define the circle radius either by clicking or by inputting in the property window or parameters dialog box (the option <P>). Four various configurations are possible.



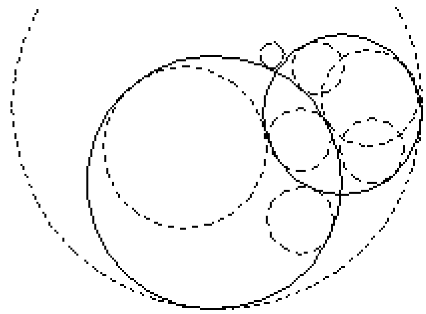
<L>, <C>, <P>

<C>, <L>, <P>

Either of the two option combinations creates a circle of a specified radius, tangent to a line and a circle. Subsequently select a line and a circle, using the options <L> and <C> respectively. Then specify the radius either by clicking or by inputting in the property window or parameters dialog box (the option <P>). Various tangency configurations are possible.

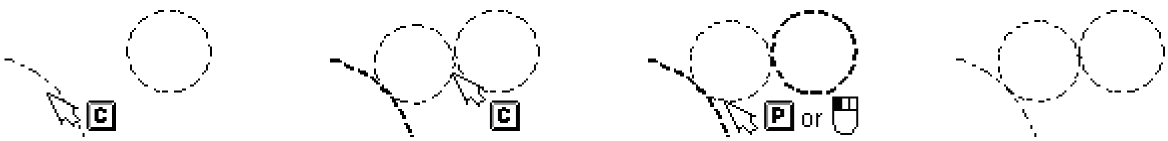


<C>, <C>, <P>



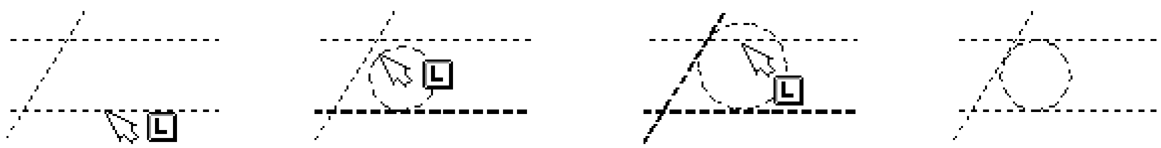
This combination creates a circle of a specified radius, tangent to two circles. Move the cursor over the first circle and type <C>. Repeat for the second circle. Then specify the radius – approximately, by clicking , or exactly, by inputting in the property window or parameters dialog box (the option <P>). This type of circle construction is most plentiful in resulting circle configurations.

Getting the desired one among various tangency types is done by appropriately positioning the cursor when selecting. In some cases, as, for instance, in the case of a surrounding circle, an appropriate selection gets easier when zoomed well out with the command "**ZW: Zoom Window**".



<L>, <L>, <L>

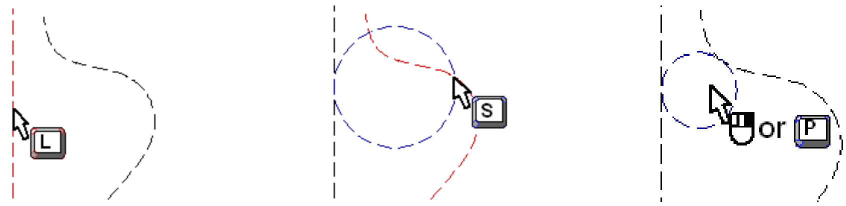
This combination creates a circle tangent to three lines. Move the cursor over the first line and type <L>. Repeat for the second and the third line. When selecting the lines, pay attention to the cursor position, which should be on the side of line intended for the circle location.



<L>, <S>, <P> or <S>, <L>, <P> for constructing a circle tangent to a line and a spline.

<L>, <E>, <P> or <E>, <L>, <P> for constructing a circle tangent to a line and an ellipse.

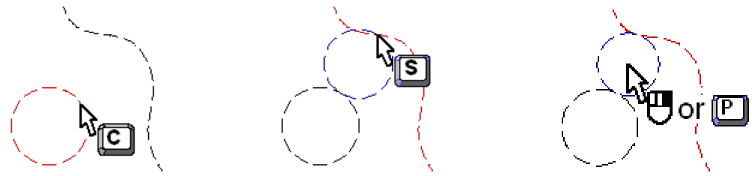
This combination creates a circle tangent to a line and a spline or an ellipse. Subsequently select a line and a spline (ellipse), using the options <L> and <S> (<E>) respectively. Then specify the radius either by a click or by inputting in the property window or parameters dialog box (the option <P>).



<C>,<S>,<P> or <S>,<C>,<P> for constructing a circle tangent to a circle and a spline.

<C>,<E>,<P> or <E>,<C>,<P> for constructing a circle tangent to a circle and an ellipse.

This combination creates a circle tangent to another circle and a spline or an ellipse. Subsequently select a circle and a spline (ellipse), using the options <C> and <S> (<E>) respectively. Then specify the radius either by a click or by inputting in the property window or parameters dialog box (the option <P>).

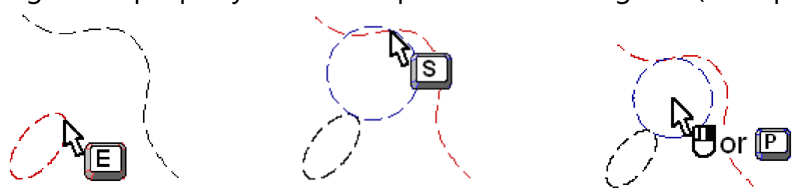


<S>,<E>,<P> or <E>,<S>,<P> for constructing a circle tangent to a spline and an ellipse.

<S>,<S>,<P> for constructing a circle tangent to two splines.

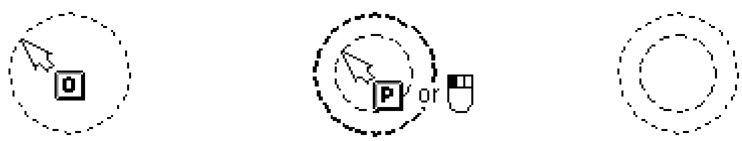
<E>,<E>,<P> for constructing a circle tangent to two ellipses.

This combination creates a circle, tangent to two splines or ellipses. Subsequently select the intended entities, spline(s) and/or ellipse(s), using the options <S> and/or <E>. Then specify the radius either by clicking or by inputting in the property window or parameters dialog box (the option <P>).



<O>, <P>

This combination creates a circle, concentric to a selected one, with the specified radius offset. Move the cursor over a circle and type <O>. Then define the radius offset by a click, or by an input in the property window or circle parameters dialog box (the option <P>).



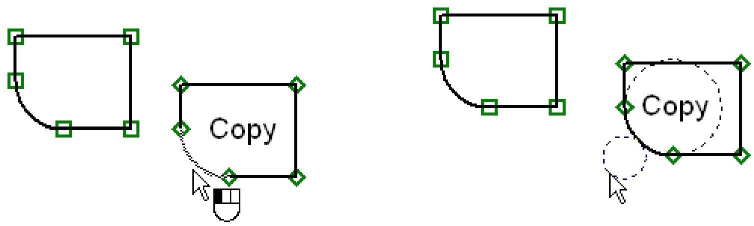
The offset is considered negative for the circles within, positive for the ones outside the defining circle.

<A>, <C>

This combination creates a circle, mirrored from another circle about a straight line. To select the symmetry axis, move the cursor over a line and type <A>. Then select a circle using <C>.

Circles Constructed Based on 2D Projection, 2D Fragment or Copy

Such circles can be created in the object snapping mode, if the respective flag is set in the command **SO: Set System Options** on the **Snap** tab. Move the cursor over a graphic entity – a circle or an arc on a 2D projection, 2D fragment or a copy. The entity will be pre-highlighted. Clicking at that moment creates a circle based on the selected entity.



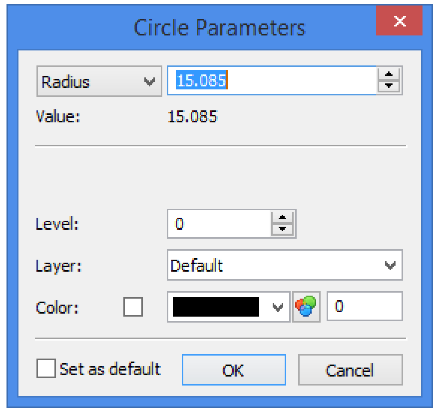
If the object-snapping mode is off, then circles can be created based on 2D projection entities only. To do so, use the option and select the desired projection on the drawing. The selected projection will be highlighted, and the cursor will gain the glyph . Now, to create a circle, simply point the cursor at a projection entity – arc or circle - and click .

The following options will then become available in the automenu:

	<P>	Set Circle parameters
	<Esc>	Cancel selection

Circle Parameters

Various parameters need to be defined when creating or editing circles. The geometrical parameters – the radius or the offset for concentric circles can be defined in the transparent mode via the property window. However, the general system parameters can only be accessed via the option that provides access to all circle parameters via a dialog box.



Radius. Defines the circle radius. Allows numerical value, variable or expression input.

Level. Places the circle being created on a certain visibility level, used for hiding certain elements from display when necessary.

Layer. Is used for linking the circle being created to a certain layer.

Color. This parameter defines the color of displaying the circle on the screen.

Set as default. When this flag is set, the parameters defined in this dialog box will be used in creation of new construction entities (except the **Radius** parameter).

Editing Circles

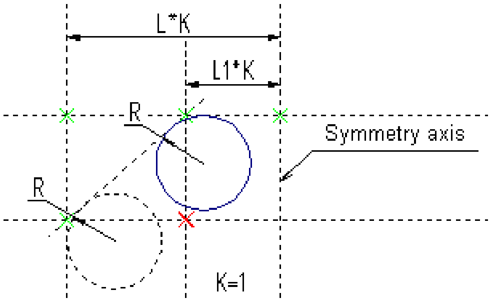
The same command **EC: Edit Construction** is used for editing circles as for any other construction entities. We have already reviewed the capabilities of this command on the example of editing construction lines in the previous chapter. Editing circles is done in a similar way.

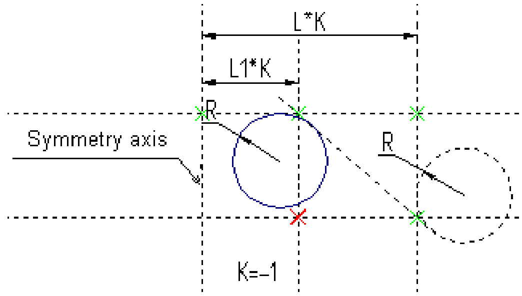
Besides, when editing a circle tangent to two lines, the automenu gains additional options for setting and clearing a "link" with a node:

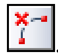
	<G>	Link Arc or Circle to Node
	<B>	Break Link with Node

The option is used for constraining a circle, tangent to two lines, by an additional node defining the tangency configuration. Upon calling the option, select the desired node by clicking. The circle will be reconstructed as the result of using this option to pass as close to the node as possible. This allows to uniquely define the circle location with respect to the reference lines.

The example below explains the use of this capability. The location of the lines on the drawing is defined with respect to the symmetry axis using a parameter *K*. Both circles are constructed tangent to the two lines. Besides, the solid-drawn circle is "linked" to the highlighted node. Meanwhile, no reference node is defined for the second circle. Originally, with the variable *K=1*, the drawing looks like shown on the diagram on the left-hand side. The right-hand side diagram shows the drawing modification per the new variable value *K=-1*. The circle linked to the node adjusted correctly. The second circle that did not reference any node, flipped with respect to the symmetry axis.





To release or re-assign a link with a node, use the option .

ELLIPSES

Ellipses in T-FLEX CAD are constructed similar to circles - by defining their geometrical relations with other construction entities. Examples of such relations are the ellipse center being snapped to a node, tangency to a line, tangency to a circle, passing through a node, symmetry with another ellipse. Ellipses, like other construction entities, are displayed in a thin dashed line.

Ellipses in T-FLEX CAD can be divided into two types:

- Ellipses whose size is defined by numerical parameters;
- Ellipses whose position and size are defined by geometrical relations.

Constructing Ellipses

To construct an ellipse, use the command **EL: Construct Ellipse.** The command can be called by one of the following means:

Icon	Ribbon
	Draw → Construct → Ellipse
Keyboard	Textual Menu
<EL>	Construct > Ellipse

T-FLEX supports most common ellipse creation modes, that are:

- Constructing an ellipse with the center at a node;
- Constructing an ellipse passing through a node.

These modes correspond to the following options:

	<T>	Select Centre Ellipse Node
	<T>	Select Node of Ellipse

Upon calling the command, one of the modes activates automatically, as indicated by the pushed icon in the automenu.

Ellipses Construction Techniques

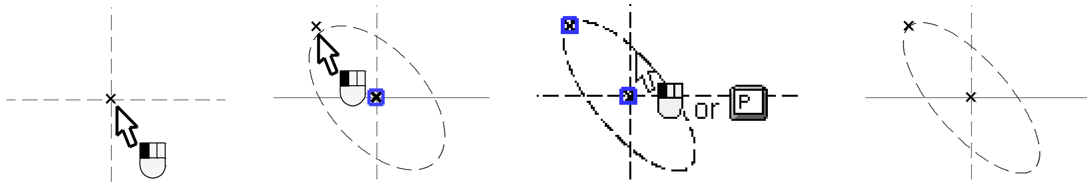
The following techniques can be used in the mode of constructing ellipses with the center at a node:

1. Center at node, through node, using parameter

This way of constructing an ellipse is realized in the following sequence of options:

<Enter><Enter><Enter> or <Enter><Enter><P>

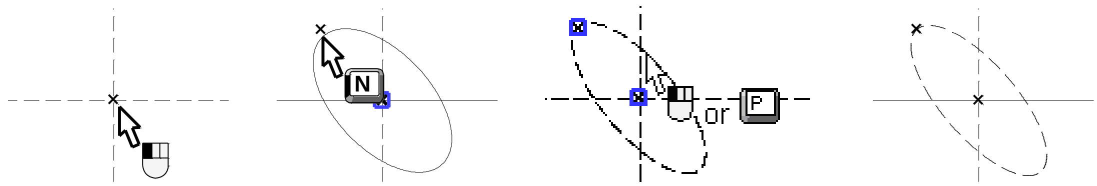
This combination creates an ellipse with the center at a node, with the first semi-axis ending at a node, and the second defined by a parameter. To create this type of ellipse, select the desired node. The node will highlight and an ellipse begin rubberbanding with the center at the selected node. After that, select another node for the ellipse's first semi-axis to pass through. Then move the cursor over desired position and click or use <P> option**.**



In the latter case, the parameters dialog box will appear on the screen allowing to enter an exact parameter value. If the mouse click was used for defining the ellipse then the numerical value for the ellipse parameter is derived from the cursor position. The property window can be used for defining the geometrical parameter of the ellipse (the semi-axis length) in transparent mode instead of using the option <P>**.**

The same way of constructing an ellipse can be realized by the following sequence of options:

<Enter><N><Enter> or <Enter><N><P>

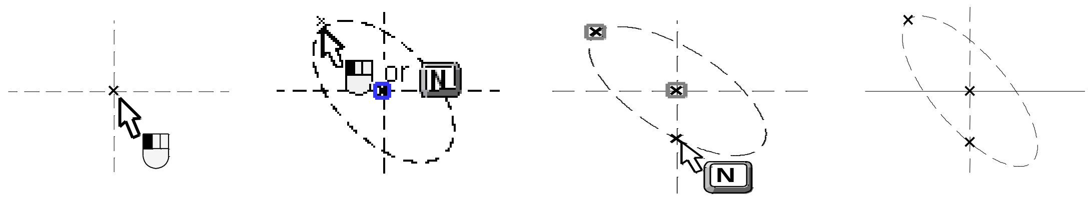


2. Center at node, through two nodes

Upon calling the command, use the following sequence of options:

<Enter><Enter><N> or <Enter><N><N>

These combinations create an ellipse with the center at a node, the first semi-axis ending at a node, and the second through another node. To create this type of ellipse, select the desired node as the ellipse center. Next, select another node for the ellipse's first semi-axis to pass through. The node can be selected using the option <Enter> or <N>. Then move the cursor over the node defining the position of the second semi-axis and type <N>. The required ellipse will be created.

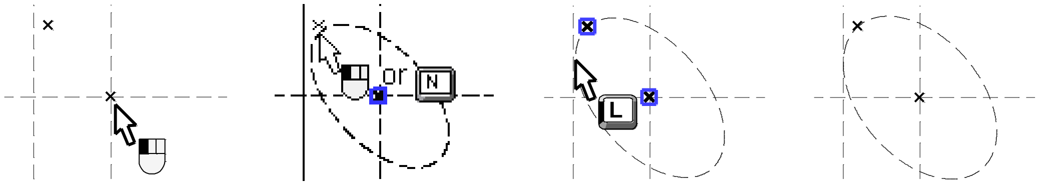


3. Ellipse with the center at a node, passing through node, tangent to line

Upon calling the command, use the following sequence of options

<Enter><Enter><L> or <Enter><N><L>

These option combinations create an ellipse with the center at a node, the first semi-axis ending at a node and the second semi-axis defined by tangency to line. To create this type of ellipse, select the desired node as the ellipse center, then use the option <Enter> or <N> and select another node for the ellipse's first semi-axis to pass through. Then move the cursor over the line to be making tangency with the ellipse and type <L>. The required ellipse will be created.

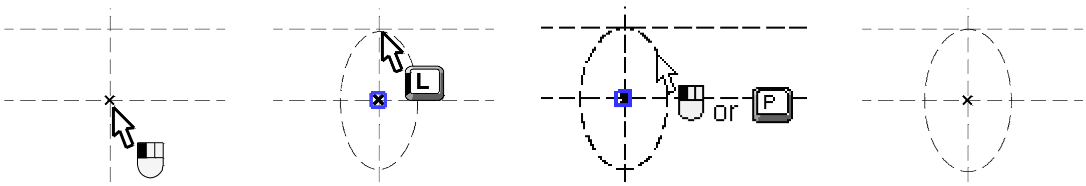


4. Ellipse with the center at a node, tangent to line, using parameter

Upon calling the command, use the following sequence of options:

<Enter><L><Enter> or <Enter><L><P>

These option combinations create an ellipse with the first semi-axis defined by tangency to a line at the tip of the semi-axis and the second semi-axis driven by the parameter. To create this type of ellipse, first select the desired node as the ellipse center. Next, use the <L> option and select the line making tangency with the ellipse at the tip of the semi-axis. Then move the cursor over desired position and click . The exact value of the second semi-axis length can be entered in the property window in transparent mode or in the parameters dialog box (the option <P>**).** The required ellipse will be created.

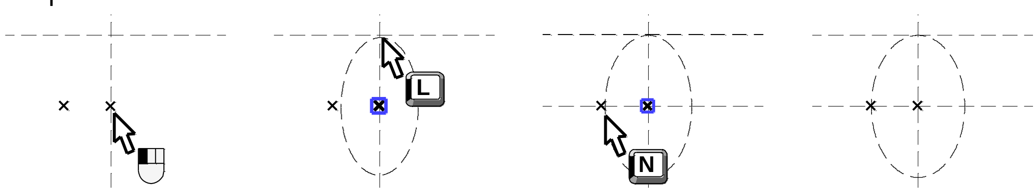


5. Ellipse with the center at a node, tangent to line, passing through node

Upon calling the command, use the following sequence of options:

<Enter><L><N>

This combination of options creates an ellipse with the first semi-axis defined by tangency to a line at the tip of the semi-axis and the second semi-axis ending at another node. To create this type of ellipse, first select the desired node as the ellipse center, and then select the line making tangency with the ellipse at the tip of the semi-axis. After that, select a node to define the position of the ellipse's second semi-axis. The required ellipse will be created.



The following techniques can be used in the mode of constructing ellipses passing through a node:

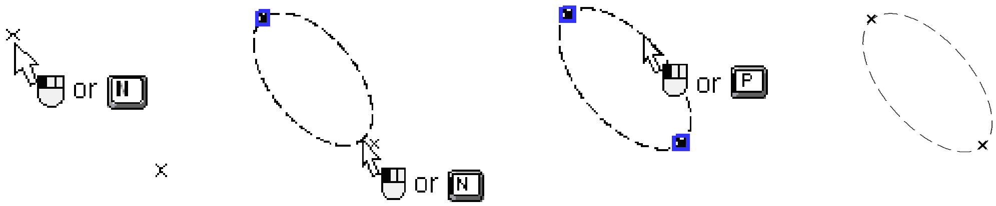
1. Ellipse passing through two nodes, using parameter

Upon calling the command, use the following sequence of options:

<Enter><Enter><Enter> or <Enter><Enter><P>

or <N><N><Enter> or <N><N><P>

These combinations create an ellipse, passing through two nodes, with the second semi-axis defined by a parameter. To create this type of ellipse, move the cursor over the first node and press the option <Enter> or <N>. Next, with the same options select the second node. Then move the cursor over desired position and click . To enter the exact value of the second semi-axis length, use the property window or the parameters dialog box (the option <P>**).** The required ellipse will be created.

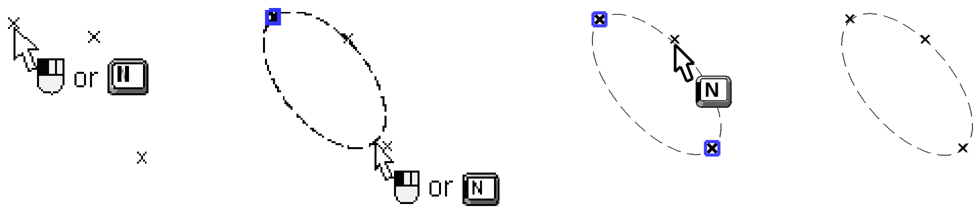


2. Ellipse passing through three nodes

Upon calling the command, use the following sequence of options:

<Enter><Enter><N> or <N><N><N>

These option combinations create an ellipse passing through two nodes, with the second semi-axis defined by the condition of passing through the third node. To create this type of ellipse, select three nodes using the <N> option (the first and second nodes can also be selected by the option <Enter>).

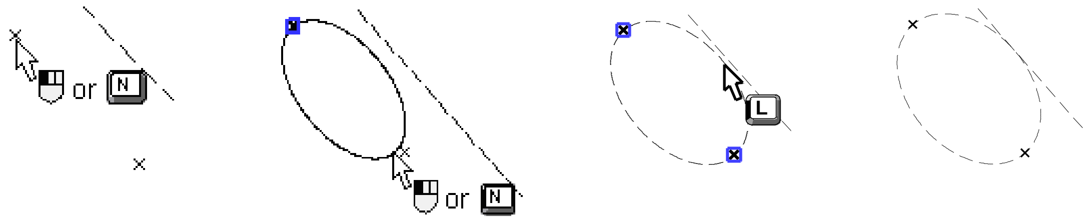


3. Ellipse passing through two nodes, tangent to line

Upon calling the command, use the following sequence of options:

<Enter><Enter><L> or <N><N><L>

These option combinations create an ellipse, passing through two nodes, with the second semi-axis defined by tangency to the line at the tip of the semi-axis. To create this type of ellipse, use the option <Enter> or <N> and select the first node, then, using the same options, select the second node. After that, move the cursor over the line that will be making tangency with the ellipse at the tip of the second semi-axis. The required ellipse will be created.



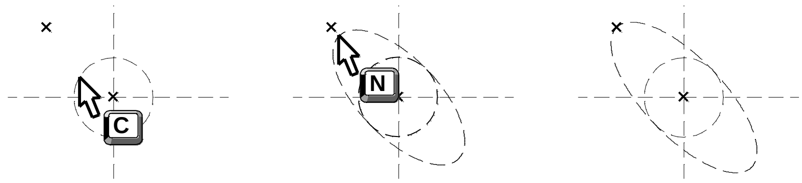
Another two types of ellipses can be constructed in any ellipse creation mode:

1. Ellipse tangent to a circle, passing through node

Upon calling the command, use the following sequence of options:

<C><N>

This set of options creates an ellipse with the first semi-axis defined by the radius of a circle concentric with the ellipse. The second semi-axis is defined by selecting its end node. To create this type of ellipse, select an existing circle using <C> option, and then move the cursor over the node to become the end of the second semi-axis, and type <N>. The required ellipse will be created.

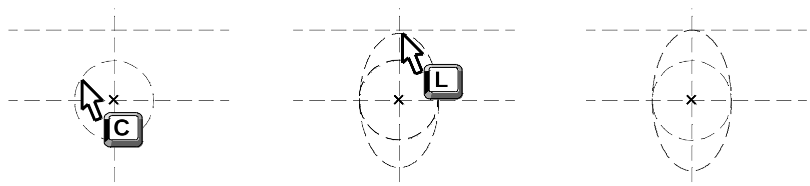


2. Ellipse, tangent to a circle and a line

Upon calling the command, use the following sequence of options:

<C><L>

This set of options creates an ellipse with the first semi-axis defined by the radius of a circle concentric with the ellipse. The second semi-axis defined by tangency to a line. To create this type of ellipse, select an existing circle using <C> option, and then move the cursor over the line to define tangency to the ellipse, and type <L>. The required ellipse will be created.



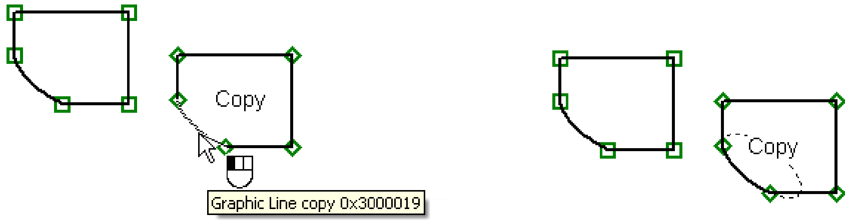
When creating ellipses by any of the above means, you can select elements using the following icons:

	<N>	Select Node
	<C>	Select tangent Circle
	<L>	Select tangent Line
	<A>	Select axis of symmetry (straight Line)

Object snapping mode can be used as well.

Ellipses Created from 2D Projection, 2D Fragment or Copy

Such ellipses can be created in object snapping mode, provided that the appropriate parameter is set on the **Snap** tab of the command **SO: Set System Options**. Move the cursor over a graphic entity – full ellipse or elliptic arc that is part of a 2D projection, 2D fragment or copy. The line will highlight. Click . An ellipse will be created on top of the selected entity.



If the object-snapping mode is off, then only 2D projection entities can be used for creating an ellipse. This is done by selecting the desired projection on the drawing using the option. The selected projection will highlight and the cursor will gain a glyph . To create a construction line now, simply point the cursor to a projection entity – ellipse or elliptic arc and click .

The following options will become available in the automenu:

	<P>	Set ellipse parameters
	<Esc>	Cancel selection

Ellipse Parameters

When creating or editing ellipses, one can control various parameters. The geometrical parameter (the semi-axis length) can be defined in transparent mode using the property window. The general system parameters are controlled via the complete ellipse parameters dialog box via the option:

	<P>	Set ellipse parameters

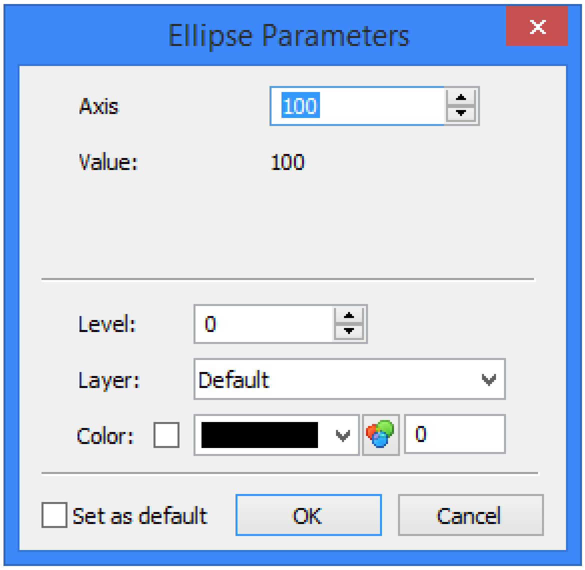
Axis. The length of one of the ellipse’s semi-axes. The input can be a number, a variable or an expression.

Value. Displays the numerical value of the “Axis” parameter.

Level. Places the ellipse on a certain visibility level.

Layer. The name of the layer the ellipse belongs to.

Set as default. Setting this flag means, the current dialog box settings will be used as the defaults for ellipses creation in future.



Editing Ellipses

Editing ellipses, just like any other construction entities is done via the command **EC: Edit Construction**.

Keyboard	Textual Menu	Icon
<EC>	"Edit > Construction > 2D Construction"	

This command allows changing ellipse parameters, assigning a name to an ellipse and deleting it.

An ellipse can be selected by pointing at with the cursor and clicking or via the option:

	<E>	Select Ellipse

The ellipse editing command, **EC: Edit Construction**, can also be accessed from within the command **EL: Construct Ellipse** via the option:

	<F4>	Execute Edit Construction command

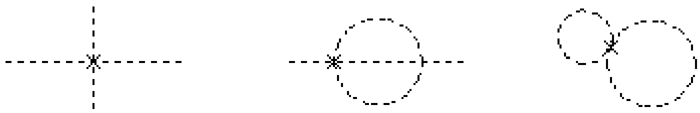
Detailed description of the command **EC: Edit Construction** can be found in the earlier chapter "Lines" (section "Editing Lines").

NODES

Node is a point whose coordinates are calculated depending on the node parameters or position of other model elements. Nodes are important construction elements in T-FLEX CAD. They represent start and end points of graphic lines. Nodes are directly involved in creation of most of the graphic elements. They also play an important role in creation of construction entities.

Creating Nodes

T-FLEX CAD supports nodes of various types, depending on the relation with other model elements. Most common are nodes constructed at an intersection or at a tangency point between two construction lines. Such nodes are displayed as small x-shape crosses.



Other types of nodes are displayed differently on the screen.

There are following types of nodes:

Node at intersection of construction lines. Such nodes are used most often in creation of parametric models. Its position is defined by the position of two construction entities and their intersection to which the node is related. In the case of multiple intersections between the entities, the particular intersection shall be identified.

Free node is defined by the absolute X and Y coordinates in the model coordinates. The values of a free node coordinates can be defined by variables. Such nodes are of limited use in parametric models being created, however, these are widely used in development of sketches, various diagrams and technical figures. Free nodes are useful in the cases when there is no strict requirement on positioning points of the image.

Node from fragment is defined by the position of another node located on a fragment of an assembly. This type of node is necessary for creation of parametric assemblies. It is used for relating some element of an assembly with a point on a fragment of this assembly.

Node created relative to another node. Its position is defined by an offset from another node. The offset values can be defined by constants or by variables. A node of this type can be used as an auxiliary fixing point in the cases when some element should be snapped at an offset position of the base node rather than to that node directly.

Node lying on a construction entity, at the specified distance from another node along the entity.

Node – a characteristic point of a construction entity. Among this type are nodes lying at a circle or ellipse center, at a start or end point of a spline or other curve.

Node on a curve, dividing the curve in a specified proportion.

Nodes placed at characteristic points of elements. This type includes nodes on dimension witness lines, on leader notes, at the ends of lines created by copying, etc.

Node dividing the distance between two other nodes in a given ratio.

For a point to become a node, the node needs to be created. This can be done in various ways:

By the command **N: Construct Node**, specifically designed for creating nodes.

By the option <Space> in the commands **L: Construct Line** and **C: Construct Circle**. In these commands, you can move the cursor over an intersection point of construction entities and press <Space>.

In the command **G: Create Graphic Line** when creating a graphic line.

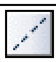
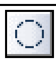
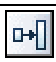
In the command **H: Create Hatch** when creating a hatch.

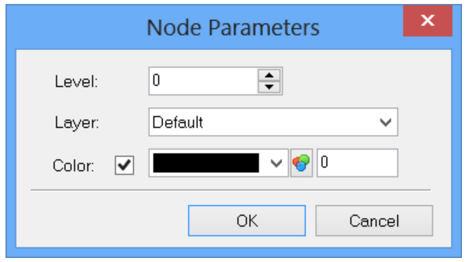
In the command **FR: Create Fragment**. As you add a drawing as a fragment into the current drawing, you can automatically create nodes from fragment in the drawing.

The last three techniques are described in the chapters that follow. In this chapter we will review in details the command **N: Construct Node**.

Icon	Ribbon
	Draw → Construct → Node
Keyboard	Textual Menu
<N>	Construct > Node

Upon calling the command, the following options become available in the automenu:

	<Ctrl><F>	Free mode on/off toggle
	<P>	Set Node parameters
	<L>	Select Line to create Node
	<C>	Select Circle to create Node
	<E>	Select Ellipse to create Node
	<S>	Select Spline to create Node
	<N>	Select Node for relative Node creation
	<F>	Select Fragment to create Node
	<R>	Select Fragment from list
	<W>	Select 2D Projection
	<F4>	Execute Edit Node command
	<Esc>	Exit command



The option / allows selecting the drawing mode - "free" or "constrained". The current mode is indicated by the kind of the option icon displayed in the automenu.

The option , called prior to node creation, opens a dialog box for defining the system-wide parameters, such as layer, level, color for new nodes. The same dialog allows to define position of various types of nodes being created.

Clicking constructs a node at the nearest intersection point of construction entities, while in "constrained" drawing, or construct a node at the position on the drawing directly under the cursor in "free" drawing.

The options , , and allow creating nodes lying on the selected entities.

To construct a node relative to another node, and also a node dividing the distance between two other nodes in a given ratio, the option is used.

The options , and help creating nodes based on fragments and on the lines of 2D projections.

Nodes Based on Construction Entities

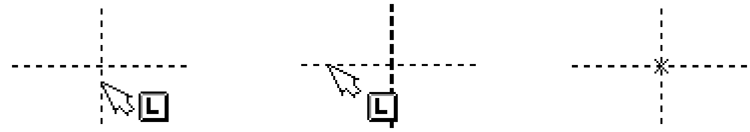
There are two main ways of creating nodes at intersections of entities in the command **N: Construct Node**:

1. Move the cursor over an intersection of two entities and click . A node is created at this point.

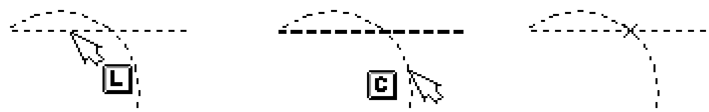
2. Subsequently create two construction entities. The node is created at their intersection point. Should there be two or more intersections, the one is used that was nearest to the cursor at the time of the last entity selection. The options used for selecting various-type construction entities are , , and .
3. The second way of creating nodes is recommended on crowded drawings and in the cases when more than two entities intersect in one point.

Examples of node creation:

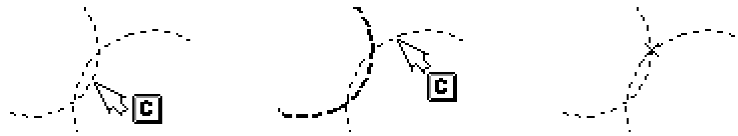
<L>,<L>



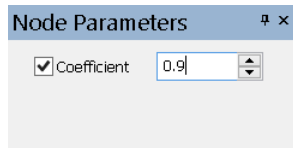
<L>,<C>



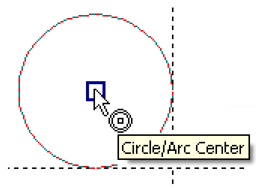
<C>,<C>



To create a node on a construction circle, select the circle using the option . A node will start rubberbanding along the circle. The position of the node on the circle can be defined roughly by mouse clicking, or exactly in the property window or in the parameters dialog box (the option).



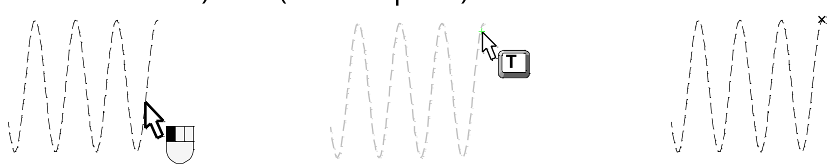
To construct a node at a circle center, select the circle and then use the option again. With snapping turned on, move the cursor over the center of the circle. The cursor will get the circle mark and the respective tooltip. Clicking now creates the node.

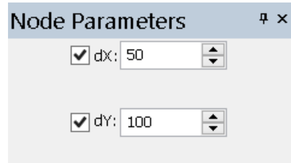


To construct a node at the start or end point of a spline or other curve, select the curve by clicking it with . A cross-shape node will start rubberbanding along the curve. Move the cursor over one of the endpoints of the selercted curve and engage the option

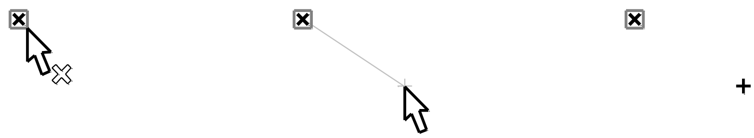
	<T>	Select Curve to create end Node

To construct a node on a curve, select the curve, and then define the position of the node on the curve. The position of the node can be roughly defined by , or specified exactly in the property window or the parameters dialog box (the option) by entering the parameter of the node on the curve in the range from 0 (the start point of the curve) to 1 (the end point).

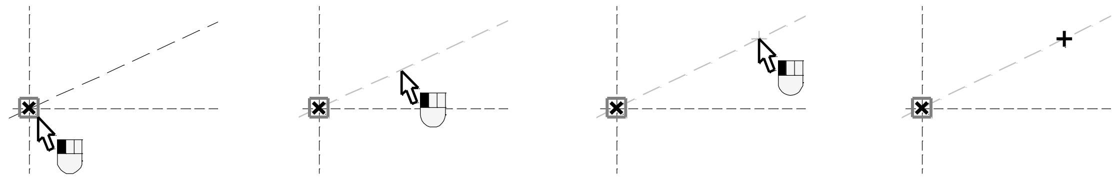




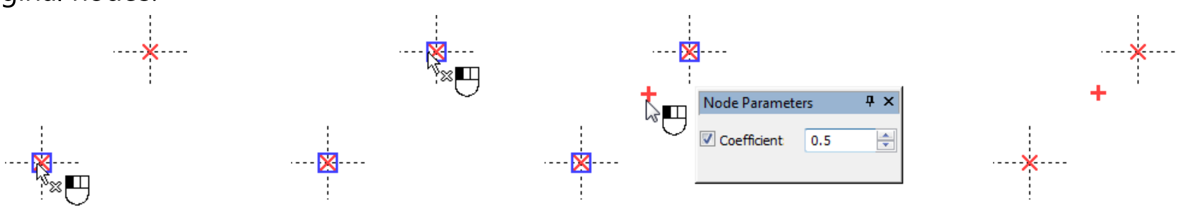
To construct a node relative to another node, with the specified offsets, select the reference node for offsetting using the option . A cross-shape node will start rubberbanding on the screen with a dashed rubberband connecting it to the reference node. The offsets can be defined freely by clicking or entered exact in the property window. The offset values in the property window can be specified by constants or variables.



or



To construct a node that divides the distance between two other nodes in a given ratio, it is necessary to successively select two original nodes by using the option. After selection of the second node, on the screen will appear the image of the node in the form of a small cross dynamically following the cursor. The node will move strictly along the line that goes through two selected nodes. The location of the node along the line can be set arbitrarily by pressing or by specifying in the properties window the exact value of the coefficient according to which the node being created will divide the segment between two original nodes.



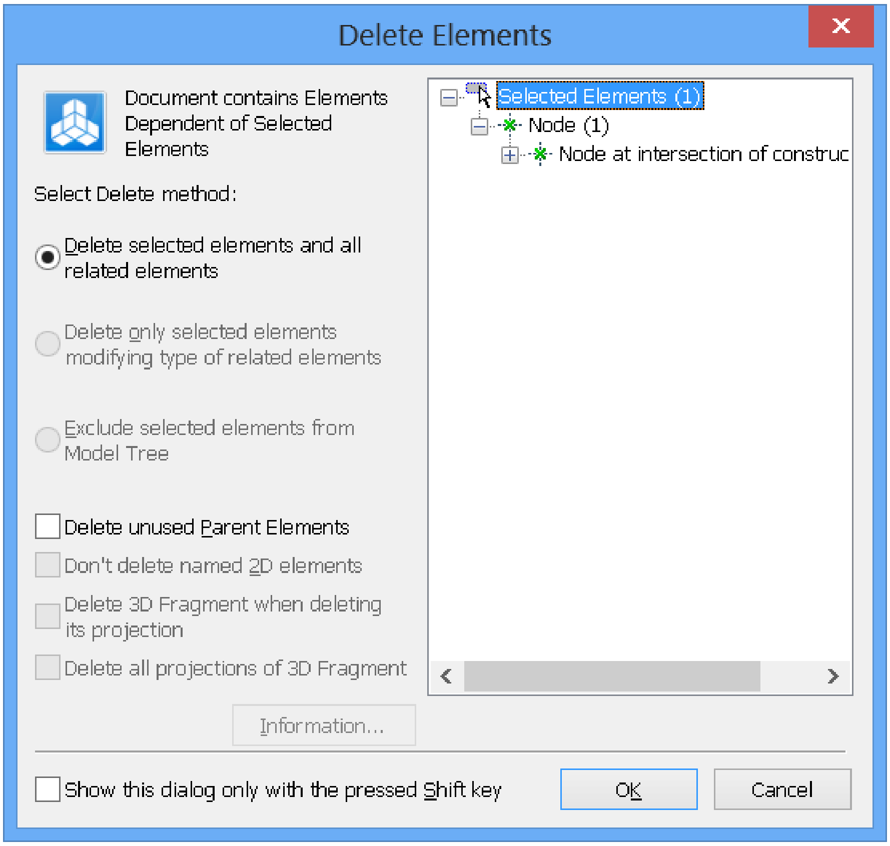
The node that divides the distance between two other nodes can also be constructed in case when the original nodes lie on the same line. The resulting node will also belong to this line. To create such a node it is required to successively select the first node (the option), then select the line (the option), then the second node (the option). Location of the node is specified in the same way as in the previous case.

In the command of node editing **EN: Edit node** there is a capability of selection of node's snapping: the snapping can be associated with intersection of construction lines, circle center, intersection of two snaps (for example, two perpendicular lines), etc. Possible snaps are highlighted with a color when the cursor of the mouse points at them and the command **EN: Edit node** is active.

To delete a node or change its parameters, use the command **EN: Edit Node**:

Keyboard	Textual Menu	Icon
<EN>	"Edit > Construction > Node"	

Selecting a node by highlights the node and the construction entities whose intersection defines the node position. At an attempt to delete a node referenced by other drawing elements, a dialog of the command for deleting the elements will emerge on the screen with specification of dependent elements and a list of possible actions of the system.



One can also select a node for editing from the command **EC: Edit Construction**. Using the option <N> in the command **EC: Edit Construction** automatically brings the system into the command **EN: Edit Node**.

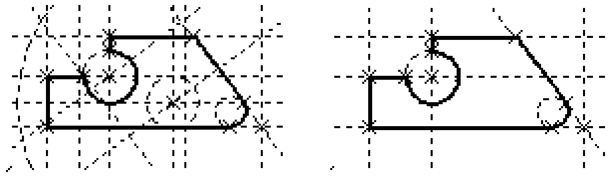
> Nodes and other construction entities can be hidden at any time. To do this, enter the command **SH: Set Levels** and set the lower limit of the visible levels for "Construction" greater than the "Level" parameter value assigned to these entities. By default, all elements have level "0". Setting the lower limit of the visibility range simply to "1" hides construction entities from display.
>
> One can also use layers for making construction entities invisible. Place those on some layer, for example, "Construct", and then make this layer invisible in the command **QL: Configure Layers**.

The display size of node symbols can be modified. To do this, use the command **SO: Set System Options**. The size in pixels can be specified in the item **2D > Node size** in the command dialog.

If extra nodes or construction entities were created for some reason along the design process, these can be quickly deleted using the command **PU: Delete Unused Construction**.

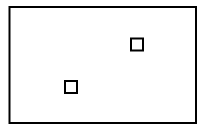
Icon	Ribbon
	Edit → Additional → Purge
Keyboard	Textual Menu
<PU>	Edit > Purge

This command will delete all construction entities that are not used in the model for defining graphic elements.



“Free” Nodes

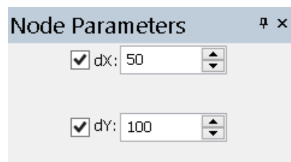
The main approach to creating drawings in T-FLEX implies use of nodes on the intersections of construction entities. However, the system also supports so-called “free” nodes. These nodes are not the points of entity intersections; rather, these are defined in absolute coordinates. Such nodes can be used just as well as usual “constrained” nodes for creating either construction entities or graphic elements. Free nodes are displayed as squares.



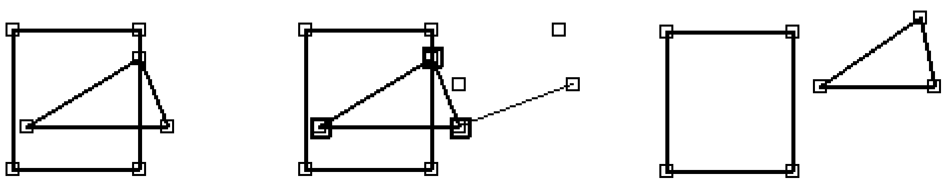
To create such nodes, turn on “free” drawing mode in the command **N: Construct Node** by using the option /.

Object snapping engages in free drawing mode similar to that provided in the sketching command. Besides, one can use the grid, with its settings defined in the command **QG: Change Grid settings**. The grid helps positioning created nodes more accurately.

To create a node, point the cursor to the desired location on the drawing and click . The node will be created right under the cursor. The exact position of the node on the drawing can be specified in the property window.



A feature of “free” nodes is the provision for moving such node or a group of nodes and, therefore, all elements related to them, by the command **EN: Edit Node**.



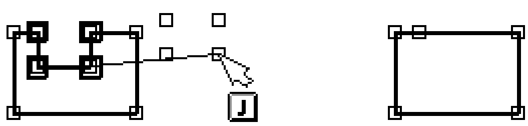
This is impossible for “constrained” nodes. The latter can only be moved by relocating construction entities used for the nodes creation.

As in the case of other drawing elements, multiple selection is done by the option <*>, or by clicking while holding down the <Shift> key (adding to the list of selected), or <Ctrl> key (excluding form the list of selected). Generally speaking, the following options are available in the command **EN: Edit Node**:

	<P>	Set selected Element(s) Parameters
	<V>	Dynamic model regeneration mode
	<O>	Create name for selected element
	<J>	Join free nodes
	<B>	Break node
	<F>	Convert to free node
	<N>	Select existing node
	<I>	Select Other element
	<Del>	Remove selected Element(s)
	<Esc>	Cancel selection

The mode of dynamic recalculation is enabled by the option. When enabling this mode the node's editing automatically leads to redrawing of the elements connected with the node. At the same time preview of the elements is shown with the same quality as the final result. The mode of dynamic recalculation of the model enhances the editing's process intuitiveness. It can be used for editing, for example, schemes, plans, etc.

Let us elaborate on the options and . These options affect a group of nodes, one of which can be constrained, and the rest - "free". The option unites several nodes into one, adjusting the graphics accordingly.



The option splits a node at a meeting point of multiple graphic entities. Each of the entities gets its own node whose position you can modify.

Keep in mind that "free" nodes are not recommended for use in parametric drawings. Drawings based on free nodes are similar to those supported by other CAD systems, and lack the advantage of parametric geometrical relations.

The option is available for nodes, not related to construction entities (for example, nodes from fragment or from 2D projection). This option allows breaking the relation between the node and its original references by converting it into a free node, whose position will not change under modifications to the original reference elements.

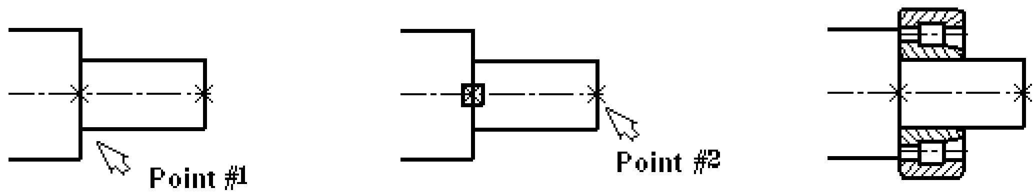
The option is an additional tool for multiple selections. To add a node to the list of selected, simply engage this option and pick the desired node.

Nodes from Fragment. Node Names

This way of node creation is quite important for support of parametric assemblies. The nodes from fragments provide the means for "tying up" construction entities and graphic elements of the current assembly to the fragment.

> Nodes from a fragment can be created automatically when inserting a fragment if in the settings of the system (the **SO: Set System Options** command, the **Fragments** tab) the **Create named nodes automatically** flag is enabled. In this case while inserting the fragment the nodes will be created on the basis of all named nodes of the fragment.
>
> In addition, nodes from a fragment can be created automatically while creating dimensions and other elements with the use of object snapping if in the **SO: Specify system settings** command, the **Snaps** > **Priority** tab, the **Fragment nodes** parameter is enabled. When this flag is disabled, then for creation of nodes on the fragments we need to perform the actions described below.

For example, you assembled a bushing on your drawing by the command **FR: Create Fragment,** and now want to create a dimension on the outer diameter of the bushing.



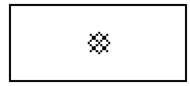
Since a dimension can't be created without nodes, you need to create two nodes from this fragment.

Nodes from fragment can be created in the command **N: Construct Node**. With the object snapping mode engaged and the flag **Fragment Nodes** set (the tab **Snap** > **Priority** of the command **SO: Set System Options**), a node from fragment is created as follows. Move the cursor over an end of a fragment graphic line. A node will highlight at the end of the line, with a tooltip saying "Fragment Node". Clicking creates a node from fragment.

With the object snapping mode off, first select a fragment via either of the options, or . The selected fragment will be highlighted.

If the fragment contains the named nodes, they will immediately be seen on the drawing. If necessary, it is also possible to display on the drawing the names of these nodes by enabling the option:

	<M>	Show Fragment Node Names



After that, nodes from the fragment can be created by pointing the cursor to the desired nodes among the highlighted ones and clicking .

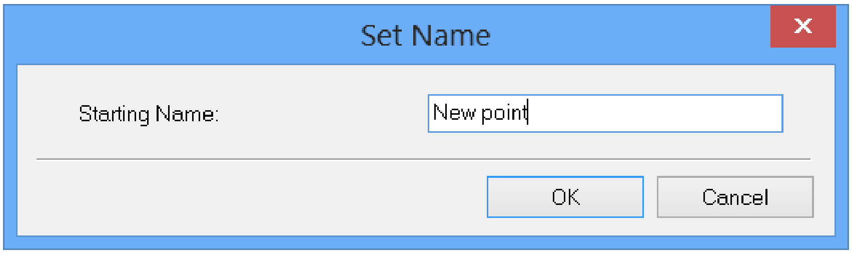
Created nodes from fragment are displayed as crossed diamonds.

The option <A> highlights and makes available for selection all nodes that exist in the fragment.

	<A>	Show all Fragment Nodes

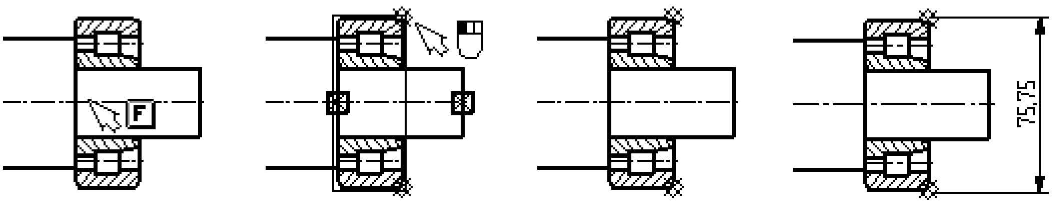
To assign a name to a node of a fragment drawing, use the following option under the command **EN: Edit Node**:

	<Ctrl><N>	Create Name for selected Element



Any name is allowed. The node now becomes named and can later be “exposed” upon assembling this drawing elsewhere.

Once the nodes from fragment are created, make the dimension via the command **D: Create Dimension**:



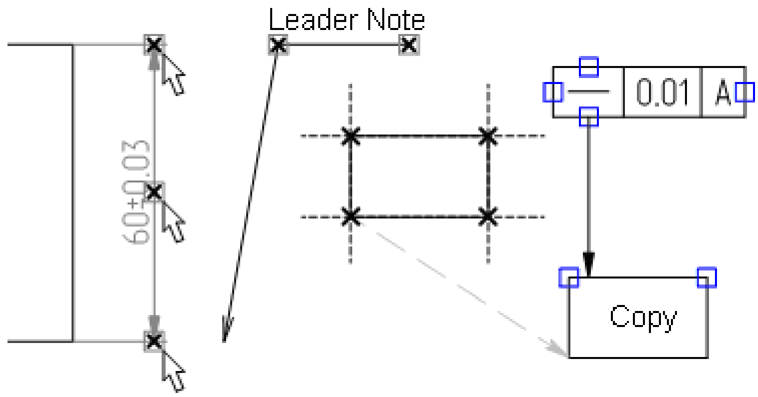
Nodes from 2D Projections

To create nodes based on the entities of 2D projections, one can use the option . Upon calling the option, select the desired projection by . The selected projection will be highlighted. After that, moving the cursor over the endpoints of graphic entities on the projection will highlight their nodes. Clicking creates a node from projection.

Option allows the user also to project nodes from a 3D model. After calling the option and selection of projection, it is sufficient to indicate a required 3D node (in 3D window or in the tree of a 3D model). A free node, which is a projection of the given 3D node, will appear on 2D projection.

Nodes Lying on Characteristic Points of Entities

Such nodes can only be created in the object snapping mode. This type of nodes includes nodes lying on dimension witness lines, leader notes, tolerances, as well as on the endpoints of graphic lines copied from or belonging to 2D projections; and also the nodes at the center of the construction line of a circle or an ellipse, the nodes at the end/start points of the spline and other curves. The respective option should be checked on the **Snap** > **Priority** tab under the command **SO: Set System Options.**



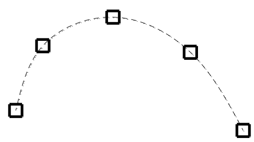
SPLINES

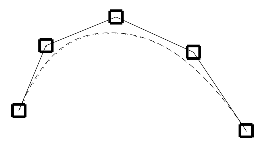
Main Concepts

The construction spline entities allow creating various curves. Unlike the straight construction lines, the splines have finite length. In general, spline-handling techniques are not different from those used for other construction entities. Nodes are created at intersections and tangency points. Graphic entities and hatch contour segments can be constructed along splines. Spline selection is done in many commands using the <S> option (the same key is used for selecting other curves as well, such as functions, offset curves and paths). T-FLEX uses NURBS-type splines.

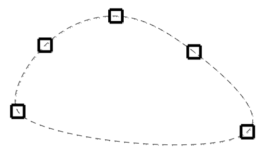
A spline is created based on a set of nodes that represent the defining points of the spline. Therefore, modification of the node positions will result in a change to the shape of the curve constructed based on these points.

Splines belong to two main types: passing through the nodes directly and using the nodes as vertices of the control polygon.

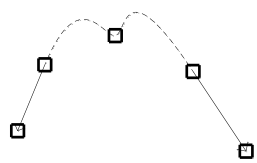




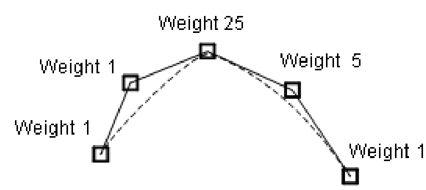
There are also closed splines.



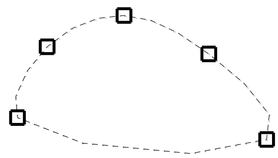
End-point conditions can be defined for splines through points via tangency vectors that are constructed based on nodes as well.

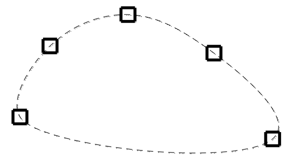


The control polygon nodes can be assigned weights. The more the node's weight, the closer the curve will pass to this node than to the neighboring ones. Vise versa, the lesser the weight, the smaller is the influence of the node on the curve shape.



Splines appear on the drawings as polylines made of numerous straight segments. The number of segments and, therefore, the accuracy of the output can be controlled by specifying the number of tessellation segments between a pairs of neighboring defining nodes. Each section of the spline will be tessellated by this number of segments when output. The more segments are used, the higher quality and accuracy will be achieved in the image. However, a too high number of segments may cause delays in spline handling by the system.





Constructing Splines

When creating spline, one can either use the existing nodes or automatically create new ones (free nodes or at construction line intersections).

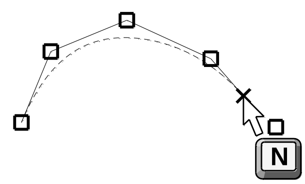
Call the command **SP: Construct Spline**:

Icon	Ribbon
	Draw → Construct → Spline
Keyboard	Textual Menu
<SP>	Construct > Spline

The following options become available to the user:

,	<Ctrl><F>	Free mode on/off toggle
	<N>	Select Node
	<P>	Set Spline parameters.
	<T>	Click to select tangent Node.
	<O>	Create Spline in Polar Coordinate System
	<A>	Select axis of symmetry (straight Line)
	<G>	Select Graphic line
	<F4>	Execute Edit Construction command
	<Esc>	Exit command

Upon entering spline creation command, the user can use or <N> in order to set the defining nodes of the spline. The curve being created will be rubberbanding on the screen. In the case of control polygon type splines, the polyline will be displayed along with the spline curve.



Now, the option for finishing the spline input becomes available in the automenu that can be used for completing spline creation.

	<Ctrl+ Enter>	Finish Spline input

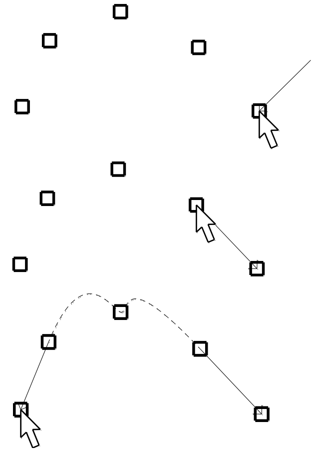
When creating a control polygon type spline, the weights of each particular point can be defined with the help of <P> option.

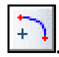
To define a spline with end-point tangency conditions, follow these steps:

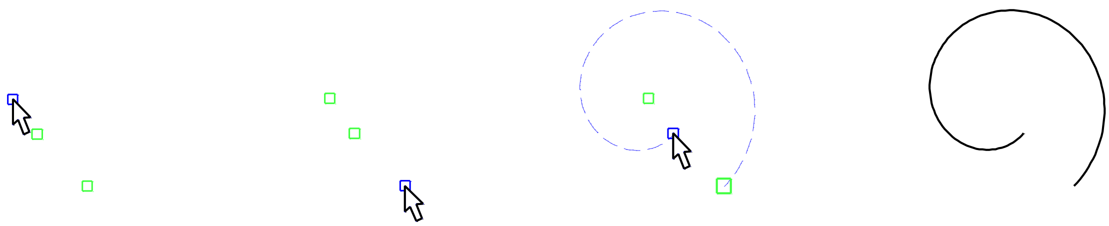
Use the option  to define start tangency direction.

Specify the desired sequence of nodes (two minimum).

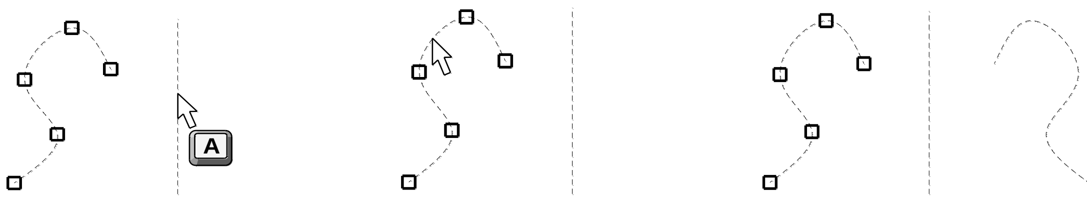
Use the option  and define the end tangency condition.



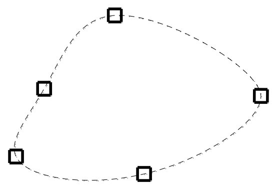
To construct a spline in a polar coordinate system, use the option . This option allows creating a spline passing through two points with tangency directions defined there. To create such a spline, define a coordinate system, the start and end points of the spline and the tangent angles at the ends. The angles are defined in the polar coordinate system by the ratio of degrees to millimeters. This way of defining a construction entity may be used, for instance, in camshaft design.

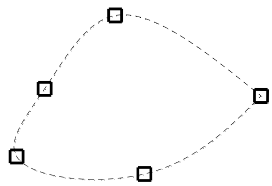


To create symmetrical splines, first engage the option  and select the symmetry axis, and then select the desired spline.

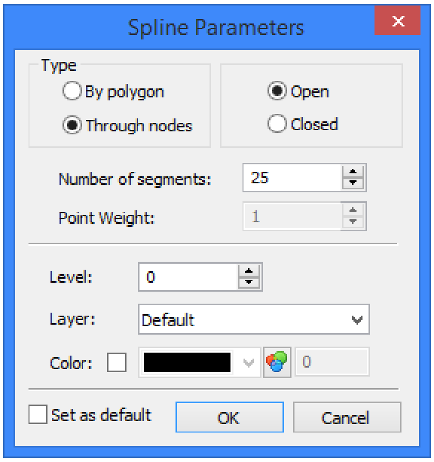


Please note special issues in creation of closed splines. If the start point was selected as the end point as well, the spline will be closed yet not necessarily smooth at the start-to-end connection. To impose such smoothness, set "Closed" option in the parameters dialog box and do not connect the rubberbanded spline to the start point.





Spline Parameters



The parameters of a spline created in a Cartesian coordinate system can be defined or modified via the option.

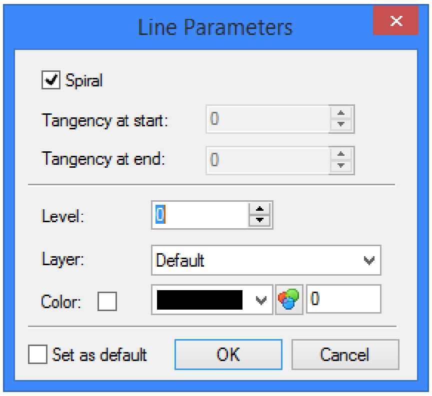
Type. This parameter defines the spline type (**By polygon**, **Through nodes**). The type can only be selected at creation time.

Next stands the spline option – **Open** or **Closed**.

Number of segments defines the number of tessellation segments between two neighboring spline nodes on a plot. This parameter can be defined by a variable.

Point weight. This item is used in creation of splines by control polygons. The weight parameter must be greater than zero.

Level, layer and **color** are defined and used in the same way as in all the rest construction entities.



A different set of parameters is used for splines defined in polar coordinates, as follows:

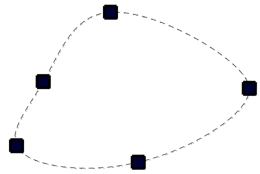
Spiral. With this flag set, a spiral is constructed with the center in the first point, start in the second point, and the end in the third point. Without this option, the following two parameters are used:

Tangency at start. Tangency at end. Define the angles from horizontal of the tangencies in the spline end points.

Level, layer and color are defined and used in the same way as in all the rest construction entities.

Editing Splines

Editing splines includes changing spline shape, adding or deleting defining nodes and modifying various parameters.



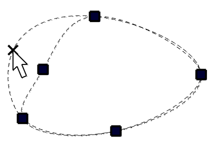
Spline editing is done in the command **EC: Edit Construction**.

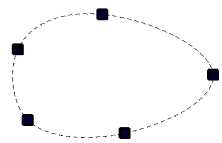
Once a particular spline is selected for editing (by pointing to with the cursor and clicking), then the spline curve will be highlighted together with the defining nodes.

The following options become available in the automenu:

	<Enter>	Select the nearest defining node of the spline to modify
	<P>	Set construction Line parameters
	<V>	Dynamic model regeneration mode
	<Y>	Create Name for selected Element
	<I>	Selected Other Element
	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

Once a defining node of the spline is selected by clicking , it can be reassigned by selecting another node, delete, or add a new one. Once a defining node is selected, the spline begins rubberbanding, following the cursor, just like at creation time. A following mouse click selects another node or creates a new one. For convenience, the old node gets deleted if it was not referenced by any other element.



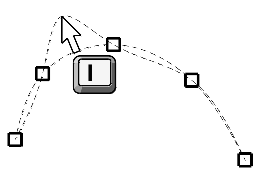


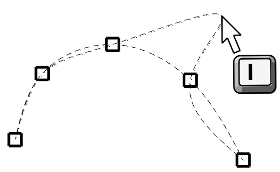
Note that the node position and, therefore, the shape of the curve, can be modified in the node editing command **EN: Edit Node**.

To add a new defining node use the option <I>.

	<I>	Switch to “Insert Point” mode

Note that the insertion of the new node will be before or after the selected node, depending on where the cursor was at the instant of option activation.





To fix the new node, click .

If the node selected for editing is part of the control polygon, its weight can be modified by typing <P>.

When editing defining nodes of a “through nodes” spline, this option is unavailable.

OFFSET CURVES

Offset curves are created by offsetting an arbitrary geometrical object by a specified amount. Offset curves are created based on existing curves (splines, ellipses, functions). The shape of an offset curve depends on the shape of the original curve and the amount of offset. The latter can be defined by a variable.

For such system entities as a circle and a line, offsets can be created on the fly together with the original entity creation.

A most typical application of offsets is pipe modeling. It is quite convenient for the user to draw just the centerline, and then create the offset lines of the pipe silhouette. Offsets are also widely used in developing structural and architectural drawings.

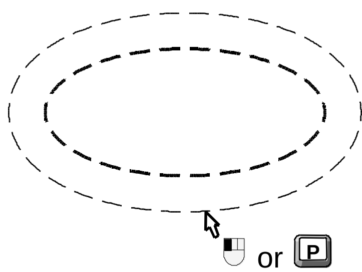
Creating Offset Curves

Offset curves are created in the command **TO: Construct Offset Curve**. The command is called as follows:

Icon	Ribbon
	Draw → Construct → Offset Curve
Keyboard	Textual Menu
<TO>	Construct > Offset Curve

Upon entering the command, the following actions become available:

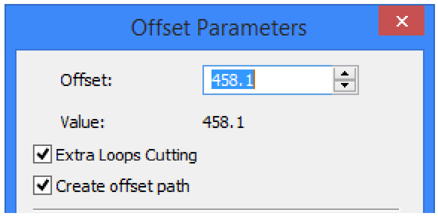
	<Enter>	Select element
	<P>	Set Construction Line parameters
	<S>	Select Spline
	<E>	Select Ellipse
	<F4>	Execute Edit Construction command
	<Esc>	Exit command



Offset curve creation begins with the selection of the reference element to use for offsetting. The reference element is selected by the cursor. For an accurate selection, use the options <S> "Select Spline" or <E> "Select Ellipse", while pointing the cursor to the respective element.

The selected element will highlight, and the offset curve will start rubberbanding. Move the cursor over the desired position and click use the option <P>.

In the latter case, the parameters dialog box will appear.

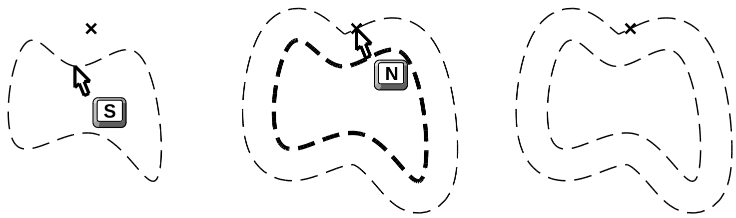


The offset distance can be entered in the “Offset” input box of the element parameters dialog. Positive offset values correspond to outer positions of the offset curve with respect to the reference element, while the negative – to inner positions, respectively. In the case of using the mouse input in offset creation, the parameters dialog is not displayed, and the value of the “Offset” parameter is derived from the cursor position. The offset curve is created on the side of the reference object pointed to by the cursor.

Offset position can also be defined using an existing node. To do this, select the reference element for the offset curve. This brings the following option in the automenu:

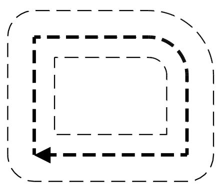
	<N>	Select Node

Use this option for selecting a node the offset curve will be passing through.



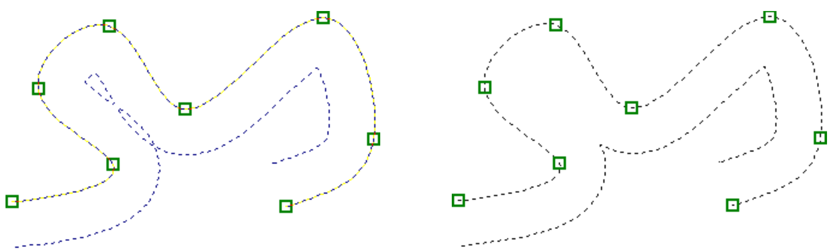
To reject a reference element selection, use the option

	<Esc>	Cancel selection



Offsets can be constructed to composite objects made of various-type entities, such as a spline and a line. To do this, first create a 2D path along the desired contour, and then use the path for offsetting. The corners of the reference curve become rounded on the offset curve, as rounding is adopted for handling offset corners in T-FLEX CAD.

When a spline offset is generated, self-intersection loops may occur in the offset contour. The offset parameters dialog box provides the option for loops stripping.

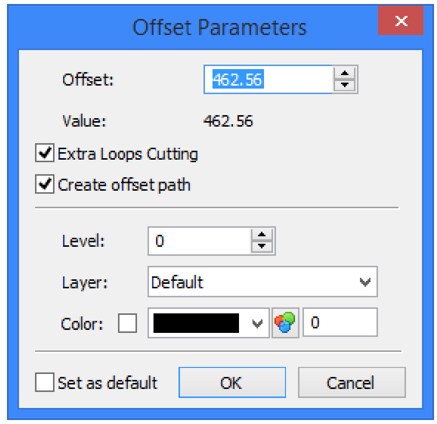


Self-intersection loops in offset generation

Offset with loops stripped

Offset Parameters

Offset parameters can be defined at creation or editing time. The parameters dialog box is called by the option .



Offset. Defines the distance between the reference element and its offset curve. The input can be a number, variable or expression.

Extra Loops Cutting. This parameter turns on the loop stripping mode. In this mode, all self-intersection loops are stripped off the offset contour.

Precise Circle and Arcs Calculation. This parameter is material only for creating an equidistant line with respect to a given 2D path. When activating this flag, instead of an equidistant construction line, the 2D path with more accurate processing of arcs and circles will be constructed.

Level. Places the offset curve on a certain visibility level.

Layer. The name of the layer the offset belongs to.

Set as default. Setting this flag means the current dialog box settings will be used as defaults for newly created construction entities.

Editing Offsets

Editing offsets, as well as other construction entities, is done in the command **EC: Edit Construction**:

Icon	Ribbon
	Draw → Additional → 2D Construction
Keyboard	Textual Menu
<EC>	Edit > 2D Construction

An offset can be selected by pointing the cursor and clicking it , or using the option

	<S>	Select Spline

The selected offset curve gets highlighted.

The following options become available in the automenu:

	<P>	Set Construction Line parameters
	<V>	Dynamic model regeneration mode
	<Y>	Create Name for selected Element
	<K>	Break link with variable

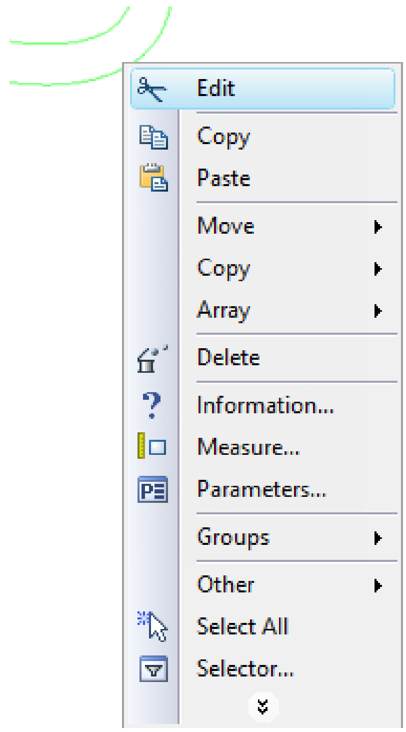
	<I>	Select Other Element
	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

Editing of an offset is none different from its creation.

The offset editing command **EC: Edit Construction** can also be accessed from the command **TO: Construct Offset Curve** using the option:

	<F4>	Execute Edit Construction command

The third way of selecting an offset curve for editing is available while no command is active (in the command waiting mode). Move the cursor over the offset and right-click for the context menu.



Selecting the menu item “Edit” launches the offset editing command. The item “Delete” allows deleting the selected element. Selecting the “Parameters” command opens the offset parameters dialog box.

Detailed description of the command “EC: Edit Construction” can be found in the earlier chapter “Lines” (section “Editing Lines”).

FUNCTIONS

T-FLEX CAD supports construction lines defined by explicit mathematical formulation. Such construction lines are called functions. To define a function, specify its definition type (parametric, plain, etc.), the start and end of the parameter range, various display parameters of the curve.

You can work in two modes: either using a predefined formula from the provided set, or creating a new function. The set of predefined functions is stored in the file “function.dat”. The file name is defined in the item **Function spline files** of the command **SO: Set System Options** tab **Files**. The file “function.dat” can be edited or replaced, if desired. Notations for the parameters, accepted in the file, are as follows: #1 – first parameter, #2 - second parameter of the function (can be optional). Follows is a detailed description of these parameters.

The system treats the resulting construction line of the function as a spline, therefore, the <S> key is used for its selection in various commands.

Defining the Function

The entities defined by a function are input via the command **FU**: **Construct Function Spline**”

Icon	Ribbon
	Draw → Construct → Function Spline
Keyboard	Textual Menu
<FU>	Construct > Function Spline

The following options are available to the user:

	<Enter>	Select nearest node or create node defining the function coordinate system
,	<Ctrl><F>	Free mode on/off toggle
	<N>	Select Node
	<P>	Set entity Parameters
	<A>	Select axis of symmetry axis (straight line)
	<F4>	Execute Edit Construction command
	<Esc>	Exit command

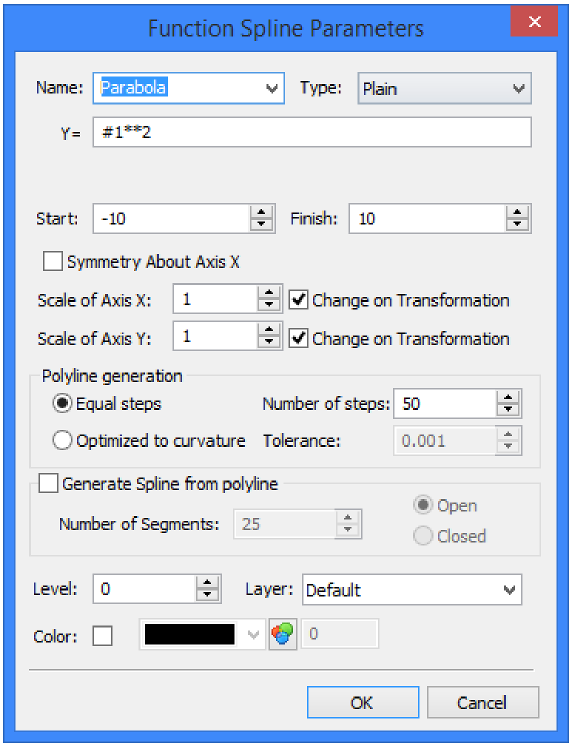
The function spline construction entity allows defining construction entities of virtually any kind. The main condition is possibility of defining construction entities in a form of functional relation between the coordinates.

The system provides an option of selecting predefined functions defining various curves, such as parabola, evolvent, spiral, etc. You can also define your functions independently, and then use them. For this purpose, create a descriptor file or append an already existing one with new formulas and values of other parameters defined in the function parameters dialog box.

The definition procedure includes two stages:

1. Defining the coordinate system position (X and Y). The coordinate system helps defining the desired position of the resulting entity on the drawing. The coordinate system is defined by sequential selection of two nodes. The first node defines the origin (0,0). The second node defines the direction of the X-axis. The Y-axis is defined automatically based on the origin and the X-axis positions.
2. Defining the functional relation and other parameters. This is done in the function parameters dialog box that appears on the screen after selecting the second node.

The following parameters are defined in the function parameters dialog box:



Name of the formula. One can select from the list a name of a standard function or define an arbitrary name for creating a new function.

Type, or the way of defining the function. Four different types of definition can be used:

– Plain definition in Cartesian coordinates (Y = f (X))

Example: Y= #1**2 defines a parabola.

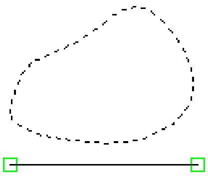
The notation #1 is used for the variable parameter (in this case - X). This special notation is used to avoid confusion with the system variable names that can be used in the expression defining the functional relation.

Besides variables, functions can be used as well as they are supported for use in the variable editor.

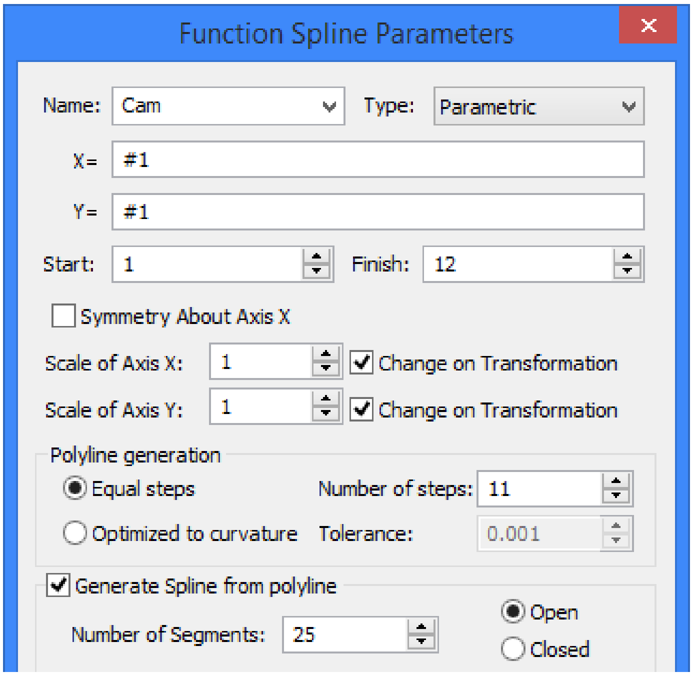
– Parametric definition in the Cartesian coordinate (X = f(t), Y = f (t)).

Example: X = #1, Y = #1 defines a straight line.

One can create, for instance, a database of cam coordinates, and then use a parametric definition for the cam contour. Use the database access functionality and define a parametric relation where the variable parameter #1 is the record number in this database. The specified number of steps must necessarily equate with the difference between the end and start values of the function variable parameter.



> The indicator of the end value, or the end record, in the database named “q” can be conveniently entered as the expression “q.#”, returning the number of the last record in the database q.



q

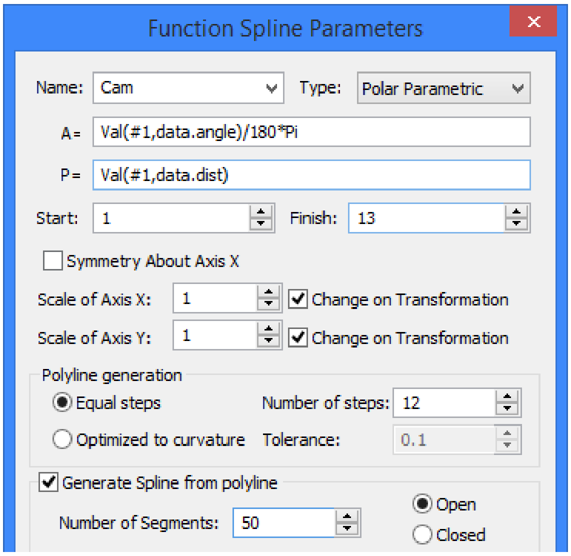
№	x	y
1	0	10
2	2	17
3	5	20
4	15	25
5	25	30
6	35	15
7	25	5
8	10	5
9	5	6
10	3	7
11	1	8
12	0	10

- Function in Polar coordinates (P = f (A))

Example: P = #2 defines a circle with the radius equal to the value of the parameter #2. #2 – is the second special notation that can be used in function defining expressions. It is equal to the distance between the nodes that define the coordinate system.

-Parametric definition in Polar coordinates (A = f(t), P = f(t)).

In T-FLEX system, such definition of functional relations may be convenient in a number of situations. For example, suppose, a database stores the values of angles and distances of a cam coordinates. Use the database access functionality and define a parametric relation where the variable parameter #1 is the record number in this database.



data

№	angle	dist
1	0	3
2	10	3
3	15	3
4	20	4
5	40	4
6	60	4
7	80	4
8	100	4
9	120	3
10	140	4
11	160	4
12	180	4
13	220	2

X=, **Y=** (or **A=**, **P=**) Depending on the type of the function being defined, these two fields describe the expressions defining X and Y (for functions in Cartesian coordinates) or A, P (for functions in Polar coordinates). The following notations are used: #1 – the first parameter, #2 – the second parameter of the function (can be optional).

The resulting function construction, as in the case of splines, is a polyline. The "Start" and "Finish" parameters define respectively the start and end values of the variable parameter, defining the beginning and the end ranges of calculating the polyline coordinates.

The "Polyline generation" group of parameters defines the way of calculating the intermediate point coordinates when creating the polyline:

Equal steps. This way implies the variable parameter to change from the start to the end value in equal increments. The number of steps is specified by the user. It defines the number of segments in the polyline being created. The more the number of steps, the higher-accuracy will be the polyline representation along the bends and the longer time will take various operations handling the created polyline.

The number of points used in the polyline creation is always equal to the number of segments plus one, and, therefore, is greater by one than the specified number of steps. The first point always corresponds to the start value of the variable parameter. The coordinates of the rest of points are based on the values of the variable parameter defined by the formula:

*Current value = Start value + I * Step*

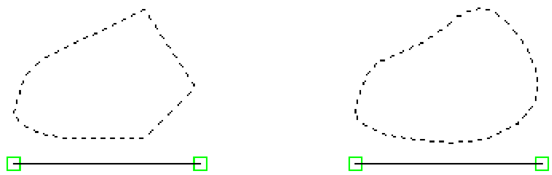
Step = (End value – Start value) / Number of steps,

I is the current step number (*1,2,..., Number of steps*).

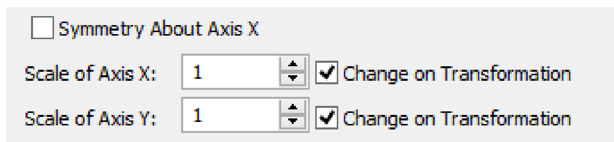
If polyline point coordinates are calculated using a database, the value of the variable parameter is usually the line number in the database. In this case, the current value of the variable parameter is replaced by the largest integer less than or equal to it. As a result, data could come at uneven steps from the database. To avoid this, always define the function parameters in such a way that the number of steps was equal to the difference between the end and the start values of the variable parameter.

Optimized to curvature. One could notice that the above approach is not always convenient for curves of complex shape, as the tessellation density was constant along “smooth” and “curved” zones unnecessarily. Optimization to curvature yields finer tessellation along high-curvature zones and, respectively, coarser otherwise. The criterion of the accuracy and quality in this case is the “Tolerance” parameter that defines the maximum permitted deviation of the calculated polyline coordinates from the true curve coordinates. The lesser is the tolerance, the more segments will be in complicated zones of the curve.

Once the polyline is built, it can directly be used as a construction entity. However, a possibility is provided for using the calculated polyline points for spline creation. For this, turn on the option “Generate spline from polyline”, define its type and the number of tessellation segments between two neighboring points of the spline for the spline tessellation polyline. This tessellation polyline will be the final output construction entity. Spline generation may be needed when a smooth curve is desired, while the number of defining points is limited.



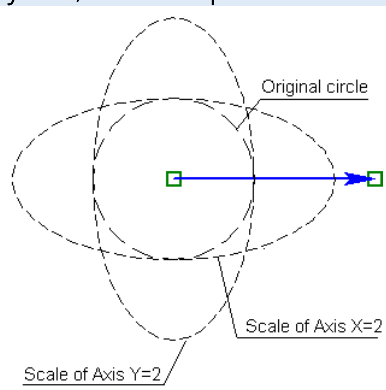
The “**Symmetry About Axis X**” flag among the function spline parameters allows mirroring of the created entity about the X-axis of the function local coordinates. (The X-axis passes through the two nodes selected at function spline creation).



If in future this construction line will be subjected to a symmetry transform (for instance, in copying or in translation), then the state of the flag of the transformed line may be changed by the system automatically.

To construct a spline symmetrical to a given one about an arbitrary line, use the option .

Additional parameters **Scale of Axis X (Y)** allow changing the scale along each axis of the function local coordinate system. The coordinate of each point used in the function creation is multiplied be the specified scale factor.

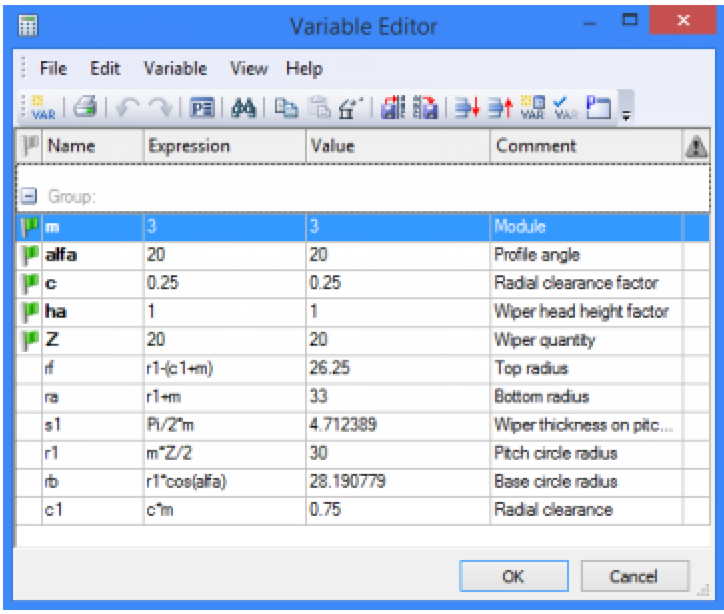


The “**Change on Transformation**” flags, to the right of the respective axis scale input boxes, allow/disallow automatic change of scale when subjecting the given entity to a scaling transform (as in copying or translation). The cleared flag prohibits automatic change of the respective scale factor, while the checked item allows it.

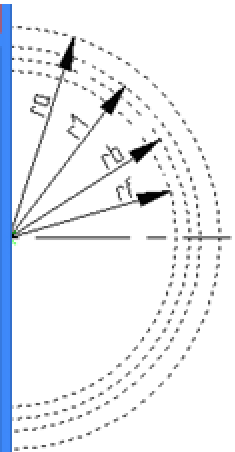
Note that special-type nodes are created at end points of function spline construction entities (as well as in other curves). These are created by using the option of the node creation command.

Evolvent Creation Example

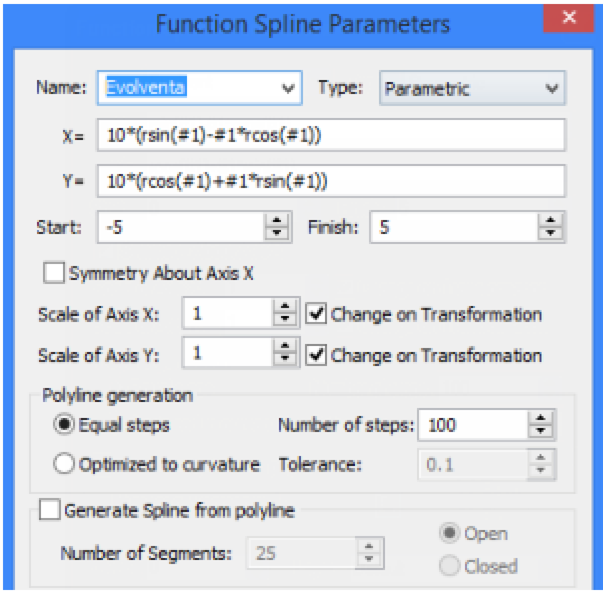
As an example, let's create a profile of a cogwheel. First, let's make all necessary calculations and constructions: circles, cog size, etc. relating these parameters by variables.

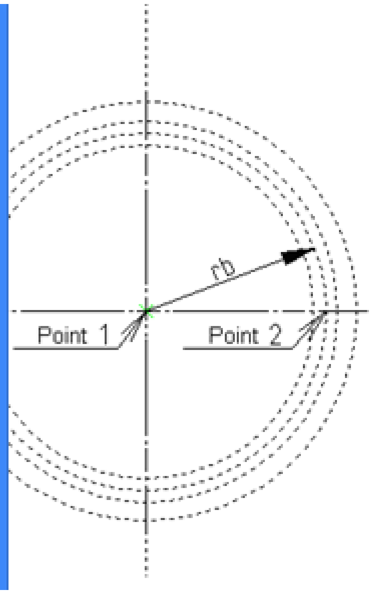


Name	Expression	Value	Comment
m	3	3	Module
alfa	20	20	Profile angle
c	0.25	0.25	Radial clearance factor
ha	1	1	Wiper head height factor
Z	20	20	Wiper quantity
rf	r1-(c1+m)	26.25	Top radius
ra	r1+m	33	Bottom radius
s1	Pi/2*m	4.712389	Wiper thickness on pitc...
r1	m*Z/2	30	Pitch circle radius
rb	r1*cos(alfa)	28.190779	Base circle radius
c1	c*m	0.75	Radial clearance



Then, using the "Evolvent" function, specify two points for defining the X-axis of the evolvent. Besides, the first point also defines the center of the main circle (rb). Each equation among the function parameters needs to be multiplied by the main radius of the wheel or by the #2 parameter, the latter being the case when defining the main circle radius by the second point.

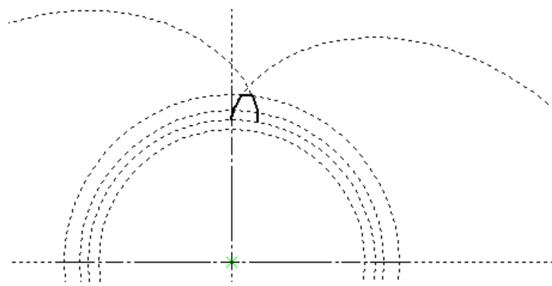




Besides, the start and end points (angles) can be defined among the evolvent parameters. This is optional, since the evolvent will be bounded by the specified radii of the wheel being created.

In the reviewed example (see the diagram) a general case of evolvent is limited to sampling an upper portion of a cog profile. This drawing is located in the folder `"Examples/2D Design/ Function Spline/Evolvent.grb"`.

Do not use this file as a template for designing cogwheels. Follow the design insights of this document and use refined formulas to create a complete profile of a cog. Use this drawing in future for creating other cogwheels.



Editing Function Spline

Editing of the function spline construction entities is supported by the command **EC: Edit Construction**.

Keyboard	Textual Menu	Icon
<EC>	"Edit > Construction > 2D Construction"	

After entering the command, you can reassign the defining nodes of the coordinate system or modify parameters. As for splines, use the option <S> for selecting function spline construction entities.

PATHS

A path is a construction line passing through a sequence of nodes. The segments between the nodes can be straight lines or a portion of a construction entity between two given nodes. The following construction entities can be used: lines, circles, ellipses, splines and other paths.

Constructing 2D Paths

The command **PA: Construct Path** is provided for constructing a 2D path.

The command is called by:

Icon	Ribbon
	Draw → Construct → Path
Keyboard	Textual Menu
<PA>	Construct > Path

The following actions become accessible upon entering the command:

	<Enter>	Select a node or create a node at the nearest construction line intersection
/	<Ctrl><F>	Free mode on/off toggle
	<P>	Set selected Element(s) Parameters
	<N>	Select Node
	<L>	Select Line
	<C>	Select Circle
	<E>	Select Ellipse
	<S>	Select Spline
	<F4>	Execute Edit Construction command
	<Esc>	Exit command

The 2D path creation procedure consists of selecting 2D nodes forming a sequence. After selecting the start node, you can select a construction line that connects this and the next node. Both nodes must belong to this construction line.

The step-by-step process is as follows:

1. Select the start node;
2. Select a construction line connecting the start node with the end node (optional);

3. Select the end node.

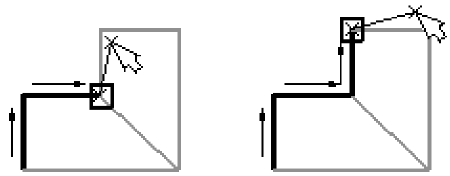
Selecting the start node of the path as the end node completes the path creation.

4. Confirm path creation or repeat the step 2. The end node selected in the step 3, becomes the start node for the next segment of the path.

Upon selecting the start node and the first segment, you can do the following:

	<Space>	Select Graphic line

This option allows defining a path contour along graphic lines. Keep in mind that this approach can only be used when the path segments coincide with the graphic lines. In the case of multiple selection choices, the cursor should be pointing to the desired graphic line when using the <Space> option.

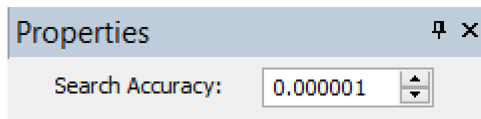


To speed up the process, one can use the option:

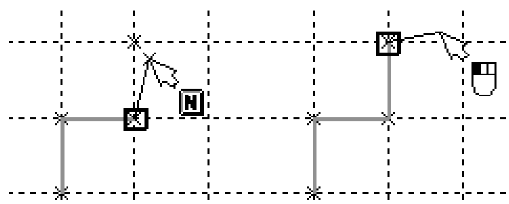
	<A>	Find contour automatically

This option will search for the next path segment automatically until the contour is closed (in the case of the closed path) or until reaches a dubious situation (when the path is ending or forking).

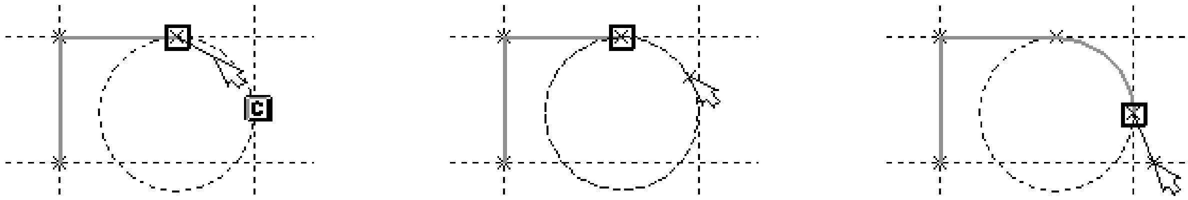
In both cases (when using the option <*Space*>, and upon automatic selection of the contour with the help of ) the system can select, as a next segment, the graphic line located at some distance from the last selected segment. This happens when the distance between neighboring nodes of graphic lines does not exceed the **search tolerance**. It determines the maximum allowed distance between nodes – line ends, for which these nodes are considered coinciding, and the graphic lines – connected. The search tolerance can be changed with the help of identically named parameter in the properties window.

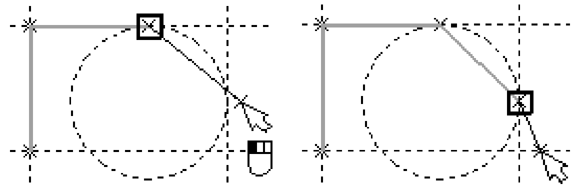


One can define a path using the same operations as in graphic line creation. In other words, one needs to define a sequence of path segment lines each having the start and the end node. To define the start or the end of a path segment line, select existing nodes (the key <N>) or create new ones (the key <Enter>) at intersections of construction line pairs.



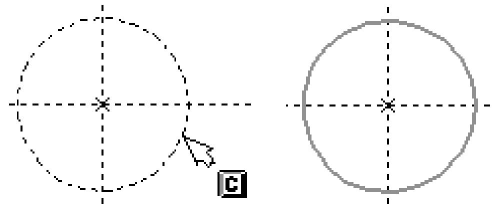
As in the case of graphic entity creation, an arc is defined by first selecting a node, and then the construction circle by typing <C>.





Otherwise, a straight line will be created between the two nodes as the path segment instead of the arc.

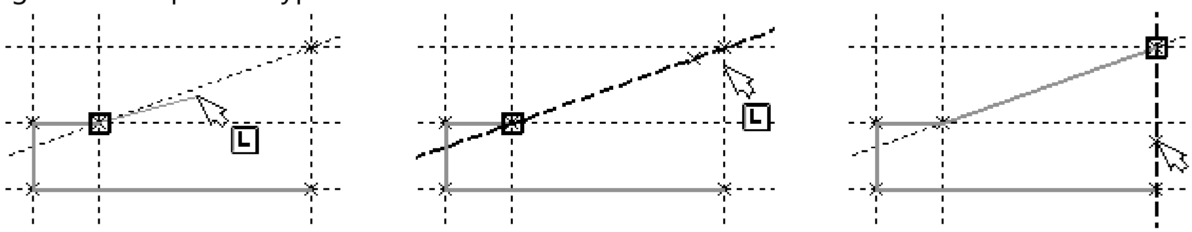
Including an elliptical arc or a spline or other curve segment in a path is similar to creating a circular arc.



To define a path represented by a full circle, then without selecting any nodes point the cursor to the circle and type <C>.

One can also construct a path from an ellipse, spline or other curve by using the options <E> and <S>, respectively.

In complex cases, when more than two construction lines intersect in one point, resulting in multiple overlapping nodes, the ending nodes of the path segments should be specified by selecting the intersecting line pairs hosting the desired node. This is done by using the keys <L>, <C>, <E>, <S>, standing for the respective types of construction entities.



In the cases when more than two construction lines intersect in one point but no nodes are created, the recommended technique is to create all necessary nodes by the command "**N: Construct Node". In this way, specify the exact lines whose intersections yield nodes.** After that, the path segments can be input using the <N> option.

To reject the last input path segment line, use the option

	<BackSpace>	Delete last Contour segment

All three above-mentioned techniques can be combined when defining a path.

If the end node of a closed path coincides with the start node, then the contour automatically closes and dehighlights, and an arrow is displayed in the end node indicating the direction of the defined path. This signals that the path has been completed.

A path can also be closed by using the option

	<Home>	Close Contour

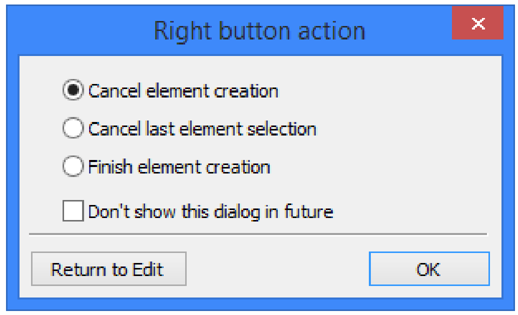
To complete definition of an open path, upon defining all segments use the option

	<Ctrl+ Enter>	Finish input

To cancel an action of path input, use the option

	<Esc>	Cancel selection

Whenever two or more path segments are input in the command **"PA: Construct Path"**, the right-click brings up the Right button action dialog box (this overrides the system settings).



In this dialog box one can: "Cancel element creation", which is equivalent to <Esc> option; "Cancel last element selection", which is equivalent to <BackSpace> option; "Finish element creation", which is equivalent to <End> option. One can also set the flag "Don't show this dialog in future". In this case, the dialog box will not come back again, and the right-click action will be the one set last together with the "Don't show this dialog in future" flag, per the system settings.

The button **[Return to Edit]** brings the user back to 2D path creation mode.

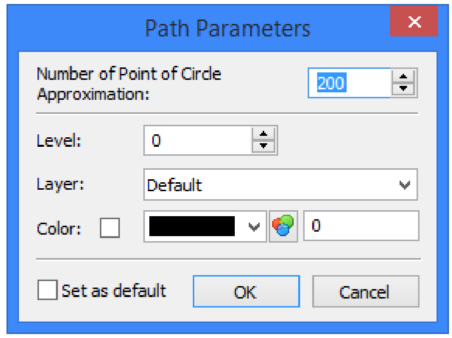
> Note that the same dialog is used in the **H: Create Hatch command. A default action setting made in this dialog box in one of these commands will work in both commands**. The selected option will be used in all newly opened documents until the end of the application session. To change the setting, close and reopen the application.

If the dialog box does not pop up upon while in the command being described, and some action is performed instead, then a default action was already assigned earlier in the session within this command or in **"H: Create Hatch"**.

2D Path Parameters

2D path parameters can be defined during its editing. The 2D path parameters dialog box is called by the option

	<P>	Set path parameters



Number of Point for Circle Approximation. This parameter defines the accuracy of drawings of circles and circular arcs that are components of 2D paths.

Level. Places the path being created on a certain visibility level used for hiding some elements from display as necessary.

Layer. This parameter assigns the path being created to a certain layer.

Color. This parameter defines the color used for the path display.

Set as default. Turning on this flag means the current dialog parameter settings will be used in future for constructing new construction lines.

Editing 2D Paths

By editing a path, one can add or delete nodes, select a different construction line to connect the end nodes of a particular path segment, as well as define new parameters.

Editing is done in the command **EC: Edit Construction**.

Keyboard	Textual Menu	Icon
<EC>	"Edit > Construction > Path"	

A path can be selected by pointing the cursor and clicking, or by the option:

	<S>	Select Spline

As a result, the selected path gets highlighted, and its nodes marked.

Editing the Type of a Particular Path Segment

To modify the type of a particular path segment, do the following steps:

Select a path;

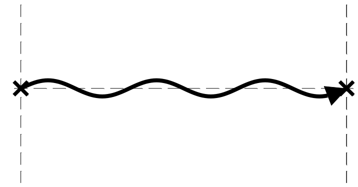
Using the mouse, select the path segment whose type needs to be modified;

Select a construction entity defining the new type of the path segment: line, circle, ellipse and spline (including other 2D paths). Selection of a construction entity is done using the appropriate option. The end nodes of the path segment being edited must belong to the selected construction entities;

Exit the particular path segment editing mode by right-clicking or pressing <Esc> on the keyboard.

Confirm changes by the option:

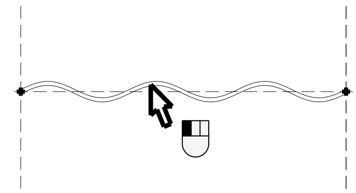
	<Ctrl+ Enter>	Finish input



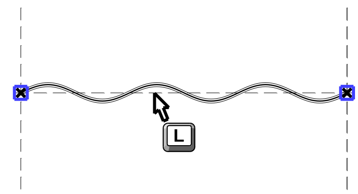
Let’s review an example explaining the process of editing a particular segment of a path contour. The diagram shows the original path.

The wave segment is to be replaced by a straight line.

Call the command “EC: Edit Construction” and select the path. The next diagram shows the situation after selecting the path.



The contour is highlighted, and the end nodes are marked by small boxes. Move the cursor over the contour segment to be edited (the wave line in this case) and click. The selected contour segment will be highlighted and its nodes marked with larger boxes. This state is shown on the next diagram.



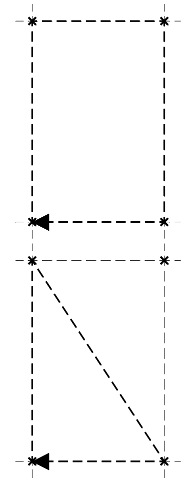
Select the straight line passing through the edited contour nodes by moving the cursor over and typing <L> key. The edited path segment now assumes the desired shape. The system still remains in the selected path segment editing mode. If transforming this segment is over, quit the current segment editing mode by right clicking or pressing <Esc> on the keyboard.

The icon becomes then accessible in the automenu. If there are no more modifications to the path, push or <*End*> key. What is left is just changing the graphic line type, if necessary.

> Similarly, one can replace a path segment with a circular or elliptic arc, or a spline segment, or a part of another path, if the circle, ellipse, spline or path is constructed based on the marked nodes. Simply use the appropriate option among <C>**,** <E> and <***S***>**.** If the new path segment was not created based on the marked nodes yet passes through them, then editing such path segment can be done using the option “Switch to ‘Insert Point’ mode” (the icon or the <I> key). This option will be fully described below.

The selected segment of the path between two nodes can be replaced as many times as necessary. The path segment will stay selected until the user quits by right clicking or pressing <Esc>. If a contour segment was modified using the option “Switch to ‘Insert Point’ mode” (the icon or <I> key), then the selected contour segment gets unselected after the change (no need to press <Esc>). Meanwhile, the system will remain in the path contour editing mode until the confirmation by or <End> key.

Deleting a Node inside Path Contour



To delete a node inside a path contour, do the following steps:

Select a path (point at by the graphic cursor and click);

Select the path segment the node belongs to (point at by the graphic cursor and click);

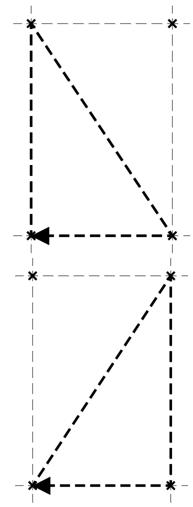
Select the node to delete (point at by the graphic cursor and click);

Delete the selected node (the icon or the <Del> key);

Confirm changes (the icon or the <End> key).

As a result, the new path segment passes through the two neighboring nodes.

Modifying a Node Position within Path Contour



To modify position of a node within a path, do the following steps:

Select a path (point at by the graphic cursor and click);

Select the path segment the node belongs to (point at by the graphic cursor and click);

Select the node to move (point at by the graphic cursor and click);

Move the node to the desired position (the segments connecting the node to neighbors will rubberband as the node is moved);

Fix the node (point the cursor at an intersection of construction lines and click , or type <N> in the case of using an existing node).

Confirm changes (the icon or the <End> key).

As a result of moving, the node will be connected with the neighbors by straight line segments, regardless of the former types of connecting entities).

Creation of Additional Nodes on a Path Contour

To create additional nodes on a path contour, do the following steps:

Select a path (point at by the graphic cursor and click);

Select the contour segment to split by new node(s) (point at by the graphic cursor and click);

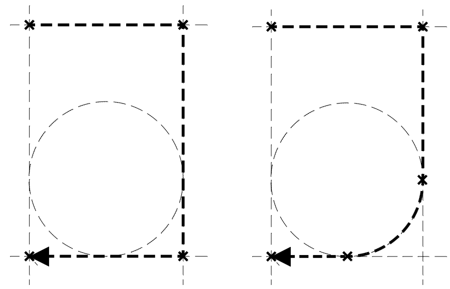
Turn on the point insertion mode (the icon or the <I> key), and click on a contour segment. The segment becomes split in two, with the new node between them. The node and the segments rubberband with the cursor, the solid line segment connecting to the previous node and the dashed line to the next one. The order of the nodes after the insertion will be determined by the system automatically, depending on the path contour direction. Do not click on a segment near a vertex, as, instead of adding a node, this will start moving the existing node;

Close the contour between the newly created node and the successive one. A shortcoming of the functionality is unavailability of the option “select graphic line”;

Contour input is complete once the closing node is selected, or the icon  or the <End> key is pressed.

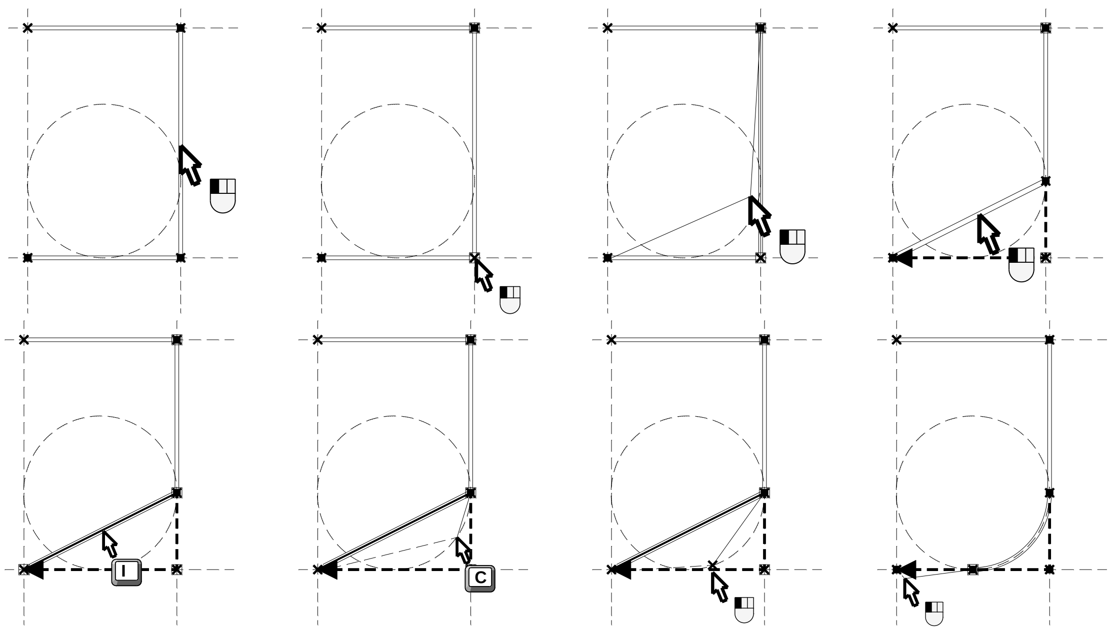
> The system returns to the mode “Contour selected for editing”. One can do other modifications, and then confirm all changes.

Confirm changes (the icon  or the <End> key).



Let us illustrate the above with a specific example. Suppose, a path contour is to be modified as shown on the diagrams.

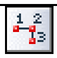
To get the result, begin with calling the command **EC: Edit Construction**. Then, using the option “Select Spline” (the icon , or the <S> key), select the path contour to be modified. Now, to get the result, perform the steps shown on the following diagrams.

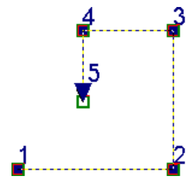


What is left is to press twice the  icon or <End> key, and the contour editing task is complete. Then, adjust the graphic lines accordingly, if necessary.

Displaying the Contour Point Numbers

To toggle the display of the contour point numbers of a 2D path, use the option:

	<Q>	Show/Hide Contour Point Numbers



With the option turned on, the points in the path are enumerated based on their position in the path and the path direction. A point number is displayed next to the respective node. When several subsequent points of a contour coincide, their numbers are displayed next to each other, separated by commas.

Creating Drawing Lines

Graphic Lines

Graphic lines are the base graphic elements that constitute the drawing image. The analogy for the graphic lines found in conventional drafting is the lines drawn in ink. Graphic lines are created based on construction lines and nodes. By “lines”, we mean straight or curved line entities described below.

The various types of graphic lines can be defined by:

A line segment between two nodes. The graphic line limits are defined by the location of these nodes.

A full construction entity. This graphic line can only be defined by the underlying construction entity. The construction entity can be of any type except straight line as the latter is infinite.

A portion of construction entity between two nodes. This type of graphic entity is defined by the underlying shape-defining construction entity and two nodes defining the line limits.

Graphic entities can be created with user-defined line types.

Creating Graphic Lines

Graphic lines are created in the command **G: Create Graphic Line.** Call the command via:

Icon	Ribbon
	Draw → Draw → Graphic Line
Keyboard	Textual Menu
<G>	Draw > Graphic Line

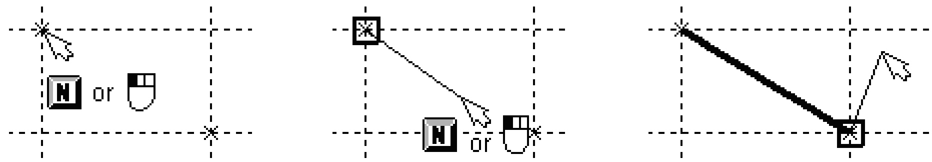
On entering the command, the following options will be available in the automenu:

,	<Ctrl><F>	Free mode on/off toggle
	<P>	Set Graphic line Parameters
	<Alt+P>	Copy Properties from Existing Element
	<Tab>	Change arc direction
	<N>	Select ending Node of Graphic line
	<L>	Select straight Construction Line
	<C>	Select Construction circle
	<E>	Select full Ellipse Contour
	<S>	Select full Spline or Polyline Contour
	<Q>	Creation of graphic lines between two intersections of construction lines

	<BkSpace>	Delete last Contour segment
	<F4>	Execute Edit Graphics command
	<Esc>	Cancel selection (available only when selecting a construction entity)
	<Esc>	Exit command

To create a **segment**:

1. **Select start node.** If the selected node is on the intersection of several lines, then use the <L> option for specifying the line to apply graphics at. If upon typing <L> key the construction line is not highlighted, the selected node may not belong to the line. This means, a wrong node was selected.
2. **Select end node.** This completes creation of the graphic line between two nodes.



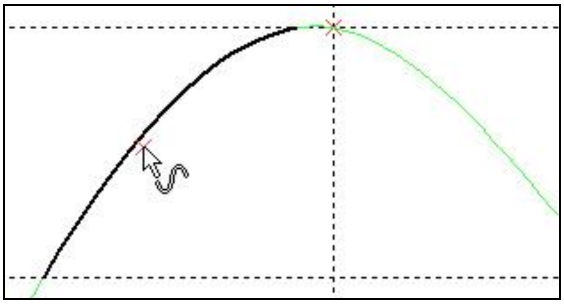
> After creating a segment, the end node stays highlighted and becomes the start node for the next graphic line. If you want to create a graphic line starting elsewhere, press <Esc> or for canceling the current node selection.

To define the start or end point of a graphic entity, simply place the cursor at the desired location and click . In "constrained" drawing mode, clicking selects the node at the nearest intersection of construction entities, if exists; otherwise, a node will be created at this intersection and then get selected. In "free" drawing mode, either a new node is created, or an existing one is selected. An existing node is selected if the graphic cursor is within the "finding" zone around the node. The size of this zone is defined by the parameter **Node join distance** in the command **Customize** > **Options…** > **Preferences.** The size is defined in pixels.

Automatic Creation of Graphic Line between Two Intersections of Construction Lines

In the **Drawing** > **View** command there is a mode of creation of graphic lines between two intersections of construction lines.

To use this option, it is required to press the button, move cursor to the construction line (line, circle, ellipse, spline, polyline) that is intersecting with other construction lines and press . Depending on location of the cursor the system finds the two nearest points of intersection of construction lines and creates the graphic line and nodes on its ends. When several lines are intersected at one point, the node is created between the lines that are created earlier than the others.



If, in this mode, by pressing the left button of the mouse we translate the cursor along the construction line, the construction line being created will automatically be continued (or shortened) in the desired direction.

Double clicking on the construction line will lead to creation of the graphic line on the entire construction line (except the straight line).

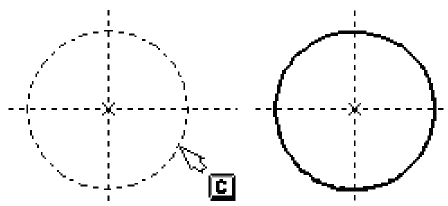
> If there is graphic line already created with the help of this option, you can’t create line of the same color on the same place again. If necessary, you can change graphic line color or hide previously created line.

The  option works similar to the click , however, unlike the latter, it only allows selection of existing nodes. New nodes won’t be created.

The options , ,  and  allow selecting construction entities of the respective types. The option behavior depends on the current state (whether there is already a pre-highlighted node or construction line).

The following is the way to create a **full curve**:

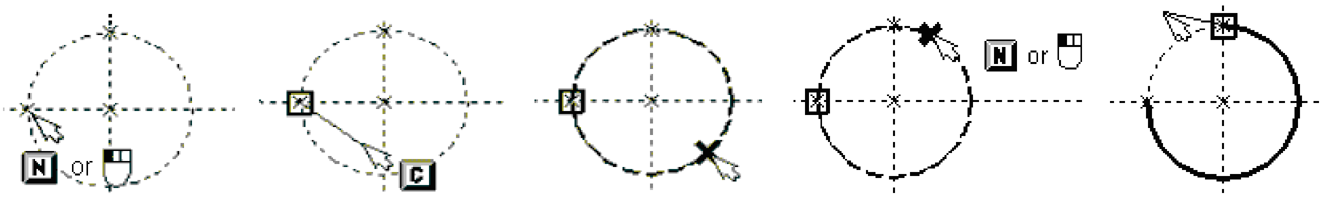
Select a respective construction entity by typing <C>, <E> or <S>, in a state when no node is selected.



To create a graphic line as a **segment of** the underlying **construction line**, do the following:

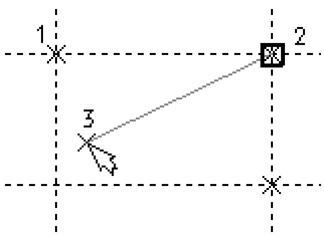
1. 1. Select the start node for the segment (arc).
2. 2. Select the intended construction entity. If a construction line does not get selected, that means the selected node does not belong to the line, and the arc can’t be created starting from this node.
3. 3. Select the end node of the segment (arc).

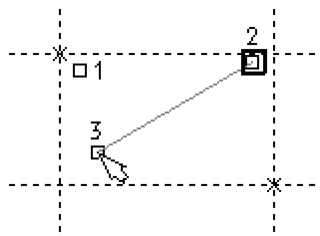
Then the arc will be created from the start to the end node. Please keep in mind that closed curves, such as a circle, are divided by two nodes into two arcs. The arc will be created nearest to the cursor at the time of selecting the end node.



Switching Drawing Mode. “Free” and “Constrained” Drawing Modes

The option / allows switching between the two modes of creating the construction elements – nodes. One of these is the “constrained” drawing mode (recommended), in which the nodes are created at intersections of construction entities only. The other is “free” drawing mode, in which the nodes are not related to other construction elements and their location is defined solely by the absolute coordinates of the drawing. The same drawing may contain both “free” and “constrained” nodes. Nodes may be created automatically while creating graphic entities. Therefore, it is important to know what drawing mode is currently set. The option icon in the automenu and the buttons on the “Modes” toolbar are not only for switching the mode. They also indicate the current mode. The  icon in the automenu indicates that the “constrained” drawing mode is active, while the icon  corresponds to the “free” drawing mode. Besides the icons, the mode can be determined by the appearance of the cursor and the nodes being created. If the cursor and the nodes are displayed as crosses, this means, the “constrained” drawing mode is on, while the “box” shape indicates the “free” drawing mode.





1 - constrained node;

2 - highlighted constrained node;

3 - cursor in the “constrained” drawing mode

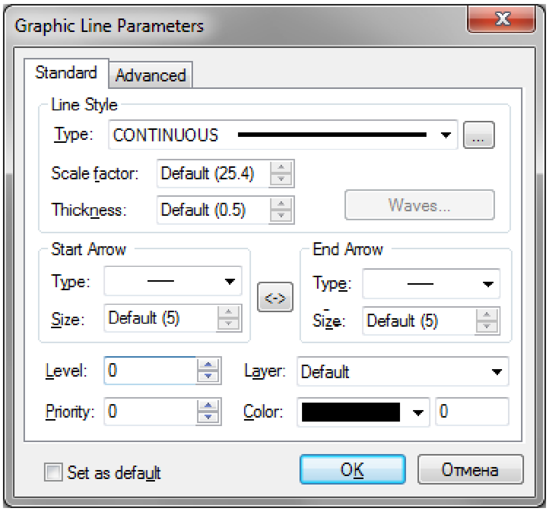
1 - free node;

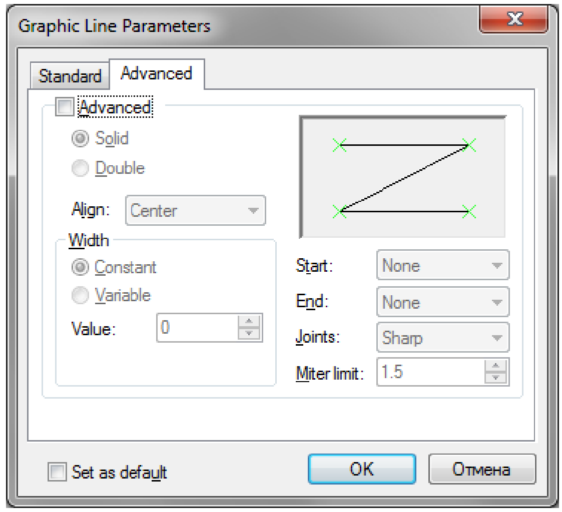
2 - highlighted free node;

3 - cursor in the “free” drawing mode.

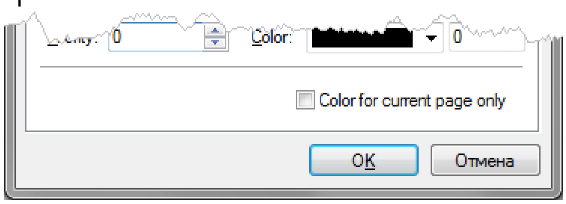
Graphic Line Parameters

The option calls the graphic line parameters dialog box.





When the dialog of parameters is invoked before creation of a line has begun, it will contain the additional parameter called **Color for the current page only**. This parameter is used only for drawing on pages of the workplanes (see the manual on three-dimensional modeling). When the flag is enabled, the specified color will not be used for all new lines in the document, but only for those lines that are created on the current page of the workplane.

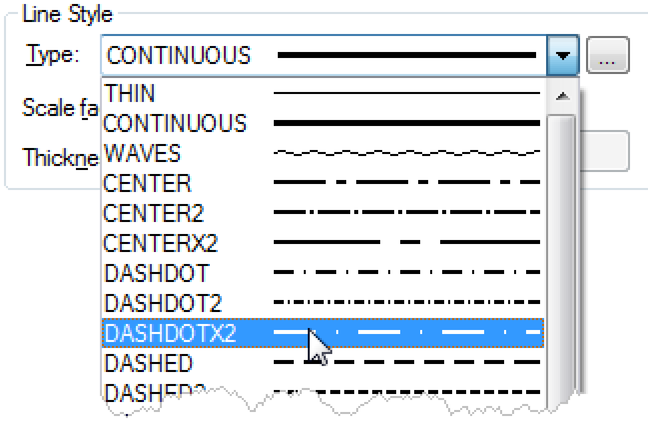


"Standard" Tab

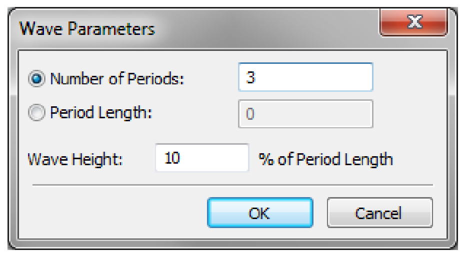
Line Style:

Type. Defines the line type for drawing graphic entities. The line type is selected from the list. The list includes both the standard (included in the system distribution) and the user-defined line types. Standard line types are defined in the file TCAD.LIN. Their description is compatible with that of AutoCAD system. The template files for user-defined types are located in the folder ...*Program\LinePatterns*.

Scale factor. Defines the Scale factor for dashed line types with respect to the dash size defined in the line type descriptor file (TCAD.LIN). Does not affect display of solid lines. If the scale is not defined (**Default**), then the scale factor will be taken from the **Scale factor for dashed lines** parameter of the **Lines** tab in the command **ST: Set Document Parameters**.

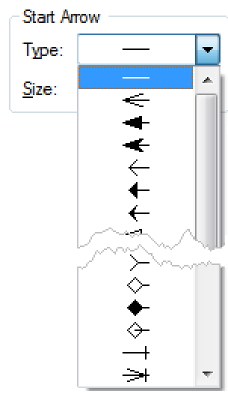


Thickness. Defines the thickness of the graphic entities. If undefined (**Default**), then the thickness for the solid thick line (CONTINUOUS) will be taken from the parameter **Line thickness > Thick lines**, while all the rest – from the parameter **Line thickness > Other lines** on the **Lines** tab in the command **ST: Set Document Parameters**.



When selecting the “Waves” line type, then, additionally, the **[Waves...]** button becomes available. This button allows setting waves line parameters: **Number of Periods** or **Period Length** of the line, and the line **Wave Height**. The Wave Height is entered as percentage of the period length.

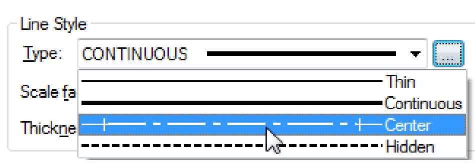
The line **start** and **end** parameters:



Start and end arrow type (arrow/symbol type). Each graphic line can have its start and end marked by a special symbol. The symbol type is selected from the list.

Start and end arrow size (size of start and end symbols). The sizes of the start and the end symbol are defined independently. Any size can be set at the user’s preference.
If the symbol size is not defined then the symbol is drawn proportional to the font size defined for the drawing on the **Font** tab in the command **ST: Set Document Parameters**.

An additional button [<->] serves to quickly swap the line start and end parameters.



All above parameters are set by default when using one of the standard system line type. The style and ending type are defined for each standard type. A standard line type can be selected from the list coming under the [...] button.

The description of the standard line types is stored in the file SPECLINE.DEF. By default, the file contains the following entries:

```
Thin THIN 0 0
Continuous CONTINUOUS 0 0
Center CENTER 28 28
Hidden HIDDEN 0 0
```

The file can be appended by the user as desired. The first parameter is a comment, the second is the line name (this name is used for identifying the line in the line descriptor file), the third and the fourth parameters are the Ids of the start and end symbols (per the enumeration in the endings list).

Color. The graphic line color.

Level. The value of the visibility level of the graphic line.

Priority. The value of the graphic line priority.

Layer. The name of the layer of the graphic line.

Some parameters of graphic lines can be entered in the system toolbar. Specifically for graphic lines, the system toolbar provides a button for defining graphic line types, and the buttons for defining the graphic line start and end points. The thickness of the line being created can also be specified with the help of the system toolbar.

"Advanced" Tab

The "Advanced" tab allows setting the following parameters:

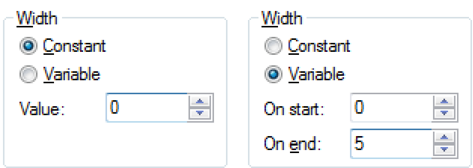
The line display mode:

Solid line. The lines whose "Width" parameter is not 0, will be drawn filled with color.

Double line. The lines whose "Width" parameter is not 0, will be drawn as a contour without filling.

Align. Defines centering of the graphic line with respect to the reference nodes: **Center**, **Left**, **Right**.

Width. Defines the width of the graphic line: **constant** or **variable**.



In the case of the constant width, the parameter **Value** defines the line width.

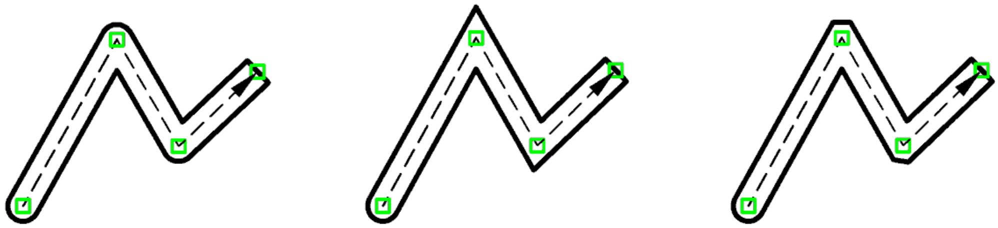
If variable width is set then the line width values **On start** and **On end** need to be defined.

The **Start** and **End** parameters define the shape of the start and the end of the line, as follows: **None**, **Round**, **Square**.

Joints. This parameter defines the shape of joints of a graphic line created from a 2D path, as follows: **Round**, **Sharp**, **Mitered**.

The size of the rounds and miters depends on the line width setting. If "Round" attribute is set, the segment joints will be rounded with the radius equal to the line half-width. With the

"Mitered" setting, the corners at the segment joints are mitered. The distance from the joint node to the miter top is defined by the parameter **Miter limit** as a ratio of the line half-width.



Copying Parameters from Existing Lines

The values of parameters of the graphic line being created can be quickly copied from already existing line. To do so, use the following option:

	<Alt+P>	Copy Properties from Existing Element

> This option can be accessed in the automenu of the command before creation of the line or in the process of creation.

After this option is invoked, it is sufficient to specify the graphic line whose values of parameters must be transferred to the line being created.

To assign the copied values of parameters to all new lines, before selection of the original graphic line it is required to enable the additional option:

	<S>	Set Properties as Default

When this option is enabled, all copied parameters will be stored as default parameters.

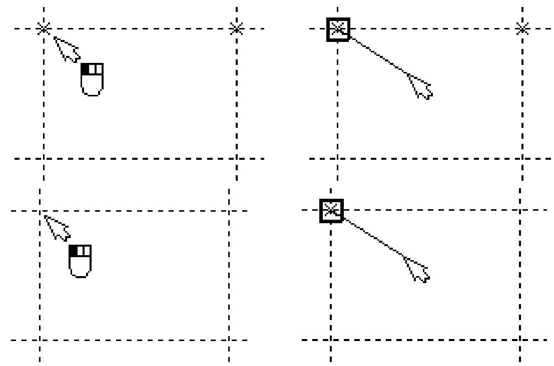
> The given option simplifies creation of the graphic lines having the same parameters. However, it does not allow us to copy individual parameters or parameters from an object of another type. In such cases it is more convenient to use the general mechanism of editing parameters of elements in the window of properties.

Using Construction Entity Selection Options

Next, we will describe uses of the listed options in various situations.

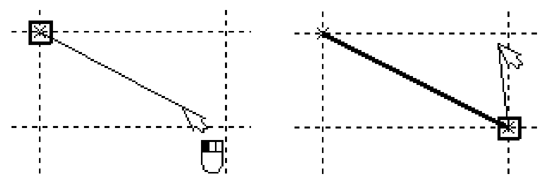
Clicking in the situations described in the headings, accounts for the following actions:

1. Neither nodes nor construction lines are selected



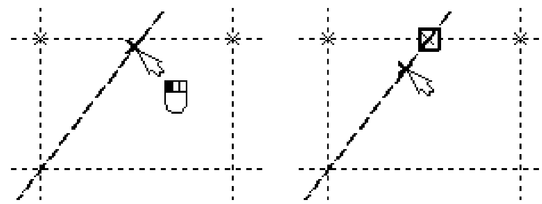
The action selects the nearest node. Line rubberbanding begins from the highlighted node. This option is sensitive to the drawing mode. In the “constrained” drawing mode, the nearest node is selected, or created in the nearest construction lines intersection (whichever is closer). In the “free” drawing mode the nearest node would be selected only within the zone around the cursor defined by “node join distance”. If such node exists then it will be selected. Otherwise, a new “free” node will be created.

2. The start node alone is selected



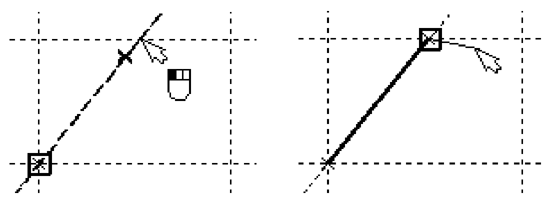
The action creates a graphic entity – straight line segment from the start node to the node defined by this step. The end node is then highlighted, and new rubberbanding begins from it.

3. A single straight construction line is selected



A node will be selected with the action, located at the nearest intersection of the highlighted line and some other construction entity (either a line or a circle). The line will stay highlighted, and a new graphic line will begin rubberbanding after the cursor from the highlighted node as the start.

4. The start node and a construction entity are selected



The action creates a graphic line entity – a straight line segment or a curve from the start node to the node defined by this step. The type of the created graphic entity depends on the selection of the underlying construction entity – line, circle, spline, or ellipse. On completing the step, the last selected node stays highlighted, with rubberbanding resuming from there.

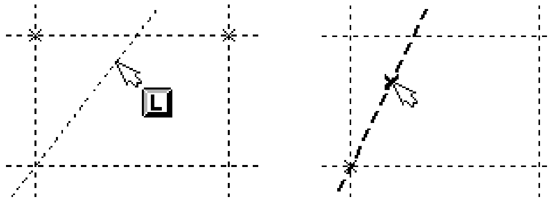
The selected node must lie on the selected construction line.

The option  in all the above situations acts in the same way as the action . The only difference is, the option <N> can only select existing nodes.

The option  acts as follows in the described situations:

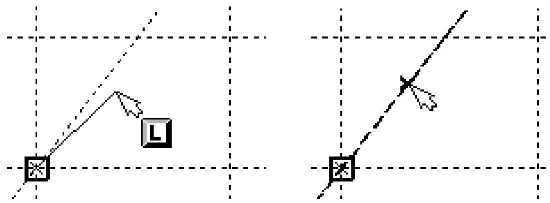
1. No nodes neither construction lines are selected

The option <L> highlights (selects) the nearest straight construction line. Highlighting the construction line in this case means the start node of the graphic line being created will be constrained to this line. A node will begin rubberbanding, constrained to sliding along the selected line. This indicates that a node can be selected only at an intersection of this line and some other construction entity.



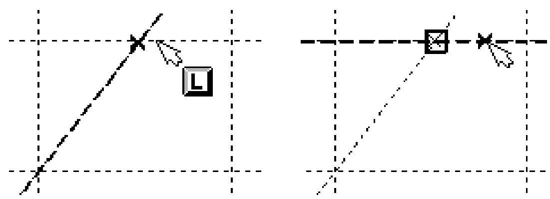
2. The start node alone is selected

The option <L> highlights a straight construction line. The start node stays highlighted, and a new line begins rubberbanding, constrained to stretching along the selected line. This indicates that the end node of the new graphic line can be selected only at an intersection of the highlighted line and some other construction entity.



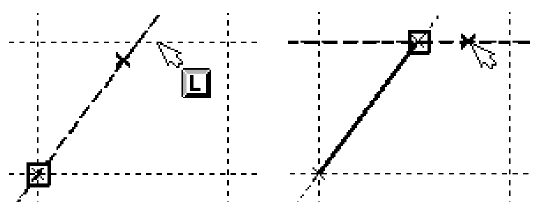
3. A single construction entity is selected

The option <L> selects the node at the intersection of the already highlighted construction entity and the newly appointed construction line. The selected node becomes the start node for the graphic line being created. The new line begins rubberbanding, constrained to the selected construction entity. If the two selected construction lines do not intersects, no action occurs.



4. The start node and a construction entity are selected

The option <L> selects the node at the intersection of the already highlighted construction entity and the newly appointed construction line. A graphic entity is created from the start node to the newly selected node,



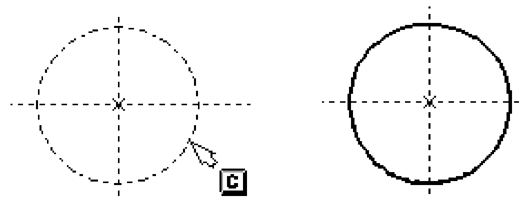
which may be a straight line segment or an arc of a circle, depending on the selected (underlying) construction entity. The newly created node and the last selected construction entity stay highlighted, and new line rubberbanding begins along the latter entity.

If the two selected construction lines do not intersect, no action occurs.

The option  is insensitive to the "free" versus "constrained" drawing mode. In either case, its use implies selection of a construction circle entity as the underlying entity for creation of a graphic entity – circle or arc. The option acts as follows in the described situations:

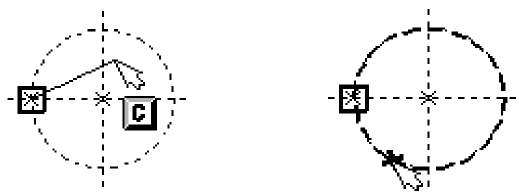
1. **No nodes neither construction lines are selected**

The option <C> creates a graphic entity – circle over the selected underlying construction circle entity.



2. **The start node alone is selected**

The option <C> selects a construction circle entity. The start node stays highlighted, and an arc starts rubberbanding along the selected circle. An additional option appears in the automenu for flipping the direction of the arc creation:



	<Tab>	Change arc direction

3. **A single construction entity is selected**

The effects of using the <C> option in this case are quite similar to the use of the option <L>.

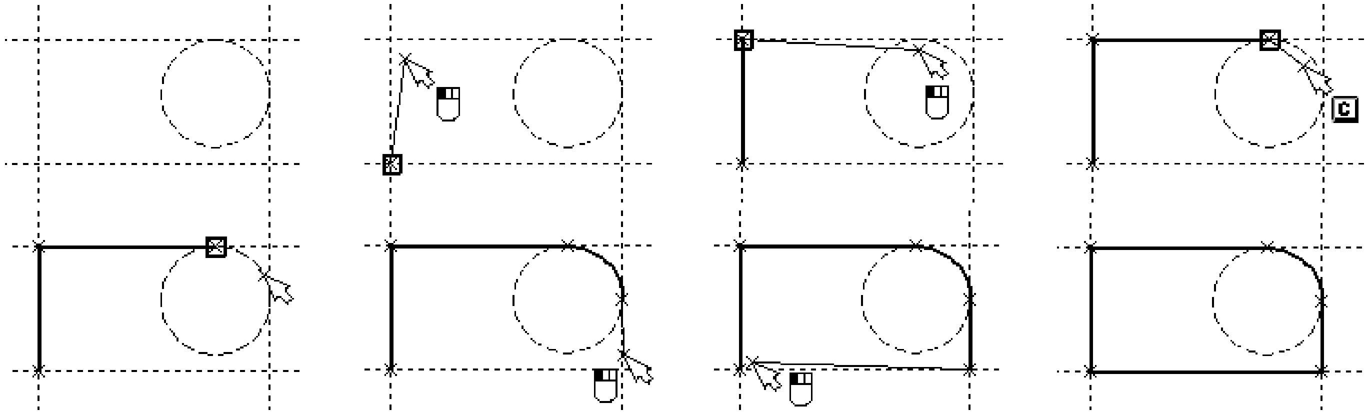
4. **The start node and a construction entity are selected (line or circle)**

The effects of using the <C> option in this case are quite similar to the use of the option <L>.

The options  **and**  are used similarly to the option .

Example of Creating a Chain of Graphic Lines

Create a few construction lines. On top of them, draw the graphic lines:



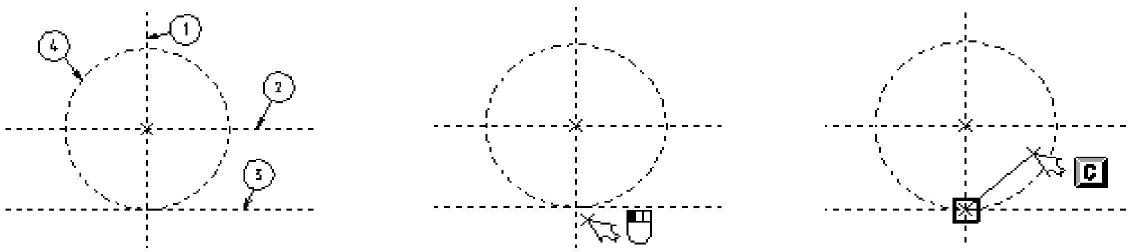
To delete the last created graphic line, one can use the option .

A Few Tips on Creating Graphic Lines

It is not recommended to use the  action for defining the start node of a graphic entity where there are more than two intersecting construction lines. This is because the node may get created at unintended lines intersection.

In the special case of two or more coinciding nodes, use an appropriate option for selecting construction entities (<L>,<C>,<E>,<S>). First, specify the construction entity to underlie the graphic one, and then either select the node, using the <N> option, or specify the intersecting entity, depending on the situation.

It is a good idea to initially create a node at the intersection of construction lines, using the command **N: Construct Node**. In this way, you can precisely specify what lines intersection to use for the node creation. Later, when applying the graphic line, this node can surely be selected using the <N> option. Consider a simple example illustrating the approach. There are several construction lines on the drawing below. **Line 1** is constructed vertical, parallel to the Y axis, **line 2** is a horizontal, parallel to the X axis, **line 3** is constructed parallel to **line 2**, and, finally, **circle 4** is constructed with the center snapped to a node, tangent to **line 3**. Now, try to apply a graphic arc using the <Enter> option. Note that the created node will not necessarily snap to the intersection of the line and the circle. The node may actually be, for instance, at the intersection of **lines 1** and **3**. In this case, the circle can't be selected for creating an arc.



A properly built model helps avoid annoying errors. If a node is supposed to be on the intersection of the circle and the line, then it ought to be constructed as such.

Basic Rules of Graphic Line Creation

The user is encouraged to follow a few rules when creating graphic line entities:

- To avoid errors when creating a parametric drawing, use the option <N> for applying graphic lines. Do not use the <Enter> option if there are more than two construction entities intersecting in one point.
- If there is a selected node, then it will be used as the start for a graphic line.
- If a node is selected and user attempts to select a construction line entity, it has to be a line passing through the selected node.
- If a node and a construction line are selected, and the user selects another construction line, then a graphic line will be created, starting in the selected node and ending at the intersection of the selected lines.
- If two construction entities intersect in more than one point (for example, a line and a circle), then the nearest intersection is selected to the graphic cursor location at the time of pressing the option key.
- If selecting a construction line entity results in nothing, that means, the lines do not intersect and the graphic line can't be created.

Using Grid in "Free" Drawing Mode

If snapping to grid is turned on, then the start and end nodes of the graphic line will be snapping to nearest grid knots. The grid parameters can be defined using the command "**Customize|Grid…**". The grid can be assigned different steps in the vertical and the horizontal directions, and different shifts with respect to the origin in each direction. When creating a graphic line, the status bar displays the coordinates of the nearest grid knot to the current mouse cursor location. In the "free" drawing mode with grid snapping turned off, a graphic line can be created at an arbitrary location in the drawing area. It does not require constraining to any construction lines.

Editing Graphic Lines

Editing graphic lines is done by the command **EG: Edit Graphic Line**. Call the command via:

Keyboard	Textual Menu	Icon
<EG>	Edit > Draw > Graphics	

The following options become available upon calling the command:

	<*>	Select All Elements
	<R>	Select element from list
	<Esc>	Exit command

When in the command, one can select a graphic line entity by pointing the cursor and clicking . Several graphic lines can be simultaneously selected by box. The graphic lines will be selected that are completely within the box.

All graphic lines can be selected at once by typing <*>. To add a graphic line to the set of already selected ones, use the combination <Shift> +. To exclude a graphic line from the selected set, use <Ctrl>+.

The following options become available after selecting one or several graphic lines:

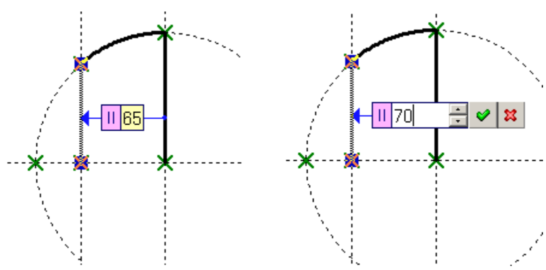
	<P>	Set selected Element(s) Parameters
	<Alt+P>	Copy Properties from Existing Element
	<I>	Select Other Element
	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

If only one graphic line is selected, then the following option is available:

	<O>	Create Name for selected Element

If a graphic arc entity is selected, the following additional options become available in the automenu:

	<Tab>	Change arc direction
	<A>	Link Arc or Circle to Node
	<B>	Break Link with Node

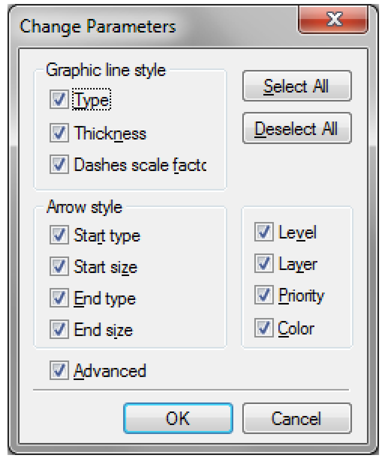


If the selected graphic line is created based on a construction line, then Relations for the parent construction line will appear in the 2D window. Those relations are *temporary*, meaning that those are created by the system automatically upon entering the mode of editing a graphic line, and are automatically deleted upon exiting the mode. Using Relations, you can modify geometrical parameters of the parent construction line in the transparent mode.

Besides that, if the selected graphic line is created based on a construction line, then the second click after selecting the line (while the cursor is pointing at the line) will invoke the command of editing the original construction line.

To modify parameters of the selected graphic line, use the option . The initial parameters are taken from the last selected graphic line.

If the user needs to bring the parameters of a graphic line in accord with a given line parameters, then the latter should be selected the last before proceeding with the modifications. This will automatically pre-set the desired parameters in the dialog.



If more than one graphic line is selected then another dialog box comes on the screen before the parameters dialog box. Its purpose is to define what parameters of the selected graphic lines are to be modified.

After that, the graphic line parameters dialog box will appear. Now, only the parameters will be modifiable that were specified in the previous dialog box. For instance, if in the previous dialog only the line type was checkmarked for modification, then only this parameter can be modified. Other parameter modifications will be discarded.

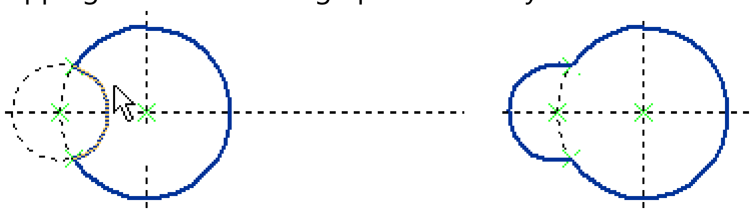
Object properties can be also set from another object (including image line) in command waiting mode with the help of the Properties window. See more details in “Main Concepts of System Operation” chapter.

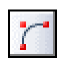
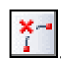
The current set of graphic line parameters defined during the editing can be saved. New graphic line creation would then be based on this particular set of parameters.

To open the parameters dialog box for a single graphic line, one can simply double-click it ().

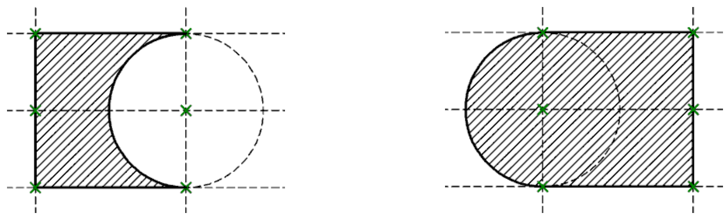
The option  is used for assigning a name to the selected graphic line. The name is unique and provides unambiguous identification to the line. The graphic line name can be used instead of its Id number. For example, the function *GET()* can be used in the variable editor to query a graphic line, named *NAME*, as follows: *GET(“NAME”, “LENGTH”)*.

The option  allows flipping the direction of graphic arc entity creation.

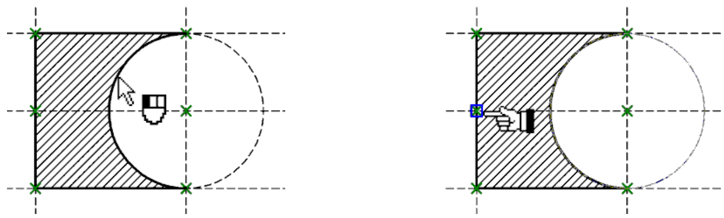


The following options are available in the automenu for graphic arc entities,  and . These options manage a locking node of graphic arcs constructed on top of a construction entity. Throughout modifications of the drawing, the graphic arc will stay over the sector of the underlying construction circle that is closer to the locking node.

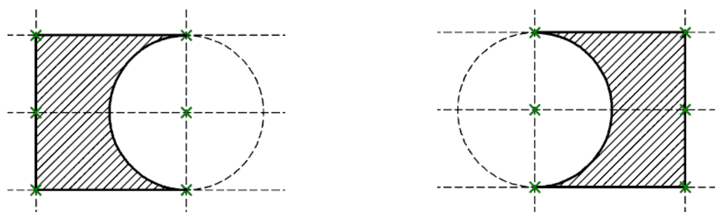
Let’s review the following example. When moving one of the original lines, the part is supposed to get mirrored. However, the graphic arc will stay in its original orientation as the vertical line is moved, resulting in the wrong final shape of the part. This can be fixed by using a locking node.



To keep the arc always in the correct sector of the circle, link it to a node. After calling the command **EG: Edit Graphic Line** select the arc and use the option . The cursor will change to "finger" . Then, select the locking node using .



Now, as the vertical line is moved, the whole drawing will be flipping, maintaining the original relative configuration.

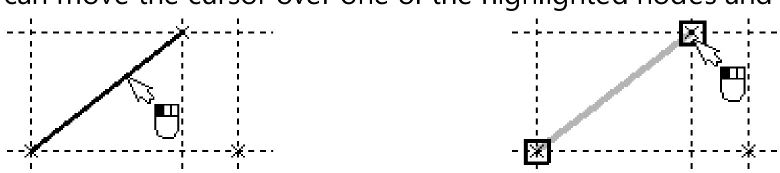


To release the link with the node, use the option .

To cancel the last graphic line selection and select the next nearest graphic line to the current cursor position, use the option . This option is convenient when there are several closely located or overlapping graphic lines, and the first selection attempt yielded the wrong graphic line.

The option deletes all selected graphic lines. The option cancels the current selection of graphic lines.

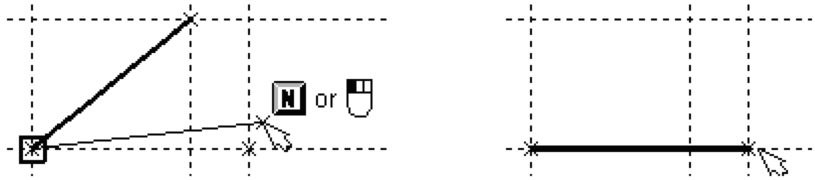
Besides, a capability is provided for reassigning the start and end nodes of the graphic line. Reassigning the nodes is possible for a single selected graphic line. Once the line is selected, the nodes get highlighted. Now, you can move the cursor over one of the highlighted nodes and click it .



The line will then start rubberbanding after the cursor, with another option becoming available:

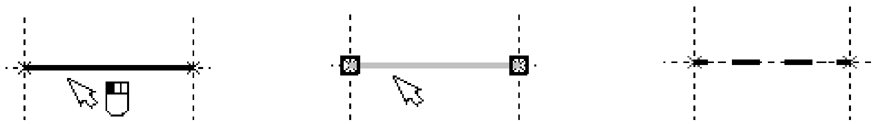
	<N>	Select existing Node as the start or end of the graphic line

You can select a node for the graphic line to snap to.

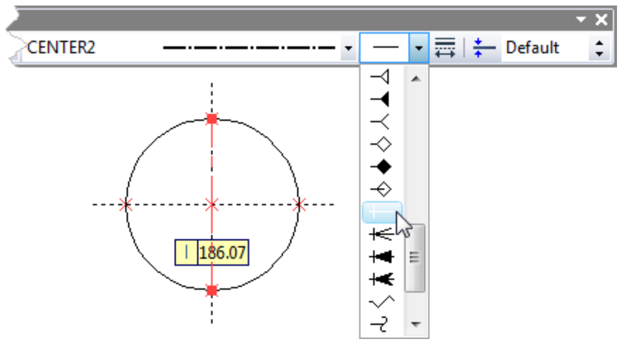


To change parameters of the selected graphic line, the system toolbar can also be conveniently used. With the help of the system toolbar it is possible to change color, type, thickness and ends of the line.

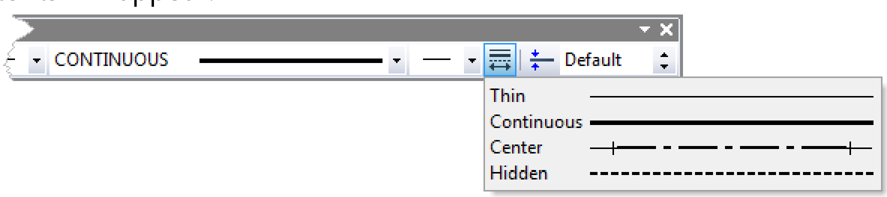
For example, for changing the type of the line with the help of the system toolbar – first select the graphic line. Move the cursor to the line type input box on the system toolbar and click. A menu of line types will appear on the screen. Select a new line type by. As the result, the appearance of the graphic line will change.



The graphic line endings can also be modified in a similar way. Use of endings is important, for instance, when applying centerlines. These are drawn in a dash-dotted style. To create centerlines with hanging extents, you would not need to create additional nodes beyond the circle (as shown on the diagram). Rather, select the start and end types of the graphic line as shown on the diagram. The size of the extents can be defined explicitly in the parameters of the graphic line, or left as *"Default"*. In the latter case, the size will be defined by the parameters in the command **ST: Set document parameters** the tab **Font**, **Size** parameter.



The same can be done easier. While in the command **G: Create Graphic Line** or **EG: Edit Graphic Line**, press the graphic button on the right-hand side of the system toolbar. The list of the most often used lines with extents will appear.



Sketch. Creating a Non-parametric Drawing. Automatic parameterization mode

T-FLEX CAD allows creating a drawing similar to most well-known CAD systems, using standard functionalities for creating various primitives, such as: line segments, arcs, circles, ellipses, splines. Sketcher functionalities, including object snapping and dynamic tooltips, significantly simplify and speed up the process of creating a non-parametric drawing. Such drawings do not share the advantage of parametric drawings in the effective usage of modifiable parameters (dimensions). However, in certain cases, development of such drawings is faster and can bring certain benefits, when large modifications are not expected.

Quick creation of graphic lines in a drawing is done with the “Sketch” command. This command can work in two modes: in the sketching mode and in the automatic parameterization mode.

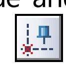
Creating Lines in a Drawing

To quickly create graphic lines use the command **SK: Create Sketch**:

Icon	Ribbon
	Draw → Sketch
Keyboard	Textual Menu
<SK>	Draw > Sketch

Upon calling the command, the options appear in the automenu that allow creating various lines in a drawing. You can use all object snaps available in the system to make your constructions.

Two working modes of the command **“SK: Create Sketch”**

The **SK: Create Sketch** command can work in one of the two modes: in the sketching mode and in the automatic parameterization mode. Switching between the modes is done with the icon  on the “View” panel. When the icon is switched off, a plain sketch is created. When the icon is On, the automatic parameterization mode is at work.

In the sketching mode, only the graphic lines based on free nodes are created. To create graphic lines, the user selects the desired type (a line segment, an arc, circle, etc.) and defines the position of the line defining points using or by entering coordinates/parameters in the command's properties window. When specifying the positions of the defining points of the lines being created, one can use object snaps to existing drawing elements (a vertical/horizontal relation, tangency, perpendicularity, etc.). As a result, you get a nonparametric drawing without construction lines (a “sketch”).

In the automatic parameterization mode the user also builds up a drawing as a sketch, using all available command tools. Instead of the free nodes, the system automatically creates construction elements beneath the graphic lines, that are tied by parametric relations.

The types of relations introduced by the system depend on:

- the options of the command that was used to create a graphic line;
- the object snaps used when creating the line;
- parameters defined in the command's properties window.

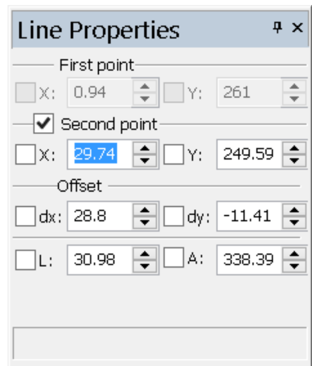
As a result, a fully functional parametric drawings is built. At the same time, you need to note that the system automatically defines the types of parametric dependences created between elements, which may not always meet the user's preferences.

Details on working in the automatic parameterization mode are provided at the end of this chapter, in the section “Working in the Automatic Parameterization Mode”.

Using Property Window

When sketching lines, the point coordinates can be simply defined by clicking in the drawing area. To enter exact node coordinates, the property window is used in this command. It allows defining absolute, relative, or polar coordinates of the elements being created and their parameters.

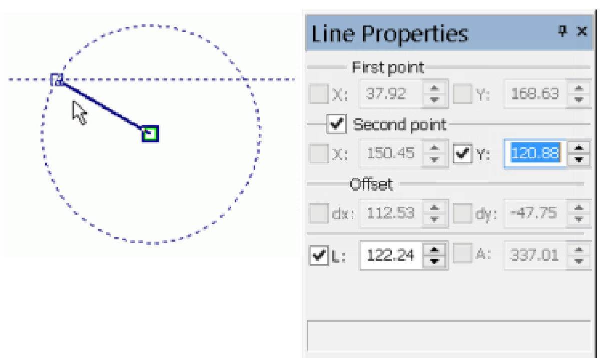
When the pointer is in the drawing area, the property window tracks the current coordinates of the pointer. If necessary, those can be changed in transparent mode by typing the desired value directly from the keyboard. The active input box can be set by pointing and clicking , or from the keyboard. The key combinations for switching to one or another input box are displayed in the ToolTips as the pointer is rested over the desired field. When entering a value in the property window, a flag before the input box is automatically set that blocks modifications of the value via the pointer in the drawing area.



To complete the point creation, simply press **[Enter]** or in the drawing area after entering the coordinates.

Either Cartesian or polar coordinates can be used for creating elements, separately or in combination. That helps creating various configurations of points in the most convenient way for the moment.

For example, when specifying the second point of the segment, one can enter the value of the distance and the length of the vector. Auxiliary elements will be displayed in the drawing: a circle with the center at the segment start, of the radius equal to the specified distance, and a horizontal line offset from the segment start at the distance equal to the Y shift (“*dy*”). The intersection points of the circle and the line define the possible configurations for the second point of the segment under the specified parameters.



As the pointer moves around the drawing, this point will appear as a free node jumping from one intersection point to the other and back. Selecting the desired point and clicking completes the segment creation.

Continuous Line Input

Continuous input of sketch elements is supported by the automenu option:

	<J>	Continuous creation

In this case, the end point of the last created element (segment, arc) becomes the start point of the next one.

This mode does not affect closed elements (circle, rectangle, polygon, ellipse, closed spline).

Using Offset from Node

When creating sketch elements, the position of any point of the elements being created can be defined relative to another point. To do this, use the automenu option

	<Z>	Offset

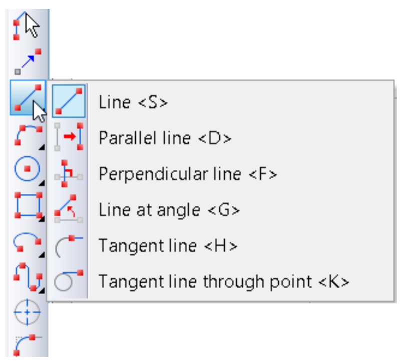
This option can be called in transparent mode at any stage of creating sketch elements. Upon selecting the option, select the point to offset from. Then, the offset distance is defined in relative or polar coordinates, using the pointer and/or the property window. The system then returns to element creation.

Creating Line Segments

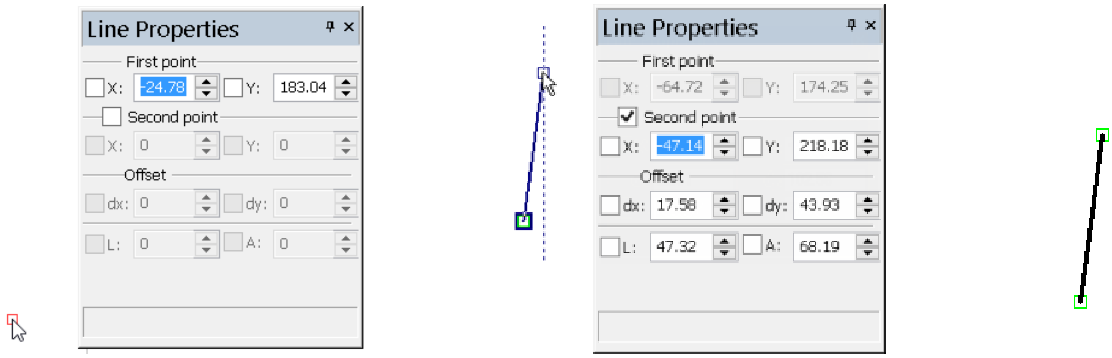
Any time the command is called, the system is ready for inputting line segments, as indicated by the pushed icon in the automenu:

	<S>	Line

A black triangle in the lower right corner of the icon indicates presence of a pull-down list of options. Holding the button depressed over the icon a bit longer opens the menu with more options.



To create a simple line segment (the option), one needs to define two points. The points can be specified arbitrarily by pointing in the drawing area and clicking or by entering the exact coordinates (offsets) in the property window. One can specify point coordinates relative to the selected point or node in the drawing using the option <Z> ().



When creating a line, existing nodes can be selected as the line ends.

Parallel line

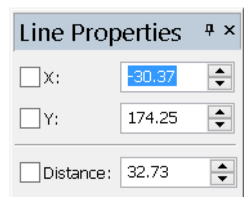
To create a parallel line, select the option:

	<D>	Parallel Line

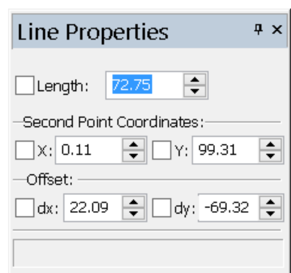
In this case, select a line by the mouse, to be used as the reference for constructing a parallel line. A line will start rubberbanding on the screen parallel to the selected line. If the continuous input mode is used, then this line will be parallel to the last input segment. To cancel a segment selection, right click.

After selecting the reference line segment, specify the distance to the segment being created, the start point and the length of new segment. This can be done freely by moving the pointer and clicking at the desired positions in the drawing, or input exactly in the property window.

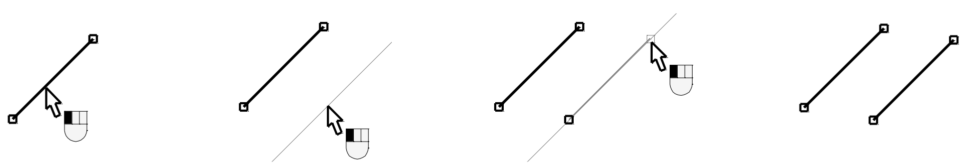
The property window allows entering coordinates of the start point of the parallel line. This also defines the distance between the lines. A node will be created in the specified point. From now on, the rubberbanded line will pass through this node.



The line end can be input arbitrarily by moving the pointer along the line and clicking, or by exact value in the properties window, entering either line length or second point coordinates.

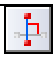


The distance between the lines can also be entered in the property window. In this case, the rubberbanded line will be fixed at the specified distance from the original line segment. A node will rubberband along the line, following the cursor, defining the start of the new segment. Its position is defined by clicking. After that it is necessary to set the line length or its second point coordinates.

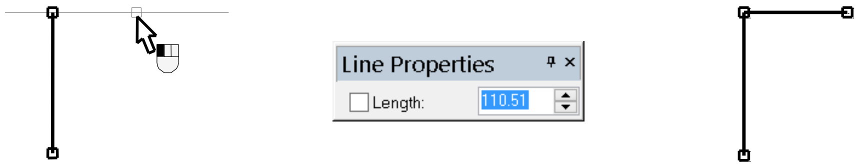


Perpendicular line segment

A perpendicular line segment can be created using the option:

	<F>	Perpendicular Line

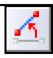
In the continuous input mode, upon this option selection, an infinite line will be displayed that is perpendicular to the last input line segment and passing through its end node. The latter node will be used as the start of the segment being created. In this case, use the mouse pointer for fixing the rubberbanded node position by clicking or entering a value in the property window.



To create a line perpendicular to another segment or to the same segment that is not passing through the end node, the current system selection can be rejected by right clicking while in the continuous input mode. One right click cancels the selection of the start point of the segment being created. The second right click cancels the selection of the reference segment, allowing manual selection of the desired segment. The further construction steps in this case are similar to creation of a parallel line.

Slanted line

To create a line segment at the specified angle to another segment, use the automenu option:

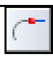
	<G>	Line at angle

This kind of line is constructed similar to the perpendicular type, except that the angle can be entered in the property window.

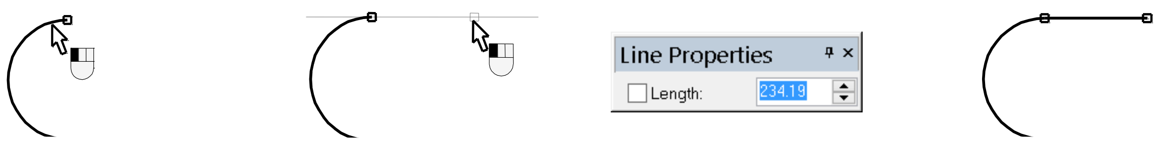
The icon  in the property window helps quickly change the specified angle for the complementary one based on 1800.

Line tangent to an arc and passing through arc end

This configuration is supported by the automenu option:

	<H>	Tangent Line

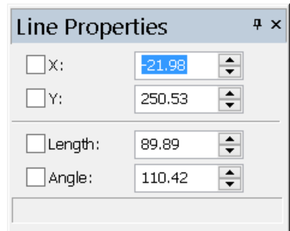
First, select the tangency arc. An auxiliary line will be displayed in the screen, tangent to this arc. The line will be snapped to the end of the arc nearest to the pointer at the time of arc selection. Move the pointer with the rubberbanded node along the line and fix the second node position by clicking or by input into property window.



Line tangent to a circle or arc

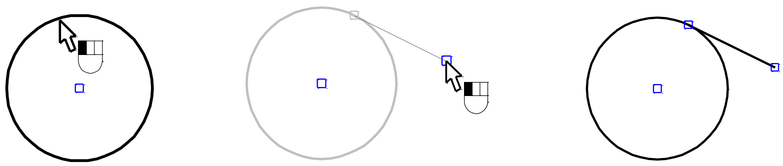
	<K>	Tangent Line through point

With this option, select a circle or arc (other elements are disallowed from selection), to which the tangency line is to be constructed. The selected element will be highlighted, and the arc will be extended by an auxiliary circle. An auxiliary node will be rubberbanding along the circle. This will be the first node of the line segment, defining the tangency point between the line and the circle. The line will be rubberbanding between the node and the pointer. To complete the tangent line, fix the position of the second node outside the circle.



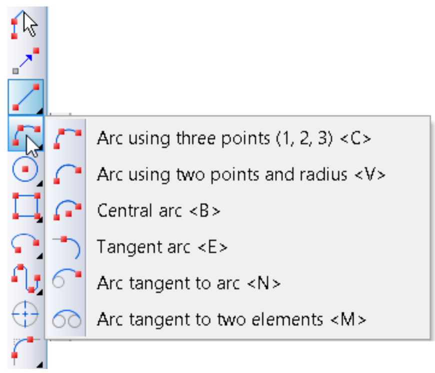
The position of the second node can be defined arbitrarily by clicking (use object snapping as appropriate), or select an existing node. Besides, one can use the property window by specifying coordinates of the node or the length of the line or else the angle to the horizontal of the radius pointing to the tangency point between the line and arc.

The position of the first line node will be defined automatically.



Constructing Arcs

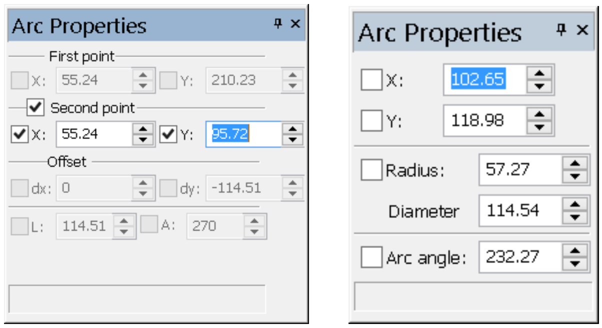
As is the case of line segments, the set of icons for creating various types of arcs is in the pull down menu. Any of the enclosed icons can be displayed at the top level of the automenu when sketching. Usually, it is the icon corresponding to the last used option in this command.



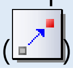
Arc by three points

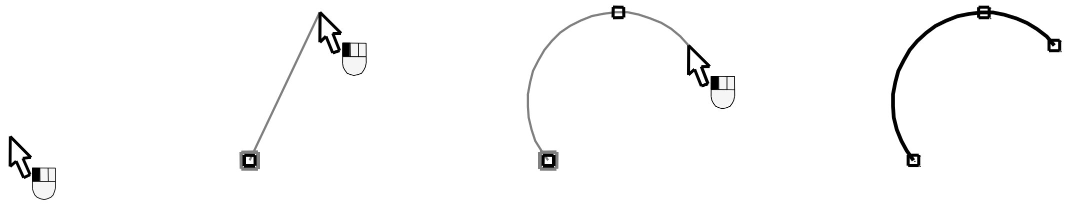
	<C>	Arc by three points

This option defines the mode of creating an arc by three points. The first and third points are the end points of the arc. The second point defines the arc position.



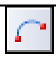
The arc can be arbitrarily input by selecting three points with the mouse, or specified exactly in the property window. In the latter case, the second point is defined in absolute coordinates or by the offsets with respect to the first point of the arc. To define the third point, one can use absolute coordinates, the radius, the diameter or the angular length of the arc in various combinations.

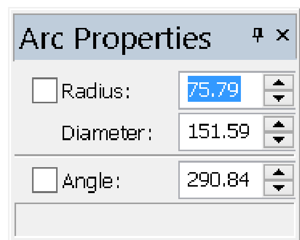
> The coordinates of any point of the arc can be specified by offsets with respect to a selected point or node in the drawing with the option <Z> ().



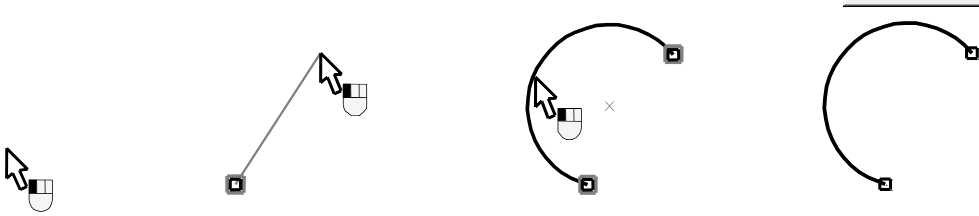
Arc by two points and radius

To create and arc by two nodes, turn on the automenu option:

	<V>	Arc by two points and radius

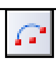


Use the mouse or the property window to define the two end points of the arc being created. After that, the arc will start rubberbanding on the screen following the pointer. To fix the arc, move the pointer to the desired position and click, or enter the value of the angle or the arc radius (diameter) into the property window.

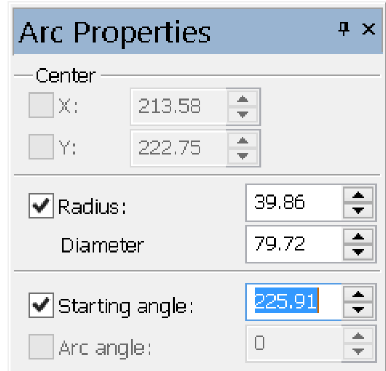


Arc by center and ends

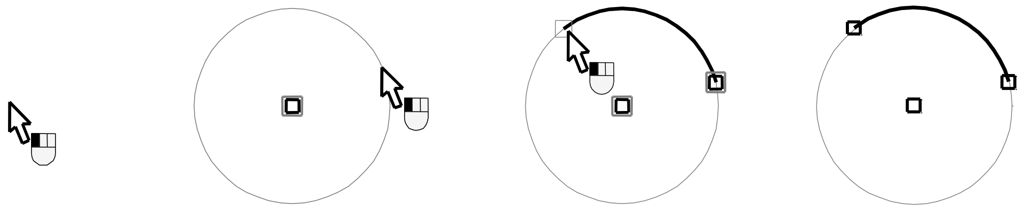
Creation of an arc with the specified center is done in the mode provided by the option:

	<B>	Central Arc

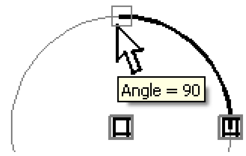
To construct a central arc, specify the center and radius (diameter), as well as the start and end angles of the arc. This can be done freely by using , or exactly in the property window. After defining the center, a circle starts rubberbanding on the screen. The input boxes become accessible in the property window for entering the radius (diameter) and the start angle of the arc. Those can be defined by moving the pointer to the desired position and clicking . Next, move the pointer along the fixed circle in the desired direction of the arc, and once more click .



To quickly flip the arc direction, one can use the <Tab> key. Besides, the arc angle can be defined in the property window after fixing the input radius and the start angle of the arc with the **[Enter]** key or clicking in the drawing area.

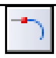


At the third step of the arc construction, the system offers snappings to typical angles of the arc.

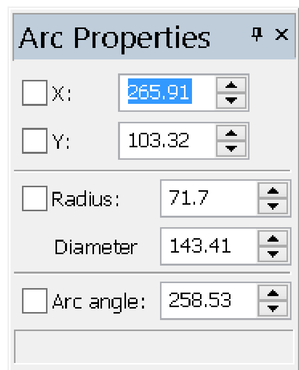


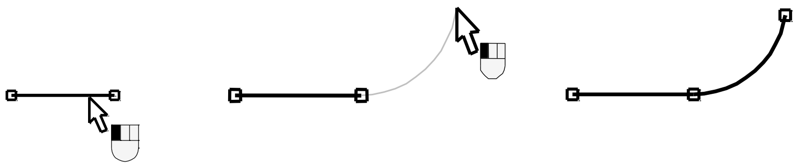
Tangent arc

To create an arc, tangent to a graphic entity (an arc or a line segment), start the respective mode with the icon:

	<E>	Tangent Arc

After that, select a graphic line. The arc will originate in the end node of the selected line nearest to the pointer position at the time of selection. In the continuous input mode, the arc originates in the end node of the last created element. The rubberbanding arc can be moved by the mouse to the desired position and fixed. The position of such arc can be specified in exact values. To do this, select a graphic line, and then specify the end point coordinates of the arc in the property window. Alternatively, one can specify the radius (diameter) and the arc angle. In the latter case, first use the mouse to specify the direction of the arc.

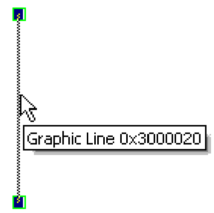




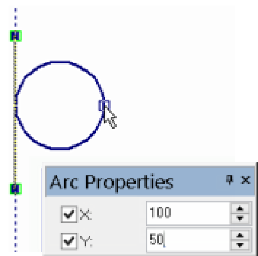
Arc tangent to another arc

	<N>	Arc tangent to arc

This automenu option allows selecting an entity (a line segment, circle or arc), to which the arc being created should be made tangent. The selected element will be highlighted and projected (extended) up to the full line or circle. A rubberbanding circle will appear on the screen, tangent to the selected element. The position and the size of the circle change as the pointer moves.

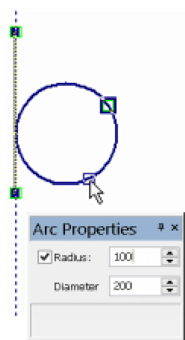


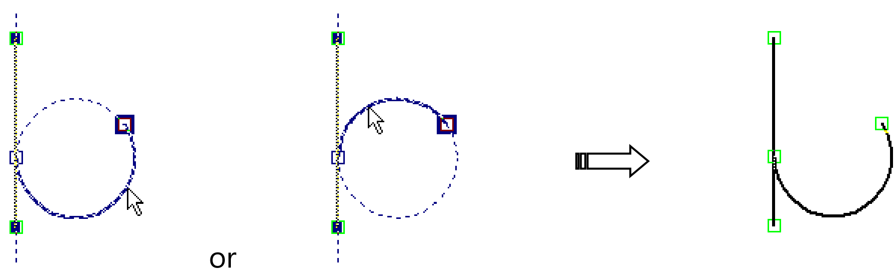
The second step is constructing the end node away from the tangency entity, for the arc being created to pass through. This node can be freely defined by placing the pointer at the desired position, by watching the coordinates displayed in the status bar X=315.16 Y=131.85, or by defining the offsets from another node or point (the option <Z>) or by entering exact coordinates in the property window. As a result, the rubberbanding circle will pass through the specified node, while staying tangent to the entity defined by the step one.



Next, specify the radius (diameter) of this auxiliary circle, either by entering a specific value in the property window, or freely by clicking . As a result, the position of the auxiliary circle will be fixed. A node will be created at the tangency point between the circle and the selected entity.

What is left is defining the direction of the arc on the auxiliary circle between the two nodes. To do this, simply point with the mouse at the desired position. The rubberbanding arc will be flipping following the pointer. After selecting the desired position, click , and the arc will be fixed.



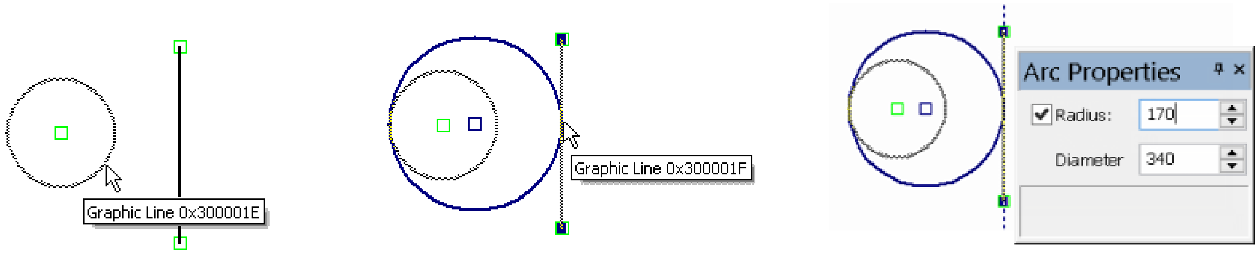


Arc tangent to two entities

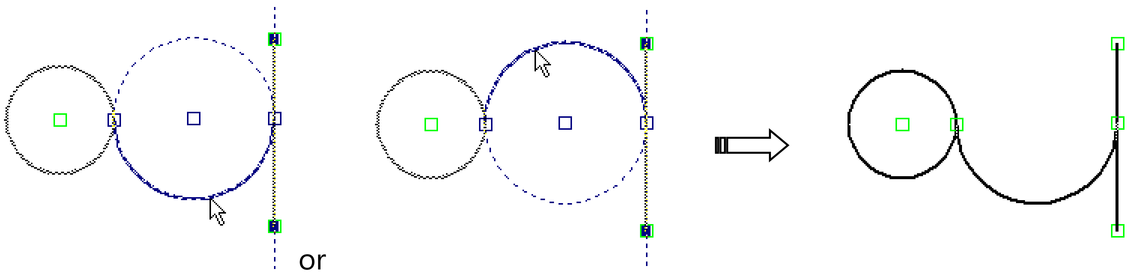
	<M>	Arc tangent to two elements

This automenu option allows creating an arc tangent to two entities simultaneously (circles, arcs or line segments). The arc construction begins with subsequent selection of two reference entities defining the tangency with the arc. As in the previous case, the selected elements are highlighted. A circle will start rubberbanding on the screen, tangent to the selected entities. The position and the size of the circle change as the pointer moves. The circle position and size can be modified using the <Spacebar> key.

Next, define the radius (diameter) of the auxiliary circle, by either entering a specific value in the property window or by clicking . As a result, the position of the auxiliary circle will be fixed. The nodes will be created at the tangency points between the circle and the entities.

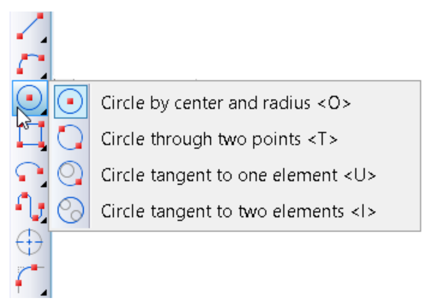


What is left is to define the position of the arc itself in the auxiliary circle between the two nodes. To do this, simply move the pointer to the desired position - the rubberbanding arc will be flipping with the pointer. Select the desired position and click . This will fix the arc.



Creating a Circle

This option, just like previous ones, contains a pull-down list of options for constructing various types of circles.

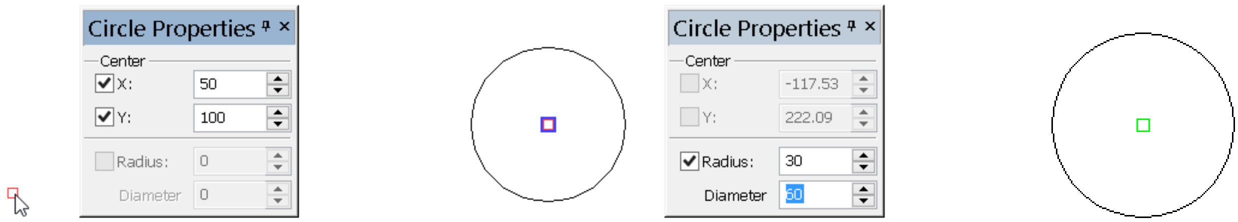


Circle by center and radius

This type of circle can be created using the automenu option:

	<O>	Circle by center and radius

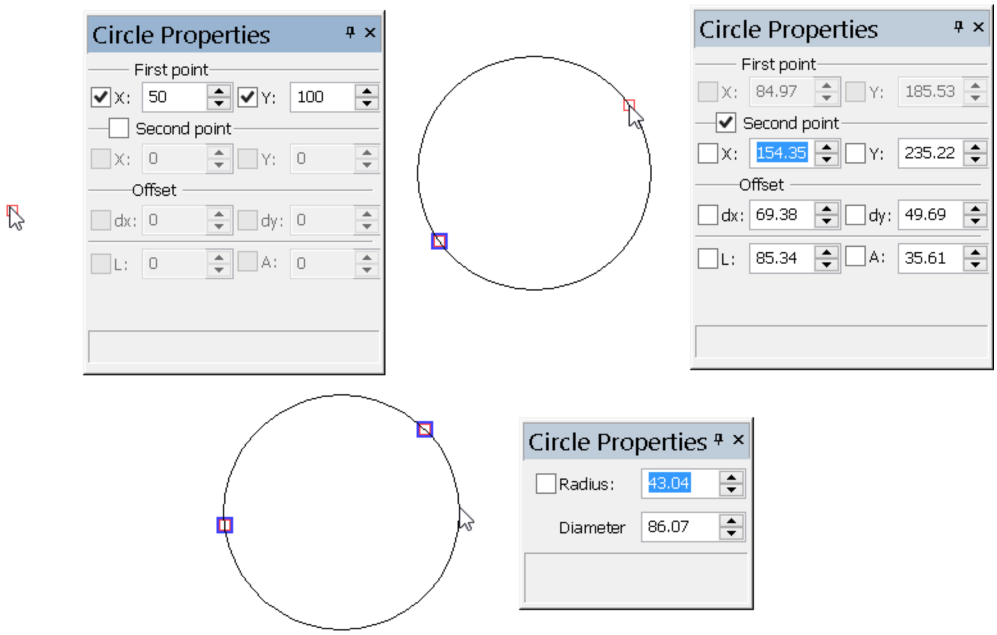
To create a circle, specify the position of the center and the radius (diameter). This can be done freely by the mouse, or, alternatively, by entering the exact values of the center coordinates and the radius (diameter) in the property window.



Circle through two points

	<T>	Circle through two points

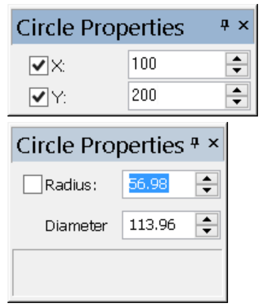
This option is intended for creating a circle passing through two points. The two points for the circle to pass through can be defined by clicking or by entering coordinates in the property window. Next, define the radius (diameter) of the circle. To do this, specify the third point with the mouse, defining the position and the radius of the circle, or, again, resort to the property window.



Circle tangent to one entity

To create a circle tangent to one entity (an arc, circle or line segment), use the automenu option:

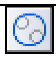
	<U>	Circle tangent to one element



The circle construction begins with selecting the element to which the circle will be tangent. The selected entity will be highlighted, and a circle will start rubberbanding on the screen, tangent to the selected entity. The position and the size of the circle change as the pointer moves. Meanwhile, the property window allows entering the exact coordinates of a point away from the tangency entity defining the circle being created. The position office point can also be specified by clicking , or using the option <Z>. Next, define the radius (diameter) of the circle by clicking or in the property window.

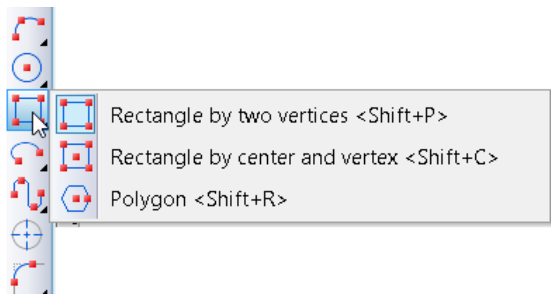
Circle tangent to two entities

To create a circle tangent to two entities, use the option:

	<I>	Circle tangent to two elements

The first step of creating the circle is subsequent selection of two tangency entities. The selected elements are highlighted, and a circle starts rubberbanding on the screen, tangent to those entities. The position and the size of the circle change as the pointer moves. The position and the size of the circle can be changed using the <Spacebar> Key. The position and the radius of the circle can be fixed by clicking at a point away from the tangency entities or via the property window.

Creating Polygons

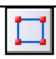


This option also contains a pull down list of icons that allow creating a common rectangle, as well as an arbitrary equilateral polygon.

The created polygons are combinations of separate segments. Each segment can be edited as a separate entity.

Creating Rectangle by Two Vertices

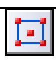
To construct a rectangle by two vertices we use following option:

	<Shift+P>	Rectangle by two vertices:

To construct a rectangle it is sufficient to specify the location of its two opposite corners. The points can be defined arbitrarily either with the help of or by specifying precise coordinates in the properties window.

Creating rectangle by center and vertex

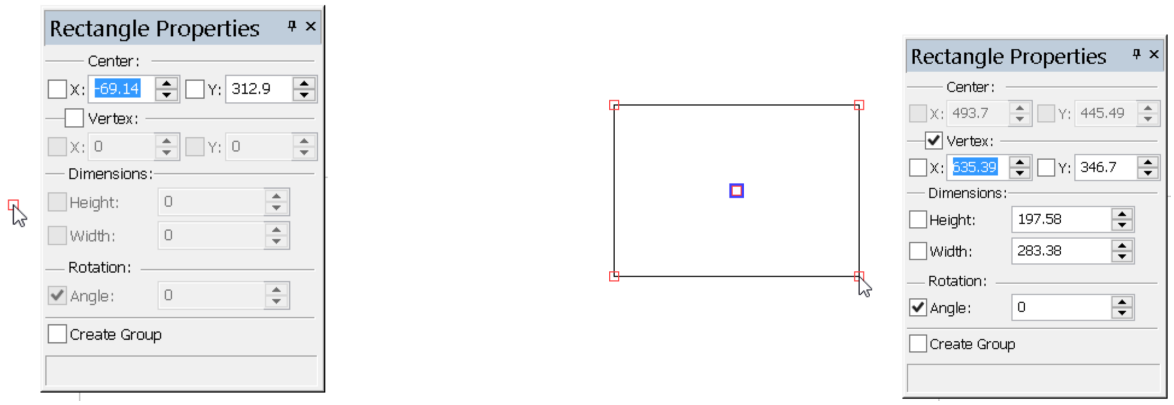
To construct a rectangle by center and vertex we use the option:

	<Shift+C>	Rectangle by center and vertex

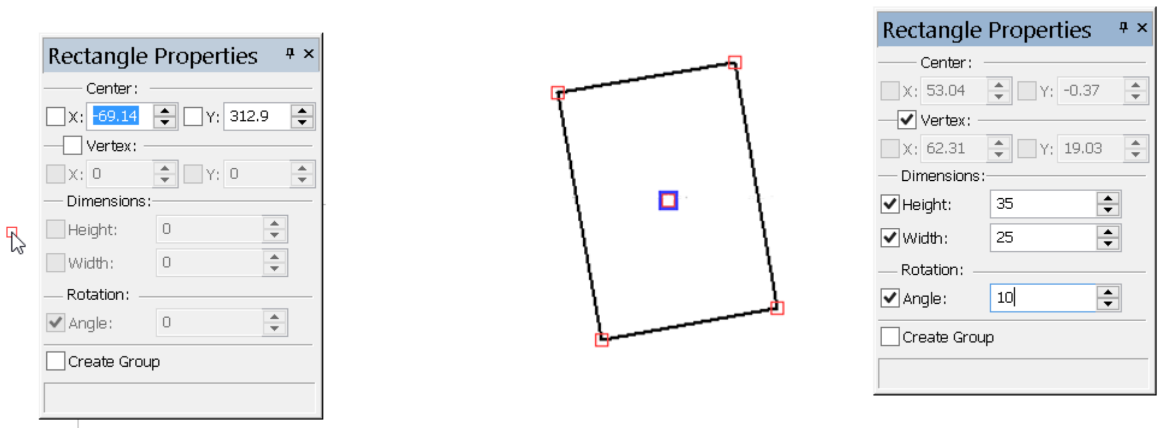
Construction of a rectangle starts with indication of a central point. Location of the point can be specified directly in the drawing's window with the help of or by specifying precise coordinates in the properties window.

Then it is required to specify location of the remaining vertices of the rectangle. This can be done by simply indicating location of one of the vertices of the rectangle on the drawing with the help of . It is also possible to specify in the properties window the precise values of the height and width of the rectangle, but in this case to complete element's creation it is required, after parameters specification, to execute "confirmation" by pressing in the drawing's window.

It is also possible to combine these two methods: to enter, for example, only the height in the properties window, and define the width by indicating the point on the drawing.



or



Before completion of rectangle's creation – i.e., before indication of the coordinates of the second point (vertex), or the values of the height and width of the rectangle – it is also possible to additionally indicate the rotation angle.

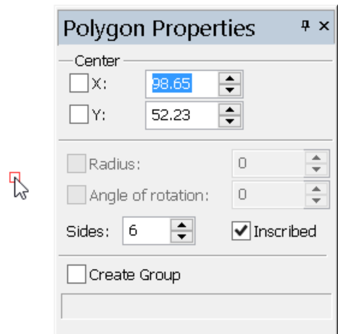
It is possible to define the rotation angle for the rectangle by precise value in the properties window (the **Angle** parameter) or directly on the drawing. By default the **Angle** parameter in the properties window is checked, i.e. the rotation angle is specified only as a numeric value. To specify it in the drawing's window it is required to uncheck it. After that it is possible to specify the rotation angle in the drawing's window.

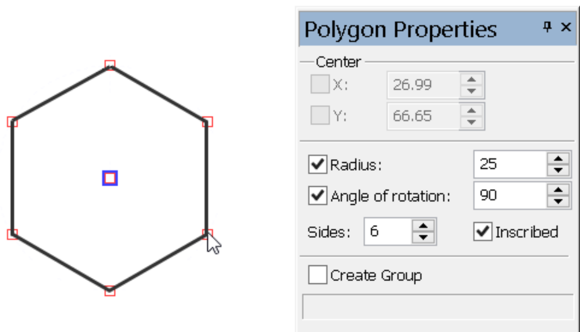
Creating equilateral polygon

To create an equilateral polygon, use the option:

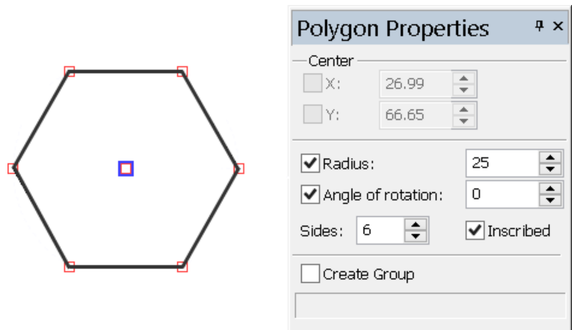
	<Shift+R>	Polygon

To first step in creating a polygon is defining its center. The point position can be defined either by clicking or by entering the exact coordinates in the property window. In the same window, one can specify the number of sides and the type of the polygon (inscribed or circumscribed). Next, define the radius and the rotation angle of the polygon. To do this, either define a point by clicking, to become a polygon vertex, if inscribed, or a midpoint of a side, if circumscribed, or else explicitly enter the radius and the polygon rotation angle in the property window.



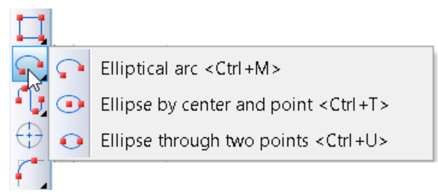


or



Creating Ellipses and Elliptical Arcs

This option, just like previous ones, has a pull down list of icons for creating an ellipse or elliptical arc.



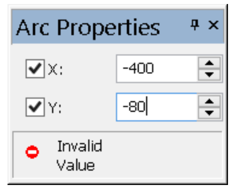
Elliptical arc

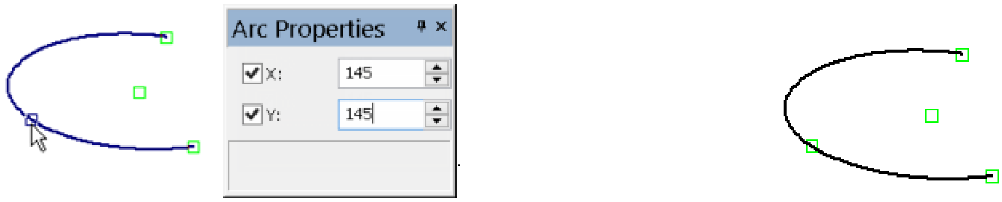
To create an elliptical arc, use the option:

	<Ctrl+M>	Elliptical Arc

After calling the option, define four points: the center of the ellipse, the start point of the arc, the end point of the arc and an additional point in the elliptical arc defining its position. The point positions can be freely defined by clicking, or entered exactly in the property window.

After defining the center and endpoints, an elliptical arc will start rubberbanding on the screen following the pointer. The pointer defines position of the additional point in the arc. If the rubberbanding arc disappears, that means, the arc cannot be created at this pointer position. The point coordinates can also be entered in the property window. In a special pane provided in the window, a relevant warning message is displayed upon an attempt to enter inadmissible point coordinates.

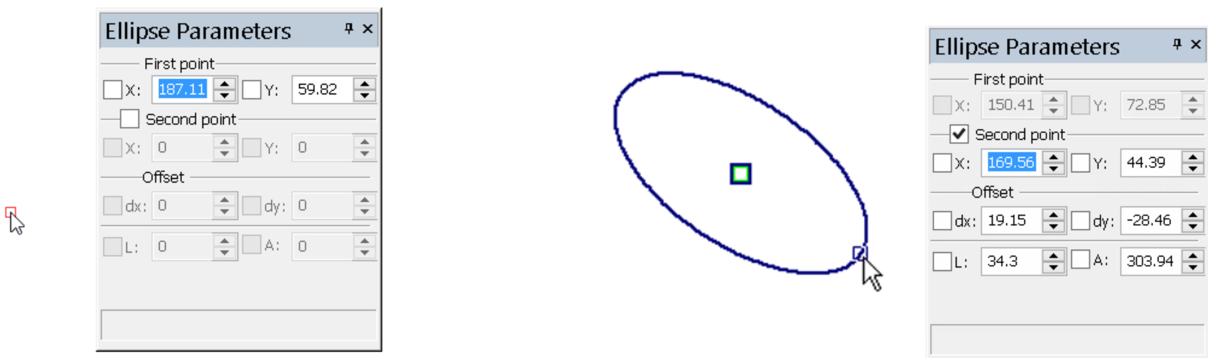


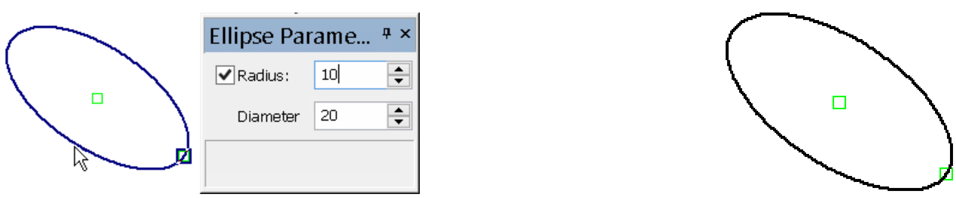


Ellipse by center and point

	<Ctrl+T>	Ellipse by center and point

This option allows creating an ellipse by subsequently defining its center, a point defining the length of one ellipse semi axis, and the length of the second semi axis (the radius). As for other sketch entities, the point positions can be specified by simply clicking within the drawing area or by exact values in the property window.

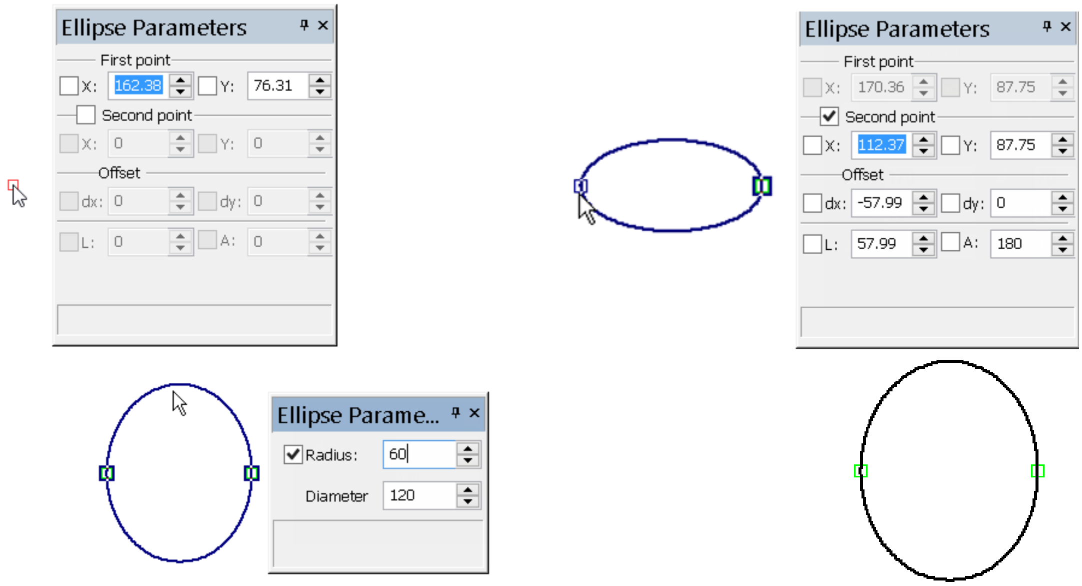




Ellipse through two points

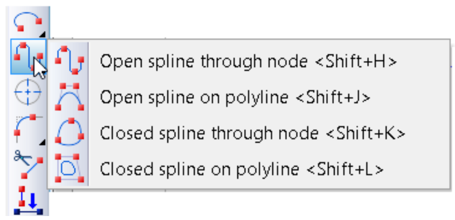
	<Ctrl+U>	Ellipse through two points

The option allows creating an ellipse by specifying subsequently two points as the ends of one of its semi axes and then defining the length (the diameter) or half length (the radius) of the second axis.



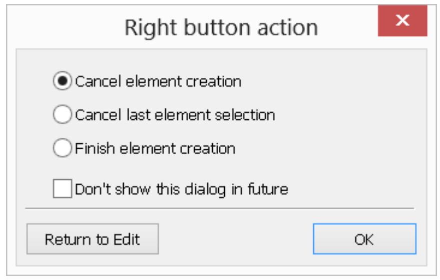
Constructing Splines

The following group of options provides for constructing splines of two main types: those directly passing through the defining nodes, and those using the nodes as vertices of the control polygon. Either type splines can be either closed or open. Creating sketch splines is mostly similar to creating construction spline entities.

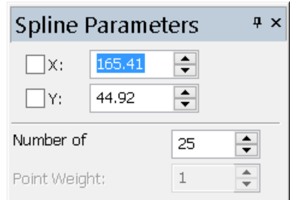


After defining the first node of any spline, the following options become available in the automenu:

	<Ctrl+ Enter>	Finish Spline input
	<P>	Set graphic line parameters
	<Z>	Offset
/	<T>	Set direction at Spline start/end
	<Esc>	Cancel selection



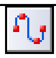
When defining spline nodes, the curve being created will be rubberbanding with the pointer. When creating a spline by control polygon, the control polygon will also rubberband. To complete spline creation, use the options , . One can also right click in the drawing area and select the desired action: cancel spline creation, reject selection of the last spline node, finish spline input.



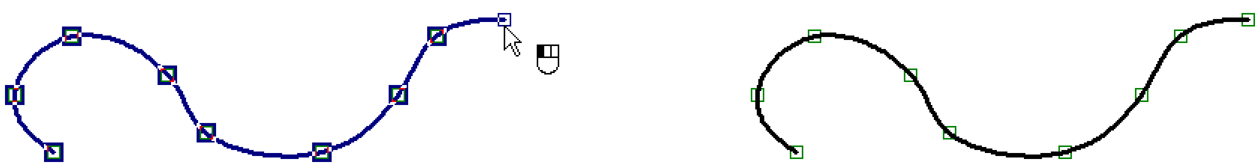
When creating a spline, the property window can be used to define the absolute coordinates over current node being created, the number of spline segments and the weights (for splines by control polygon).

Open splines through nodes

An open spline passing through nodes is constructed using the option:

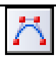
	<Shift+H>	Open spline through node

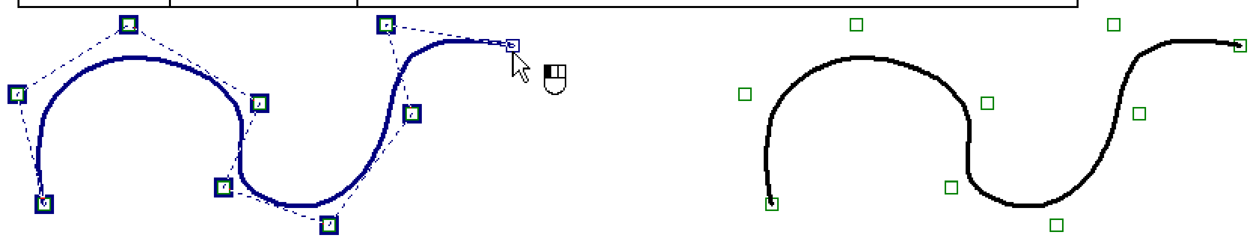
When creating this kind of spline, one can additionally specify end point conditions by using tangency vectors. The vector directions are defined by specifying an additional node using the option . If called right after specifying the first spline node, the option creates the tangency vector for the spline start. Calling the option in the situation, when more than one spline node has been already defined, creates the tangency vector for the end of spline. In this case, the last defined node of the spline is considered its end, and spline creation completes automatically at this point.



Open spline by control polygon

An open spline by control polygon is constructed using the option:

	<Shift+J>	Open spline on polyline



Closed spline through nodes

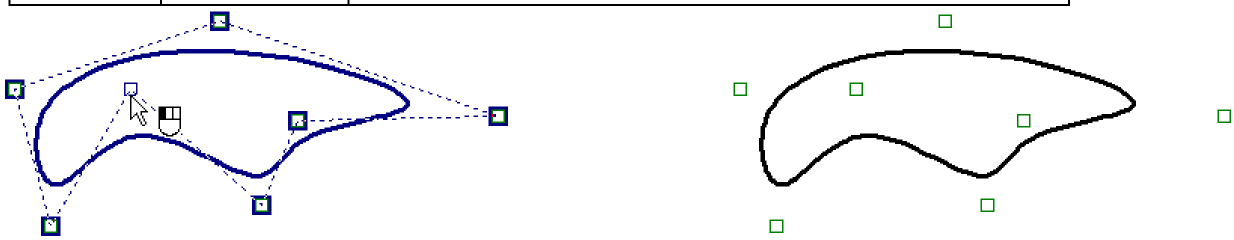
A closed spline passing through nodes is constructed using the option:

	<Shift+K>	Closed spline through node

Closed spline by control polygon

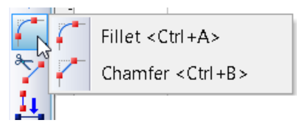
A closed spline by control polygon is constructed using the option:

	<Shift+L>	Closed spline on polyline



Creating Fillets and Chamfers

The options for constructing chamfers and various kinds of fillets between two existing entities are also united in one pull down menu.

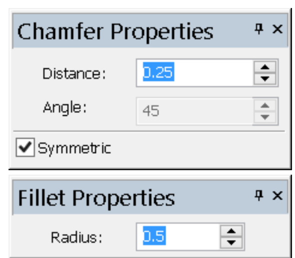


	<Ctrl+A>	Fillet
	<Ctrl+B>	Chamfer

When constructing chamfers and fillets, the existing graphic lines are modified, and new ones are created.

Constructing chamfers and fillets itself is done by simply selecting two intersecting segments, or the segments whose extensions intersect.

Meanwhile, the property window allows defining the width for symmetrical chamfers, or the angle for asymmetrical ones, or the fillet radius.

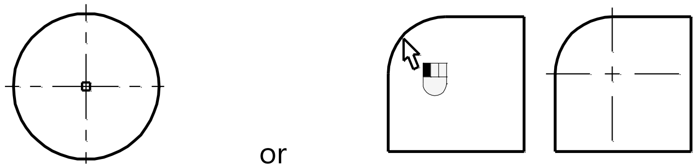


Center Lines

To create center lines for such entities as arcs, circles and ellipses, enter the appropriate mode by calling the automenu option:

	<Ctrl+Q>	Axis lines

After the, simply select any circle, arc or fillet, and center lines will be created automatically.

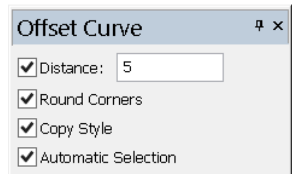


Creating Offsets

	<Ctrl+J>	Offset of a group of Graphic Lines

This option is provided for creating offsets to a group of connected graphic entities. The group of entities can include line segments and arcs connected into a continuous sequence.

The offset type is defined in the property window. The flag **Automatic selection** sets the mode of automatic search for line sequences. In this mode, simply select by clicking at least one line in a continuous sequence, and all the rest will be found automatically. Should forks be encountered, the automatic search stops. It will resume only after specifying the continuation for the search. If the automatic search mode is off, each line needs to be selected manually.



Setting the flag **Round corners** causes automatic filleting of the chain being created. The flag **Copy style** allows transmitting the original line properties onto the offset lines (the type, width, color, etc.).

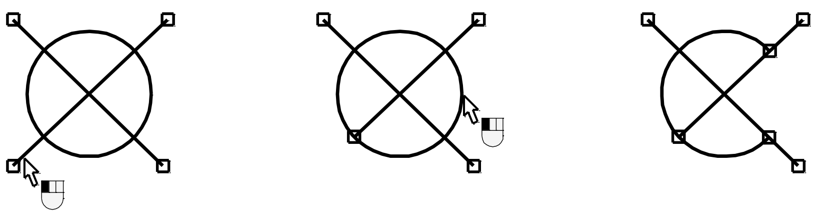
In the process of offset creation, the offset being created will be rubberbanding following the pointer. The offset position can be defined freely by clicking or exactly by entering the offset value in the property window.

Manipulations with Line Segments

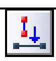
Existing line segments can be modified using the option:

	<Shift+I>	Trim Graphic Line

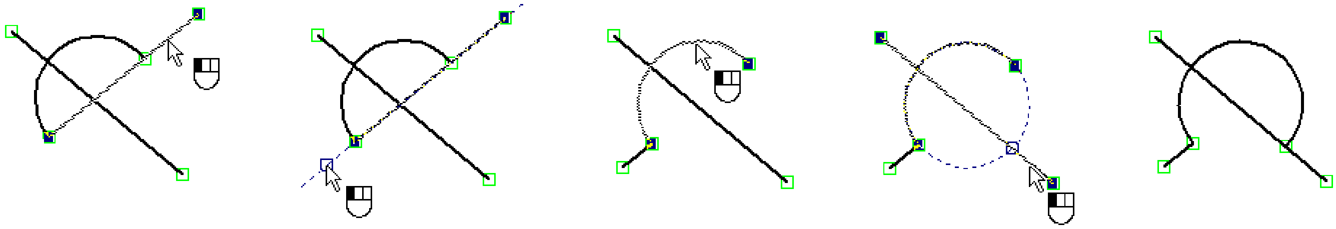
To trim off a piece of a line segment or any other sketch entity, use the mouse and select the piece to trim off. If a free end of a segment was selected, it will be cut off by the nearest intersecting line. If the selected portion of a segment or arc is between two intersections, then the selected entity will be cut at those intersections.



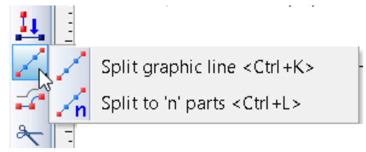
The following option allows modifying any sketch entities except a full circle:

	<Ctrl+I>	Extend Graphic Line

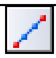
In this case, the selected entity will be extended or shortened. The position of the pointer at selection time is important. The selected element and its extension will be highlighted. If a line segment was selected, then the infinite line extension will be highlighted. If it was an arc, then the extension circle will be highlighted. The end node nearest to the pointer at the time of selection will also be highlighted. This node can be moved by the mouse to either side. The node position and, hence, the new image of the entity can be fixed by clicking . Alternatively, you can select a graphic line to which the current entity needs to be extended or shortened.



To divide existing graphic lines into several pieces, use the options in the respective pull down menu.

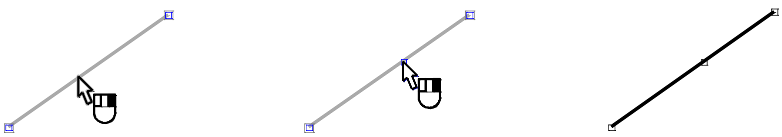


To divide an existing graphic entity into two at the specified point, use the option:

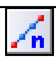
	<Ctrl+K>	Split graphic line

To do this, select any graphic entity created by the command **SK: Create Sketch** or **G: Create Graphic Line**. The selected entity will be highlighted, with a node rubberbanding along the entity, dividing the entity into two. Clicking fixes the node position. The node can also be specified as an intersection point of the selected entity with another graphic entity (a line segment, circle or arc). To do this, selecte a graphic line, whose intersection with the current entity will be the dividing point of the current entity.

As a result, a node will be created at the specified position, dividing the original entity into two.

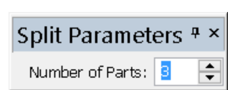


To divide an entity into an arbitrary number of equal parts, use the automenu option:

	<Ctrl+L>	Split to 'n' parts

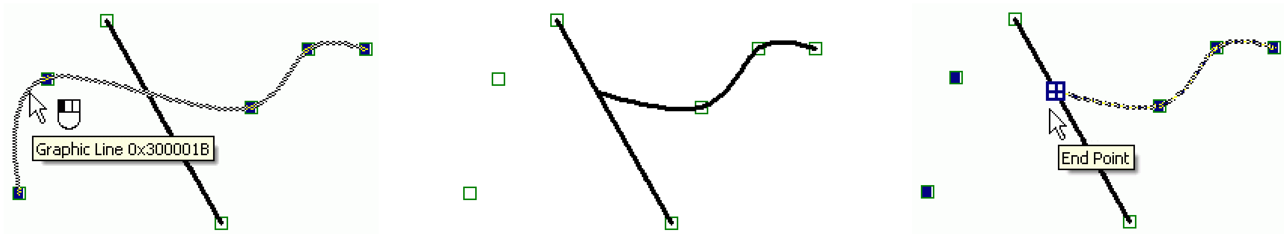
Upon calling the option, specify an entity to divide. If a closed entity was selected (a circle, ellipse or a closed spline), additionally specify the start point of division. As a result, the selected entity will be divided into the specified number of equal parts. The nodes will be created at the division points.

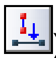
The number of division parts of the entity is specified in the property window.



Modifying splines

Spline modifications are done somewhat different then other sketch entities. Let's review an example of trimming spline using the option . Upon calling the option, pick on the spline near one of its ends hanging off the intersection with another graphic entity - a line segment. As a result, the spline will be trimmed up to the intersection point. However, the geometrical characteristics of the spline are not modified by the section. The spline keeps the same set of defining nodes, it is only the visible image that has been trimmed. Instead of a node (that would disturb the spline geometry), a special section point is created at the intersection, called a "graphic line intersection". This point becomes visible only when the spline is selected.



Shortening a spline (the option ) is done in the same way as the trimming – the spline geometry is not modified, however, an intersection point is created in the specified position, limiting the spline visible image.

When dividing spline into parts (the options  and ), the new geometrically coinciding splines are created, based on the same nodes as the original spline. The number of created splines will correspond to the number of divisions. The visible image of each spline will be limited by the trimming points according to the bounds of the respective parts of the original spline. In this case, the spline may have two trimming points limiting its image.

The position or the trimming point of a spline can be modified when editing the spline (see the section "Editing sketch").

Graphic Line Parameters

Parameters of a graphic entity can be defined or modified at any moment of sketch creation or editing. The dialog box defining the parameters is called by the option:

	<P>	Set Graphic Line Parameters

Detailed description of graphic entity parameters can be found in the chapter "Graphic lines".

Working in the Automatic Parameterization Mode

Let's review in detail working in the auto-parameterization mode. The general rules of creating lines in the drawing are the same here as in the sketching mode. Those fully correspond to what was said in the previous sections of this chapter. What is different is the result of construction – construction lines and based on them constrained nodes are automatically created instead of free nodes.

The constructions created by the system in the auto-parameterization mode depend on the type of the line being created, the object snaps used in its creation and the line parameters defined in the command's properties window. Let's review working in the auto-parameterization mode using several simple examples.

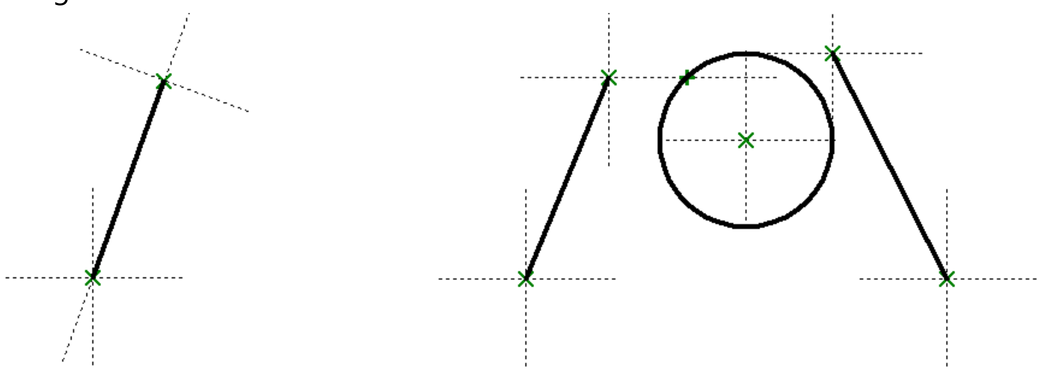
Let's start with constructing *a simple segment*. To construct such a segment, we need to specify the positions of its two nodes. In the sketching mode, the system will create two free nodes in the specified points, and a graphic line between them. In the auto-parameterization mode, the created nodes will be constrained. The method of constructing constrained nodes may vary.

For example, if no snaps are used for creating the nodes of such a segment and you do not enter parameters in the properties window, then the segment nodes are constructed as follows:

- For the first node, a vertical and a horizontal line are created at the specified point, whose intersection will be the node position;
- For the second node, two lines are created as well. The first is constructed as going through the first segment node at an angle to the horizontal. The second is constructed perpendicular to the first line at a distance from the first segment node.

Using snaps and defining parameters in the properties window could change the described rules when constructing a segment.

Suppose, when defining the second segment node, a horizontal snap aligned to another node was used. As a result, the node will be created different from the described above; it will be constructed at the intersection of the horizontal line going through the snap node and a line perpendicular to it. If a double snap to a circle was used in the same situation, such as “Vertical tangency-Horizontal tangency”, then the segment node will be created at the intersection of the lines, which are the horizontal and vertical tensions to the given circle.

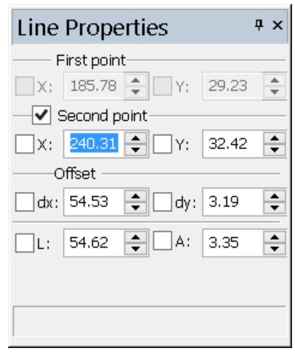


The result of constructing a segment without using snaps (by two clicks in the 2D window)

The result of constructing a segment using the "Horizontal tangent" snap when constructing the second node

The result of constructing a segment using a double snap "Vertical tangent-Horizontal tangent" when constructing the second node

Parameters defined in the command's properties window have a similar effect on the result of auto-parameterization.



For example, if the user defines the X or Y coordinate of the node in the properties window, then the node will be constructed on the vertical or horizontal construction lines with the specified coordinate.

If, on the other hand, the offsets dx or dy were defined, then the lines for the node construction will be created parallel to the vertical or horizontal line going through the first node (from which the dx or dy offset was counted). The parallel line parameter will have the value from the respective field of the properties window. If no horizontal or vertical lines go through the node, from which the offset was counted, then those will be automatically created by the system.

When defining an angle value, a line will be created going at the angle to the horizontal; when defining a length value - a circle of the respective radius.

> If some parameter value is defined by a variable or expression in the properties window, then that variable or expression will be entered in the parameters of the respective construction line. This rule is applied to all sketch options.

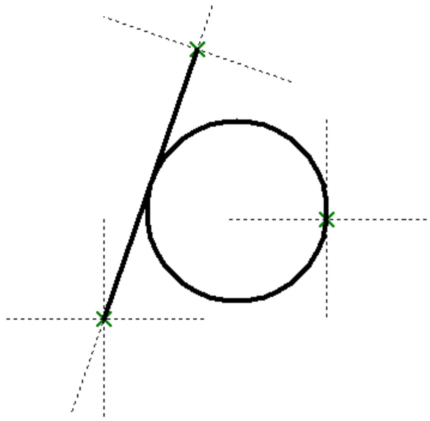
In the case of a combination of any of the described possibilities, a node is created at the intersection of the respective construction lines.

This approach (defining parameters in the properties window) can be combined with using snaps. For example, if only one of the node position-defining parameters is entered in the properties window, and a snap is caught, then, after clicking, the node will be created at the intersection of the construction lines corresponding to the parameter defined in the properties window, and the snap.

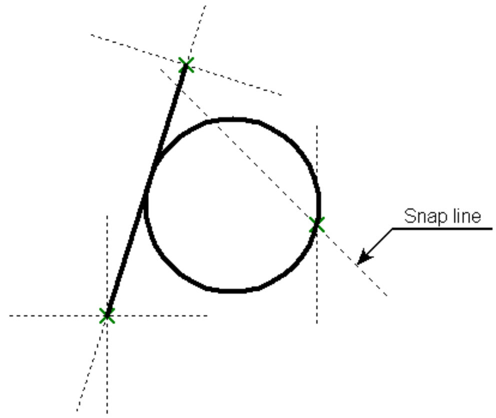
It is also possible to define only one of the parameters in the properties window and not use any snaps (the precise point position in this case is set by clicking in the 2D window). In this case, the first of the construction lines, at whose intersection the created node will be positioned, is defined by the specified parameter. As for the second construction line, the following can be used:

- a vertical or horizontal line (if the X, Y, dx, or dy value was entered in the command's properties window);
- a line at a distance from the start node and perpendicular to the first line (if an angle value was entered in the properties window).

For the second example, let's review constructing *a circle tangent to one element*. When creating such a circle, a node is constructed after selecting the tangency element, through which the circle should go. In the sketching mode, then the node is created free. In the auto-parameterization mode, it is created as lying on the intersection of two lines (a vertical in the horizontal), or defined by the snaps used in its creation. For example, if snapping to a straight construction line was used to define the node position, then the node will be created lying on the intersection of that line and a horizontal/vertical line going through the specified point. After that, a construction circle is created through the resulting node, tangent to the selected element; it is then used to create a graphic circle.



Result of creating a circle not using snaps



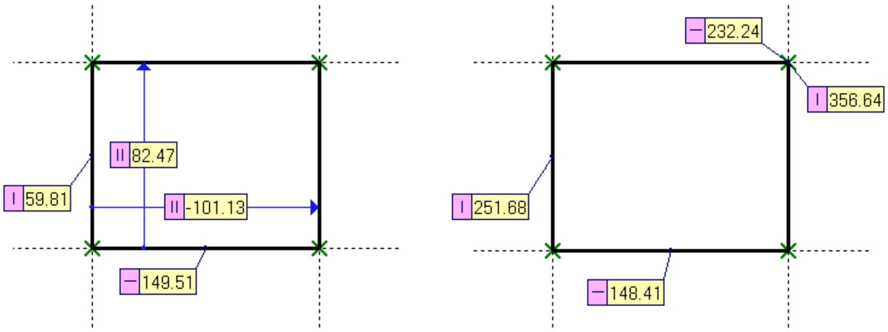
Result of creating a circle using a snap to a construction line

Consider another example - constructing a *rectangle*. In this case, the user needs to define the nodes of its diagonal corners. The two other nodes, the rectangle side graphic lines and all necessary constructions will be created by the system automatically. The dialog layout in the properties window is fully analogous to the dialog for a simple segment.

The first node of the rectangle is created as a node at the intersection of a vertical and horizontal lines (if its position was picked with 🖱 in the drawing window or X, Y coordinates entered in the properties window), or based on the used snaps.

The type of other constructions depends on the method of defining the second rectangle node. If it was defined by simply clicking in the drawing window with 🖱 or by defining the dx, dy offsets in the properties window, then the other lines will be created based on the lines of the first node as parallel to them. When defining the position of the second node using the “Length” or “Angle” parameters, then the construction will be created by being also based on the lines of the first node, but according to the specified parameters (similar to the rules described for a segment). Such construction allows obtaining a parametric model of a rectangle with a pair of base lines.

When the second node position of the rectangle is defined by the X, Y coordinates in the properties window or using snaps, the constructions are created independent (or just partially dependent) on the lines of the first node. For example, if the second node of the rectangle was defined with the absolute coordinates in the properties window, then it is created on the intersection of another pair of vertical and horizontal lines. Two other nodes of the rectangle are constructed on the lines parallel to the lines of his first node in going through the second node. As a result, a parametric model is obtained with two pairs of base lines. When editing such a rectangle, two its diagonal nodes will move independent from each other.



Creating a rectangle with one pair of base lines (both nodes are defined by clicking in the 2D window)

Creating a rectangle with two pairs of base lines (the first node is defined by clicking in the 2D window, the second - by entering the X and Y coordinates in the properties window)

Auto parameterization of other line types is done according to similar rules.

Please note that snapping to graphic lines (snapping to a graphic line itself, to an intersection of graphic lines, to a midpoint of a graphic line, etc.) with the purpose of creating nodes and lines of the drawing is implemented by using construction lines underlying the selected graphic lines. By a "*construction line underlying a graphic line*" we mean a construction line that is geometrically aligned with the given graphic line and serves as its parent.

If a graphic line without a parent construction line was selected for snapping, the system will create the construction line automatically. The created construction line becomes the parent of the graphic line. For example, if a graphic circle line was selected, with a certain radius and the center aligned to a node, then the system will create a construction circle with the center at the same node and the same radius. The new construction line becomes the geometrical basis for the graphic circle line. Similarly, a spline graphic line is converted to a graphic line lying on a construction spline going through the same nodes. For a segment with no underlying construction lines, a line is created that is going through the two segment end nodes.

Another common situation is when the general snapping rules in the auto parameterization mode require creation of a new construction line which would coincide with an existing one. In such a case, the system will be using the already existing line, which helps reduce the number of created construction lines. For example, when making a vertical snap to a node, the system must construct a vertical line going through the selected node. But if there is already a vertical line through that note, the system will use it without creating a new line.

The drawing process in the automatic parameterization mode can be combined with the conventional method of creating a parametric drawing.

Editing Sketch

Sketch editing is done by the command **ESK: Edit Sketch**. The command is called by one of the following ways:

Keyboard	Textual Menu	Icon
<ESK>	Edit > Draw > Sketch	

The command can also be called by pointing at a sketch entity in the command waiting mode and clicking , or right clicking and selecting the item "Edit" in the context menu. Alternatively, the editing command can be accessed directly from the sketch creation command by selecting the option .

> Sketch lines are treated as regular graphic lines. Thus, for their editing it is also possible to use the command **EG: Edit Graphic Line**.

After invoking this command, the following choices become available:

	<*>	Select All Elements
	<R>	Select element from list
	<Esc>	Exit command

After invoking this command, a user can select the sketch line by pointing at it with the cursor and pressing . Selected element will be highlighted. Several elements can be chosen either by using selection with a window or by selecting successively several elements with the help of <Shift>+. To undo selection of the element, the mouse together with the pressed *left* key <Ctrl> can be used.

After choosing one or several sketch lines the following options will be available in the auto-menu:

	<P>	Set selected Element(s) Parameters
	<D>	Copy Properties from Existing Element
	<I>	Select Other Element
	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

If only one line is selected, the following option also appears in the automenu:

	<O>	Create Name for selected Element

When editing sketch lines, please pay attention to the status of the icon in the system panel. If it is disabled, then editing will be done in the sketching mode. When the option is enabled, editing will be done in the automatic parameterization mode. In this mode, just like when creating a sketch, the system will be slipping construction elements tied by parametric relations beneath the edited lines. In addition, the system will attempt to parameterize not only the edited elements themselves, but also the lines that will be employed for the editing. For example, if the position of one of a sketched line segment nodes is changed in the automatic parameterization mode by defining its new position with an object snap to another sketched line, then construction lines will be slipped beneath both segments.

To delete selected elements, use the option .

Editing a Line Segment

After selecting a line segment, select one of the segment nodes, the one to move. At this moment, new coordinates can be defined for the selected node in the property window. Besides, after selecting the node, the segment starts rubberbanding. The rubberbanding image defines the new position of the segment being edited. Rubberband the segment by the mouse to the desired position and click . This fixes the segment in the new position.

Editing a Circle, Ellipse, Arc and Elliptical Arc

If the selected element is a circle, ellipse or an arc (except an arc through three points), the second click on the selected entity launches the mode of editing the radius of the circle, ellipse or arc. A new value of the radius (diameter) can also be assigned via the property window or by rubberbanding the entity image to the desired position and clicking .

This way of editing is not suitable for an arc constructed through three points, and an elliptic arc. In this case, after selecting the arc, you need to select a node the arc is passing through, and move it to the desired position or enter the new placement coordinates of the node in the property window.

There is yet another way to modify any type of an arc or fillet. Select one of the end nodes of the selected arc, and then move the rubberbanding image of the arc into the desired position.

After selecting the arc of a circle an additional option becomes available in the automenu:

	<Tab>	Change Arc Direction

This option replaces the selected arc with another arc of the same circle. When applying this option to the arc constructed by 3 points, the middle (second) node of the initial arc is automatically removed.

Editing Spline

After selecting a spline, the automenu gets an additional option:

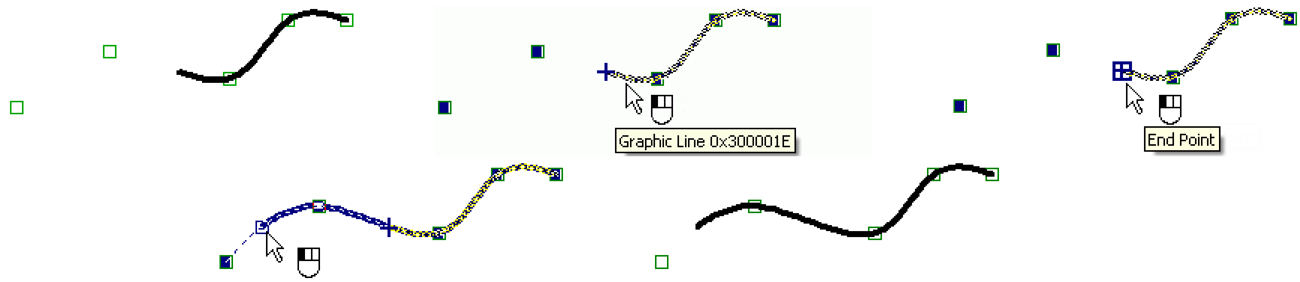
	<Ctrl+X>	Switch to “Insert Point” mode

This option allows adding an additional point to a spline. After calling the option, select a spline node nearest to the additional node location. Then pick the side where the new node should be created with respect to the selected node, and then define its position.

To edit an existing node, select it after selecting the spline. The node and the whole spline will start rubberbanding following the pointer. The new position of the node can be specified by simply moving the pointer to any location and clicking , or by specifying new coordinates of the node in the property window. The same window provides for entering the new weight value of the node (for splines by control polygon), as well as the number of spline segments.

To delete an existing spline node, select the node and then use the option .

When editing a spline that was divided or trimmed, one can also change the position of the trimming points (the intersection points) of the spline image. These points are highlighted when the spline is selected, and become available for selection. The selected point can be moved along the spline and fixed at a new position by clicking .



When selecting the open spline for editing by nodes, the options for specifying directions of the tangent vectors for the spline end points will be also available in the automenu:

	<F>	Set direction at Spline start
	<E>	Set direction at Spline end

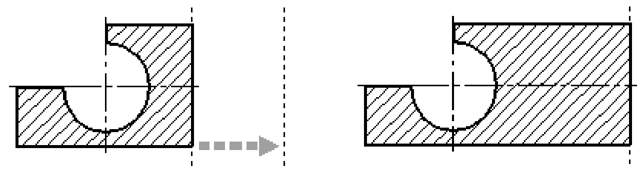
Detailing Elements

HATCHES AND FILLS

Hatches and fills are created by the command **H: Create Hatch**. Hatches are used, besides the primary purpose, as a means of various manipulations: as contours for hidden line removal, as profiles and as base data for creating three-dimensional models (in T-FLEX CAD 3D only).

The hatch or fill area may consist of a single or multiple contours. The left diagram shows a single-contour hatch, and the right one – a three-contour hatch.

Since the contour lines are “tied” to construction elements, modifications of the latter result in appropriate adjustment of the boundaries of hatch contours.



Various hatch attributes provide control over the contour filling pattern, ranging from standard to special technical ones to even various artistic types. Fills provide filling of the profile area with the specified color.

Custom hatch types can be defined if not found among the standard ones provided by T-FLEX CAD. See details in the chapter “Creating user-defined lines and hatches”.

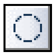
Applying Hatches

Enter the command **H: Create Hatch**. The command is called as:

Icon	Ribbon
	Draw → Draw → Hatch
Keyboard	Textual Menu
<H>	Construct > Hatch

The following options are available to the user:

,	<Ctrl><F>	Free mode on/off toggle
	<P>	Set selected Element(s) Parameters
	<Alt+P>	Copy Properties from Existing Element
	<F5>	Preview mode
	<X>	Automatic Contour search parameters
	<A>	Automatic Contour search mode

	<A>	Manual Contour input mode
	<N>	Select Node (in Manual Contour input mode)
	<C>	Create full Circle Contour (in Manual Contour input mode)
	<E>	Create full Ellipse Contour (in Manual Contour input mode)
	<S>	Create full Spline or Polyline Contour (in Manual Contour input mode)
	<F4>	Execute Edit Hatch command
	<Esc>	Exit command

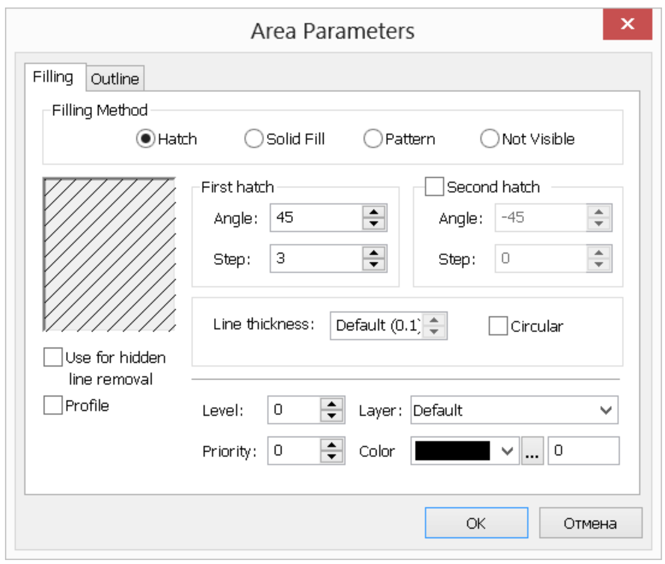
Hatch Parameters

To define hatch parameters, call the option <P>. This will bring a dialog box on the screen, named “Area Parameters”. Some of the hatch parameters can be defined on the system toolbar (see the topic “Defining Hatch Parameters on the System Toolbar”).

Note that defining parameters prior to inputting the hatch contour makes the settings default for all future hatch creations. To set parameters for one particular hatch, do so in the middle of the hatch creation.

“Filling” Tab

Common parameters for all types of filling



Filling method. This item defines the way of filling the contour. The parameters for each of the ways are described below.

Use for hidden line removal. In this case, the contour will be used for removing hidden lines. Any elements with lower priority will be hidden behind the hatch. This is true for assemblies as well.

Profile. With this parameter set, the hatch will be used as a profile for generating a profile file in the command **PR: Write Profile**. This is necessary for displaying geometrical information about the contour of a part for further processing.

Level. This is an integer in the range from -126 to 127. The level defines whether the hatch will be displayed upon a redraw.

Priority. This is an integer in the range from -126 to 127. The priority defines the order of drawing graphic elements on the screen (the greater the priority, the "more prominent" is the element).

Layer. Defines the name of the current layer.

Color. The hatch color can be selected from a table or by number (0-256).

Hatch parameters

A hatch can be filled with solid lines with any slant angle in one or two directions.

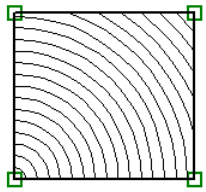
Angle. The slant angle in degrees with respect to the X-axis.

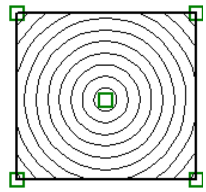
Step. The distance between hatch lines.

Second hatch. With this attribute set, hatching is done in two directions.

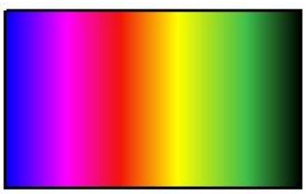
Line thickness. Defines the hatching line thickness.

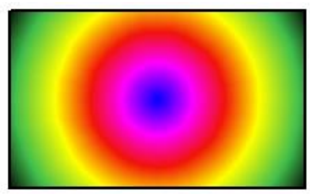
Circular. With this flag on, hatching is done in concentric circles with specified parameters (step, color, line thickness, etc.). In the case attachment point is not selected, the position of the center is defined by the system automatically. Otherwise, the center is placed in the attachment point.



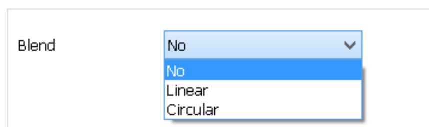


Fill parameters





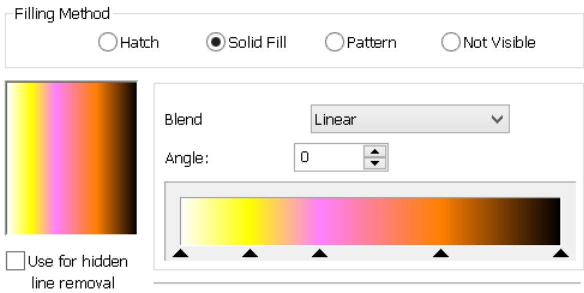
A fill type is defined by the **Blend Colors** options. You can make the following choices in the drop-down list:



No. Filling is done with solid color. This hatch does not have additional parameters, besides the common ones for all filling methods.

Linear. This fill uses a linear color transition. The scale displayed in the parameters dialog shows the fill palette. By default, a scale of gray is used for the color transition (from white through tones of gray to black).

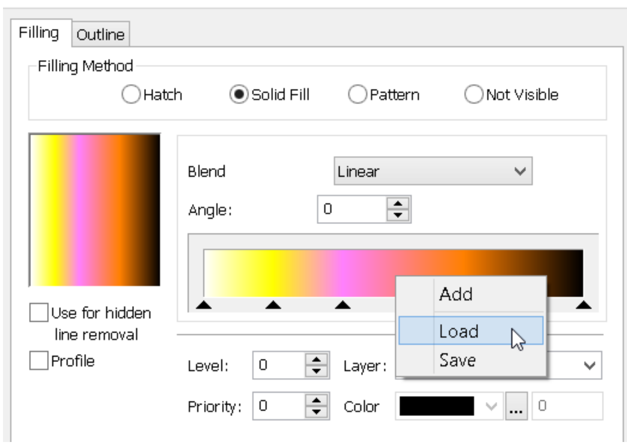
The scale palette can be modified by defining arbitrary colors in arbitrary quantities.



To define a new color, double-click anywhere on the scale. A standard Windows dialog for defining color will appear. After selecting a color and closing this window, the specified color will appear on the scale. A color position on the hatch transition scale is indicated with a triangular marker below the scale. Using the marker, you can set the new position of the color on the scale. To do this, point to the desired marker, depress and, while holding the button down, drag the marker to the desired position.

To delete a color from the transition scale, drag its marker out of the scale. To modify one of the transition scale colors, double-click on the respective color marker.

You can also use the context menu accessible by right clicking on the scale image in order to set up the color transition scale.



A color scale setting can be saved into an external file "*.col" in order to quickly load it in the future.

An additional parameter **Angle** defines the fill rotation angle.

Circular. This is a fill with a circular color transition. Its parameters are similar to those of a fill with the linear color transition, except for the fill angle (it is not defined for a circular hatch).

> You can define the center of a fill with a circular transition. To do this, when editing such a field, define its start point. For details, refer to the section “Editing Hatches and Fills”, the topic “Defining the Hatch Start Point”.

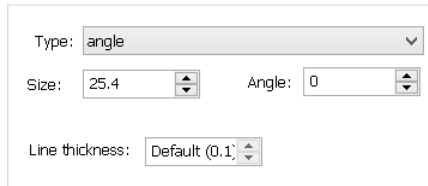
Filling pattern parameters

With this area filling method, the hatch type is defined by a description stored in a file of a special format. The descriptor file of the standard T-FLEX CAD hatches is stored in the “PROGRAM” folder under the name “TCAD.PAT”. The name of the standard hatch pattern file is defined in the command **Customize > Options…**.

> The same format is used for the hatch pattern descriptor file as in AutoCAD system. If some type of hatching is not available, it can be created by the user or copied from AutoCAD system.

In the cases when standard hatches do not suit the user, then user-defined hatch types can be created. Special format files “*.grb” are used for the user-defined hatch descriptors. These files are located in the …/Program/HatchPatterns folder. More on this is in the chapter “Creating user-defined lines and hatches”.

The additional parameters of filling by pattern include:



Type. Defines the filling pattern. The type is selected from a list containing standard and user-defined hatch types.

Size. Defines the scale factor of the pattern hatch. With a very small scale factor the hatch may appear as a solid fill.

- **Angle.** Defines the slant angle of the hatch.
- **Line thickness**. Defines line thickness, used when drawing a hatch by pattern.

Not Visible hatch

With this hatch type set, the hatch will not have its own graphical representation on the drawing. This may be necessary if the hatch is used solely for hidden line removal, or for creating a 3D profile or a 3D model.

“Outline” Tab

The hatch contour can additionally be outlined with lines. This is handy when the hatch contour is defined based on construction lines and nodes, in the absence of graphic lines. The hatch outline lines are set up in the same way as common graphic lines.

Defining Hatch Parameters on the System Toolbar

When creating and editing a hatch, some of the parameters can be defined directly in the system toolbar, without using the <P> option:

Color box . Indicates the line color of the hatch being created or edited.

Fill method box Hatch. Defines the hatch contour filling method.

"Use for hidden line removal" icon .

Depending on the fill method setting, the system toolbar may have additional fields:

Hatch:

Hatch slant angle input box 45.

Hatch line step input box 3.

By pattern:

Hatch slant angle input box 0.

Hatch scale factor input box 25.4. Defines the scale factor of the hatch.

Pattern type input box angle.

Copying Parameters from Existing Hatches

The values of parameters of the hatch being created can be quickly copied from an already existing hatch. To do so, use the option:

	<Alt+P>	Copy Properties from Existing Element

> This option is available in the automenu of the command before creation of the hatch has started or during the process of its creation.

After the option is invoked, it is sufficient to indicate the hatch whose parameters' values must be transferred to the hatch being created.

To assign the copied values of parameters to all new hatches, enable the additional option before selection of the original hatch:

	<S>	Set Properties as Default

When this option is enabled, the copied parameters will be stored as default parameters.

> This option simplifies creation of hatches with identical parameters. However, it does not allow us to copy individual parameters or parameters from an object of another type. In such cases, it is more convenient to use the general mechanism of editing parameters of the elements in the properties window.

Hatch Result Preview

There is an option to view the result of hatching without confirming the operation's creation in the command:

	<F5>	Preview mode

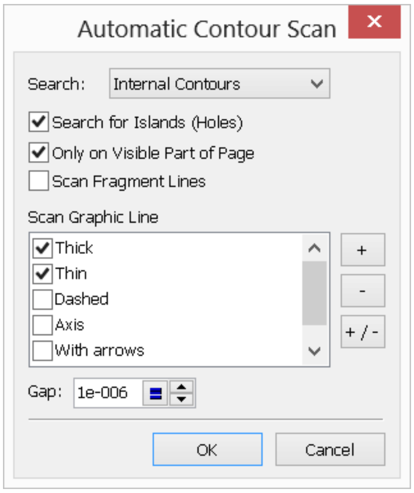
> When this option is enabled, in case of presence of a closed contour or contours, the hatch being created is immediately displayed on the screen with the same parameters as defined in the command.

Defining Hatch Contour

The hatch contour can be defined in two modes, the automatic contour search mode and the manual contour input mode.

Automatic Hatch Contour Search Mode

To activate this mode, press the icon in the automenu or type <A> on the keyboard. This mode works only with the graphic line contours. To find a hatch contour, position the pointer at a point lying inside the anticipated hatch contour, and click . The found contour will be highlighted. If automatic contour search lasts longer than three seconds, a window is displayed with the **[Cancel]** button.



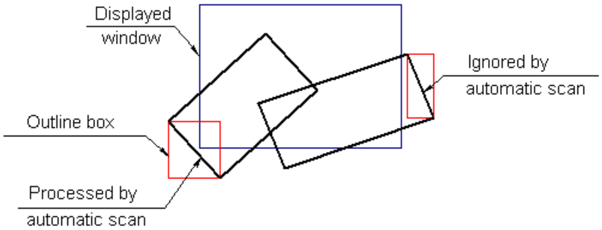
Successful definition of a hatch contour requires the graphic lines to form "tight" contour (with no gaps).

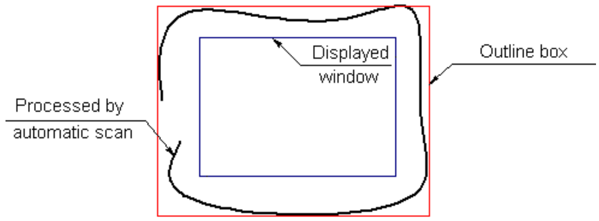
The result of the automatic tracing is affected by the parameters set for this mode. The automatic search parameters are defined in a dialog called by the option . In this dialog window, first of all specify the types of the graphic lines to be considered in the automatic search for the hatch contour.

The threshold value of the gaps between graphic lines is determined by the parameter "Gap". If the graphic lines pass at the distance, which is less than or equal to that defined by the given parameter, then the system will assume that they have an intersection point and may therefore include them in the hatch contour.

The **Search:** parameter allows defining more specific requirements to the result of tracing a hatch contour. If this parameter has the **External contour** value, then the system will search for the largest closed contour. The internal contours are ignored in this case. When using the **Internal Contours** value, the system finds the minimal contour which still contains the position of the pointer while searching. The internal contours processing depends in this case on the state of the additional flag **Search for Islands (Holes)**. If this flag is set, then be found internal contours are included in the resulting hatch contour (so that the identified islands are not filled with a hatch). When the flag is cleared, the internal contours are ignored.

Scanning for hatch contours may take a while on very crowded drawings. Search can be restricted per the **Only on Visible Part of Page** flag. In this case, only the entities will be processed whose outline boxes overlap with a portion of the drawing display on the screen. An outline box is the least horizontally oriented rectangle that fully covers the entity. Examples of outline boxes and contour processing situations are shown on the following diagrams.

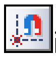


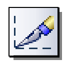


Scan Fragment Lines flag will add lines of 2D fragments for scanning the hatch contour. This flag is set off by default meaning that lines of the fragments will not be considered when searching the closed contours.

Several contours can be selected subsequently. Should there be common graphic lines between different contours, the contours get automatically joined along these lines.

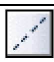
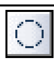
Manual Hatch Contour Input Mode

When manually defining a hatch contour, object snapping is active. Moving the cursor over a drawing element modifies the cursor appearance accordingly and pre-highlights the element. The object snapping can be turned off by pressing on the icon on the "View" toolbar. In complicated situations, when snapping to the right element is difficult, the elements can be selected by typing commands on the keyboard as described below.

If you would like to use already created construction entities as a contour, make sure to be in the constrained drawing mode rather than in free drawing. See that the icon  appears in the automenu.

The first step in the manual definition of a hatch contour is selection of the start point. One can select an appropriate 2D node, or create one by clicking at an intersection of construction lines. Next, define the contour sequentially.

The following options become available after selecting a node:

, 	<Ctrl><F>	Free Mode On/Off toggle
	<End>	Close Contour
	<Tab>	Change arc direction (available when making contour along a circle)
	<Space>	Select Graphic line (available after selecting a node)
	<N>	Select Node
	<L>	Select Line
	<C>	Select Circle
	<E>	Select Ellipse

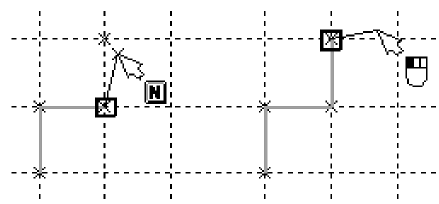
	<S>	Select Spline
	<A>	Select contour automatically
	<BackSpace>	Delete last Contour segment
	<Esc>	Cancel selection

The simplest way of defining a hatch contour is using the <Space> bar that will let you traverse the contour by following graphic lines. Note that this way can only be used when the hatch contour follows graphic lines. In the case of multiple choices, the cursor should point at the desired graphic line when pressing the <Space> bar.

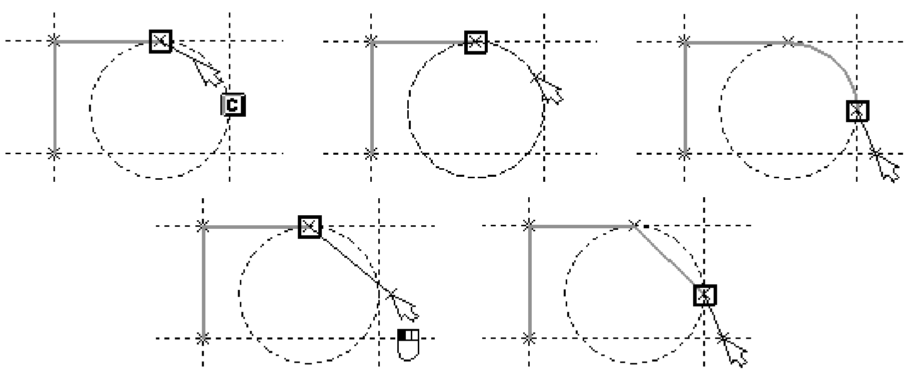
For faster selection, use the <A> option that automatically searches for the next contour line until the contour is closed or a dubious situation is encountered (such as branching lines).

The system can be set up for selecting only construction lines when defining a hatch contour in snapping mode, and not graphic lines. This option can be set in the system customization dialog box under **Customize** > **Options...**, the **Snap** tab.

A contour can be defined using the same actions as in graphic line creation. That's defining subsequently the lines of the contour, each having the start and end nodes and is constrained to construction entities – a line, a circle, an ellipse or a spline. To define the start or end of a line of the contour, select existing nodes (the <N> key) or create new ones (the <Enter> key or ) at two-line intersections.



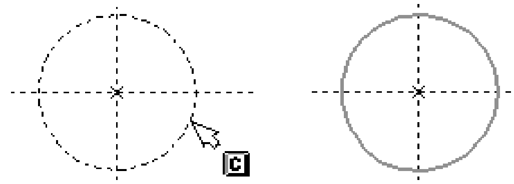
As in graphic entity creation, to define an arc one needs first select a start node on the arc, and then select the circle by typing <C>. Otherwise, the contour will gain the segment between the two nodes instead of the arc.



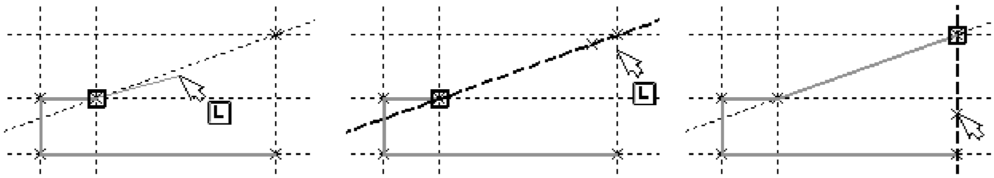
When creating a contour along an arc, the arc direction can be flipped by the option .

To create a full circle contour, point at the circle and type <C>.

Similarly, ellipses, splines, 2D paths and functions can be included in a contour.



In complex cases, when several nodes coincide, the end nodes of the contour lines can be specified by selecting two construction lines whose intersection yields the desired node. This is done in object snapping mode via the options <L>, <C>, <E>, <S>, for lines, circles, ellipses and splines respectively.



In dubious cases, when several construction lines intersect in one point, all necessary nodes can better be created in advance by using the command **N: Construct Node**. The contour then can be defined using the option <N>.

To cancel the last contour line input use the <BackSpace> key.

If the end node of a contour line coincides with the start node then the contour automatically closes, as indicated by a changed color of contour lines display on the screen.

The contour can also be closed using the option:

	<End>, <Home>	Close Contour

This closes the contour with a straight line from the current node to the contour start.

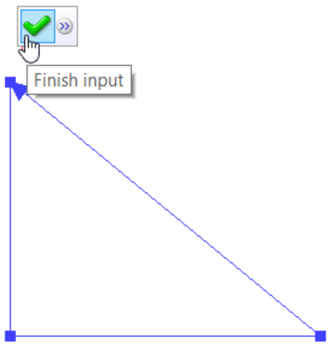
If the hatch contains several contours then the next contour can be input after closing the current one.

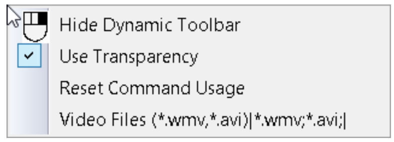
To complete hatching, upon defining the contours use the following option:

	<End>	Finish input (creates hatching)

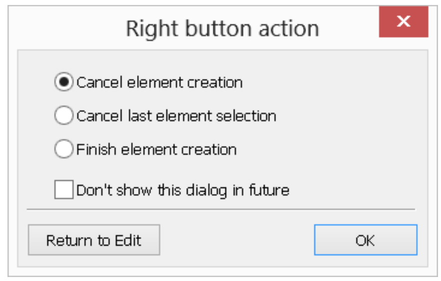
The hatch area then gets filled according to the defined hatch parameters.

To confirm the hatch user can also apply the dynamic toolbar.





In case if the user wants to refuse the usage of this capability in the given mode, it can be disabled by using the "Do not use in current command" command found in the context menu invoked for the given toolbar.



To cancel an action of a contour segment input or the whole contour definition, right-click or press <Esc> key. Upon clicking the “Right button action” dialog box pops up. You can select the command to execute in this dialog:

Cancel element creation. This cancels hatch contour input.

Cancel last element selection. This cancels the input of the last segment of the hatch contour, bringing the contour definition one step back.

Finish element creation. This command closes the contour automatically with a straight line.

The dialog box won’t be displayed again if the option **Don’t show this dialog in future** was set. The right mouse button click will in this case execute the command checked in the dialog box last time it was used.

Editing Hatches and Fills

To modify hatches or fills, use the command “**EH: Edit Hatch**”:

Keyboard	Textual Menu	Icon
<EH>	“Edit > Draw > Hatch”	

Select a hatch or a fill by clicking . One can also use element selection from list, if the element was named. Upon selecting a hatch or fill, the following actions can be performed:

Modifying Hatch or Fill Parameters

This is done via the option <P> that lets modifying hatch (fill) parameters (see parameter description above). One can change the hatch type, for example, from pattern to solid fill. This will fill the contour per the settings for new fill creation.

As in hatch creation, some of the parameters are accessible from the system toolbar upon hatch selection.

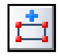
Deleting the Whole Hatch or Fill

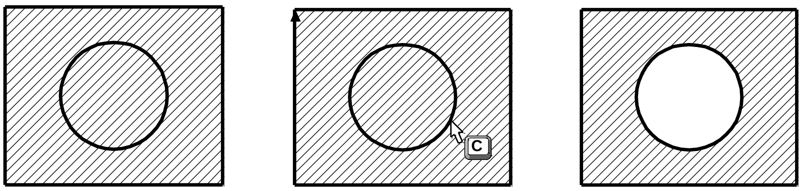
To do this, press the <Del> key (the icon  in the automenu).

In the case when a hatch is referenced by the 3D model, its direct deletion is impossible. The user will have to delete all dependent elements referencing the hatch, which may not be desirable. In such cases, it may be possible to edit the hatch contour. This opportunity is described below.

As in the case of any other model elements, multiple selection of hatches is supported for simultaneous deletion or modifying their parameters.

Adding a Contour to a Hatch or Fill

This is done similar to hatch (fill) contour initial input. Suppose, given a hatch, we'd like to make a hole within. To do this, select the hatch and turn on the contour addition mode (the icon  or <M> key). Then, using the option <C>, input the additional contour – a full circle, and press <End>. The result will be a hatch with a "cut" hole.



Redefining Hatch Contour

Select the hatch whose contour is to be redefined, and use the option

	<K>	Redefine Hatch Contours

To input a new hatch contour, using manual or automatic contour input mode. The edited hatch will assume the new shape upon confirming the input with the  option.

Defining the Hatch Start Point

A start point can be defined for a hatch (either the regular or by pattern or for a fill with a circular color transition). The start point defines the location from which hatching begins.

The start node can be defined by the option:

	<O>	Select Starting Node of Hatch

Upon calling the option, simply click the desired node with .

To reject the start node input, use the option:

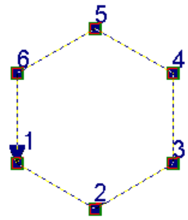
	<T>	Cancel selection of starting Node of Hatch

Displaying the Contour Point Numbers of a Hatch

To toggle the display of the contour point numbers of a hatch, use the option:

	<Q>	Show/Hide contour point numbers

With the option turned on, the points in the all hatch contours are enumerated based on their position in the contours and the contours direction. Points, belonging to the different contours, are enumerated independently. The point numbers are displayed next to the respective nodes. When several subsequent points of a contour coincide, their numbers are displayed next to each other, separated by commas.



Editing a Particular Contour

> Note: only manually defined hatch contours can be edited.

First, select the hatch that includes the desired contour. Then, turn on the contour editing option using the option:

	<M>	Contour edit mode

Now, select the desired contour. The selected contour can be deleted or edited. To delete it, press the <Del> key. When picking a contour, the segment of the contour nearest to the cursor gets automatically selected. Now, the necessary modifications can be done with the help of the automenu.

The following options become available upon selecting a contour segment:

	<I>	Switch to “Insert Point” mode
	<Q>	Show/Hide contour point numbers
	<Del>	Delete selected Contour
	<R>	Change Contour direction
	<F>	Move Contour starting point forward
	<B>	Move Contour starting point backward
	<N>	Select Node
	<L>	Straighten Contour Segment
	<C>	Select Circle
	<E>	Select Ellipse
	<S>	Select Spline
	<Tab>	Change arc direction (available when editing an arc segment of a contour)

	<A>	Link Arc or Circle to Node
	<K>	Break Link with Node
	<Esc>	Cancel selection

Note: the options and only are available for editing the contours defined automatically with the option , constructed on top of 2D projections or as a copy of an existing hatch.

When editing a contour, the following actions are supported: node deletion, node addition and redefinition of the type of lines connecting the nodes. Besides, one can change the contour direction, move the start point back and forth, link arc segments of a contour to nodes.

Flipping contour direction to the opposite

The contour direction is an important hatch property when creating 3D elements. This parameter is inherited by the 3D profiles created based on hatches. For example, the 3D operation **Loft** matches profiles by the start points of the respective contours and requires matching of contour directions as well.

To change the direction of a 3D profile constructed based on a hatch, you need to change the hatch contour direction.

The contour direction is shown by the arrow displayed upon selecting the contour. This arrow also indicates the start point of the contour.

The following steps are to be done in order to change a hatch contour direction:

- Call the command **EH: Edit Hatch;**

- Select a hatch for editing;

- Turn on the contour editing mode (the icon or <M> key);

- Select the contour (point the graphic cursor at and click);

- Flip the contour direction (the icon or **<R>** key);

- Confirm changes by the icon or <End> key.

Moving around contour start

The contour start is the first node selected during the manual hatch contour input.

If the contour is defined automatically or without node selection (as by a full circle), the start point is defined automatically.

The start point placement and contour direction (see above) can only be changed for the contours defined manually.

To move the contour start point, do the following steps:

- Call the command **EH: Edit Hatch;**

- Select a hatch for editing;

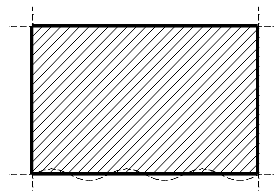
- Turn on the contour editing mode (the icon or <M> key);
- Select the contour (point at it with the graphic cursor and click);
- Move the start point forward (the icon or **<F>** key) or backward (the icon or **<B>** key);
- Confirm changes by the icon or the <End> key.

Editing a particular segment in contour

To edit a contour segment, do the following steps:

- while in contour editing mode, select the desired contour segment;
- select the construction element defining the new contour segment: line, circle, ellipse or spline (use the appropriate option for element selection). The end nodes of the contour segment must be snapped to the selected construction element;
- confirm changes by the or <End> key.

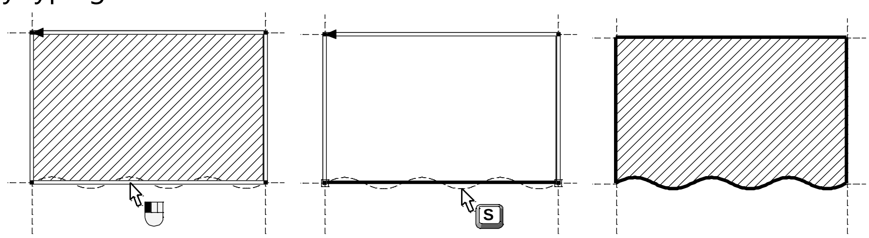
Let's review an example illustrating a particular contour segment editing. The diagram shows the original hatch constructed along construction lines using the "Outline" option. Here, a straight-line segment is to be replaced by a spline.



To do so, call the command **EH: Edit Hatch** and select the hatch. Then, turn on the contour editing mode using the option:

	<M>	Contour edit mode

Select the hatch contour. The following diagram shows the situation after the hatch selection. The contour is highlighted, and the nodes are marked by little boxes. Move the cursor over the desired segment of the contour and click the left mouse button. The selected contour segment also gets highlighted, and the nodes marked by larger boxes. This state is shown on the center diagram. Move the cursor over the spline constructed through the nodes of the contour being edited, and select it as a contour segment by typing <S>.

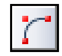


Similarly, one can replace a contour segment by an arc or ellipse, if the respective circle or ellipse is constructed based on the marked nodes. Simply use the appropriate option: ***<C> or <E>***. If the new contour segment was not constructed based on the marked nodes yet passes through them then the contour segment editing should be done via the option "Switch to 'Insert Point' mode" (the icon or <I> key). This option functionality is described below.

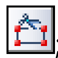
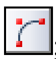
The edited segment will then assume the desired shape. The system is still in the mode of modifying the selected contour segment. If the manipulations with this segment are over, complete the contour segment modification mode. To do so, press the  icon in the automenu.

> The option <Space> is not available for editing the contour. Therefore, it is impossible to snap a contour line to a graphic line. If a contour segment is to be replaced by a graphic line while editing, as, for instance, a wave line, then you need to construct a spline based on the wave line. The intended hatch contour segment can then straightforwardly be replaced by the spline.

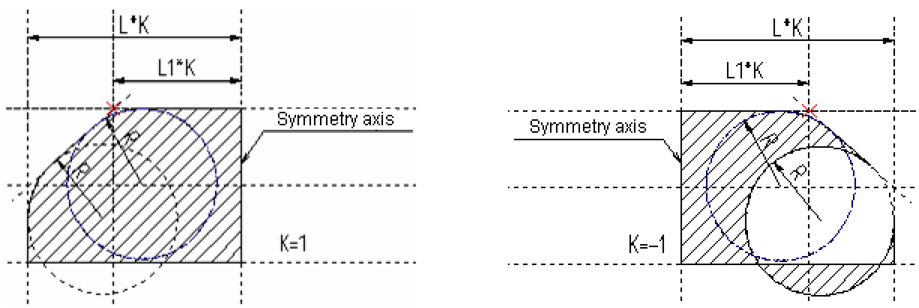
Linking arc contour segment to helper node

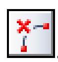
If the edited contour segment is an arc then the option for linking to a helper node  becomes available in the automenu. This node will be defining the choice of an arc constructed on top of the circle to be used for the hatch contour. As the drawing gets modified, the hatch contour will pass by the arc nearest to the linking node.

Let's review an example where a construction line position is modified with respect to a reference line driven by a variable *"K"*. Both circles here are linked to a node on the drawing in order to ensure correct position. The left diagram shows the hatch contour in the original configuration (K=1). Let's link the arc segment of the hatch contour on top of the outer circle to the marked node. To do this, do the following steps once in **EH: Edit Hatch** command:

- Call the hatch contour editing option ;
- Select the arc;
- Call the option ;
- Select the linking node.

The right diagram shows the modified drawing with the *"K"* key equal to *(-1)*. The contour segment on top of the upper circle was linked to the marked node and adjusted correctly. The contour segment on top of the second circle was not linked to the node, which resulted in an incorrect configuration.



To clear or redefine the linking node, use the option .

Deleting a node on a contour

To delete a node on a contour, do the following steps:

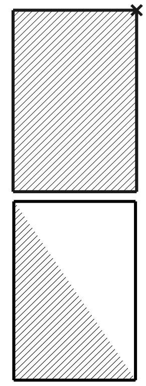
- Call the command **EH: Edit Hatch;**

- Select a hatch for editing;

- Turn on the contour editing mode (the icon or <M> key);

- Select the contour segment the node is snapped to (point at with the graphic cursor and click);

- Select the node (point at with the graphic cursor and click);



- Delete the node (the icon or <Del> key);

- Confirm changes (the icon or the <End> key).

The resulting new hatch contour passes through two adjacent nodes.

Modifying a contour node position

To modify a contour node position, do the following steps:

- Call the command **EH: Edit Hatch;**

- Select a hatch for editing;

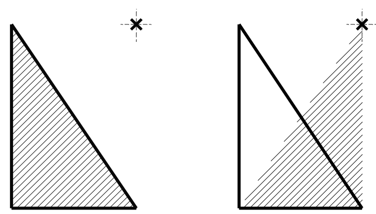
- Turn on the contour editing mode (the icon or <M> key);

- Select the contour segment of the node (point at with the graphic cursor and click);

- Select the node (point at with the graphic cursor and click);

- Move the node to the desired position (the contour rubberbands after the cursor as the node is being dragged);

- Fix the node (click over an intersection point or type <N> key for an existing node);



- Confirm changes (the icon or the <End> key).

As a result of moving, the node point will be connected with the neighbor nodes by straight line segments belonging to the contour (regardless of the former adjacent segment entity types).

Creation of additional nodes on a contour

To create additional nodes on a contour, do the following steps:

- Call the command **EH: Edit Hatch;**

- Select a hatch for editing;

- Turn on the contour editing mode(the icon or <M> key);

- Select the contour segment to split by new node(s) (point at with the graphic cursor and click);

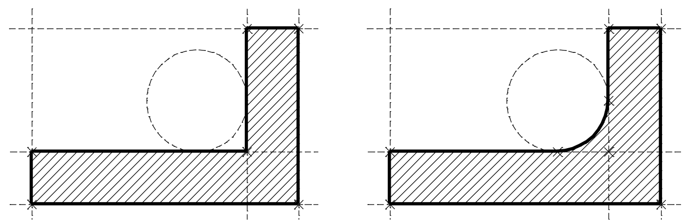
- Turn on the point insertion mode (the icon  or the <I> key), and click on a contour segment. The segment becomes split in two, with the new node between them. The node and the segments rubberband with the cursor, the solid line segment connecting to the previous node and the dashed line one to the next. The order of the nodes after the insertion will be determined by the system automatically, depending on the hatch contour direction. Do not click on a segment near a vertex, as, instead of adding a node, this will start moving the existing node;
- Close the contour between the newly created node and the successive one.

Contour input is complete once the closing node is selected, or the icon  or the <End> key is pressed.

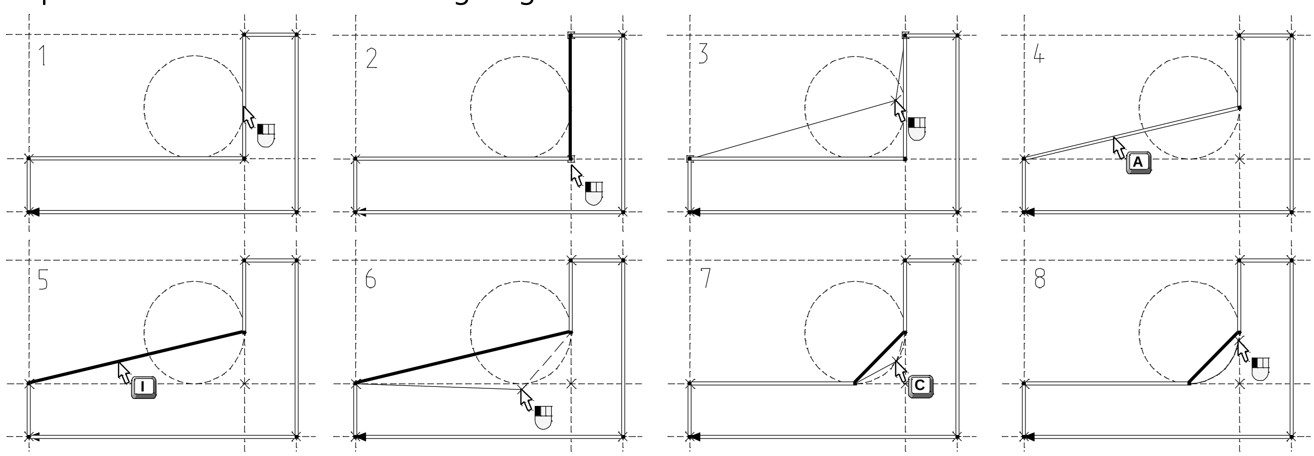
> The system returns to the mode "Contour selected for editing". One can do other modifications, and then confirm all changes.

- Confirm changes (the icon  or the <End> key).

Let us illustrate the above with a specific example. Suppose, a hatch contour needs to be edited that was originally constructed along construction lines using the "Outline" option. The original and target configurations are shown on the diagrams.



To get the result, begin with calling the command **EH: Edit Hatch**. Next, select the hatch and turn on the contour editing mode. Select the hatch contour to be modified. Now, to get the result, perform manipulations shown on the following diagrams 1 – 8.



Upon selecting the closing node (see diagram 8) the contour automatically closes, and the point insertion mode exits. What is left is to press the  icon or the <End> key, and the contour editing task is completed.

One comment to the described procedure: as the contour segment to be modified is selected (see diagram 5), the system will define the previous and the next nodes in the sequence, depending on the hatch contour direction. These will be identified by the dashed and solid rubberbanded lines. In this particular example, the hatch contour direction is clock-wise.

Dimensions

T-FLEX CAD supports all types of dimensions, recommended by the standards ISO, ANSI and AR_ANSI.

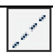
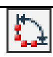
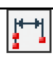
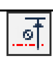
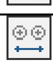
T-FLEX CAD dimensions are tied to straight construction and graphic lines and nodes, except for the radius and diameter dimensions, whose position is defined by the circle they dimension.

Applying Dimensions

To apply a dimension, use the command **D: Create dimension**:

Icon	Ribbon
	Draw → Title Block → Dimension
Keyboard	Textual Menu
<D>	Construct > Dimension

The user then gets access to the following options:

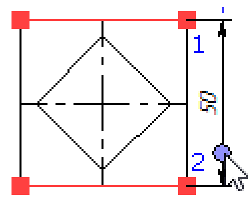
	<P>	Set Dimension parameters
	<Alt+P>	Copy Properties from Existing Element
	<N>	Select Node
	<L>	Select Line
	<C>	Select Circle
	<Y>	Create radial Dimension with stepped line
	<A>	Create Arc dimensions
	<Q>	Create Dimensions by one Image Line
	<F>	Angular Dimensions by Four Nodes
	<T>	Linear Dimensions by Three Nodes
	<O>	Set Cone Dimension
	<B>	Create Dimensions from one baseline
	<Ctrl+B>	Create Dimension Chain
	<S>	Create ordinate dimension
	<E>	Leader Dimension
	<X>	Dimensions from Axis
	<Shift+C>	Dimension between two circles

	<F4>	Edit Dimension
	<Esc>	Exit command

Upon calling the dimension creation command, one can click near any construction or graphic line. The line will be highlighted. Alternatively, point the mouse at a line and type <L>. In addition, one can select a node (the key <N>) or a circle (the key <C>).

Depending on what was selected at this step, different options are provided for further construction.

Dimensions between Two Straight Lines or between Line and Node



If the first selected element was a line, proceed with defining the second dimension reference element.

To construct a linear dimension, the second element can be another line parallel to the first one, or a node. To construct an angular dimension, select another line positioned at an angle to the first line.

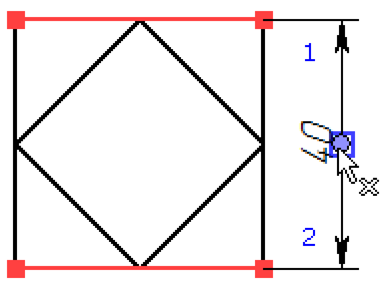
Upon drawing dimension between two lines, the system itself finds the nearest nodes lying on these lines and ties the origins of the extension lines to them. While doing it, however, there is always a possibility to reassign the nodes to which the dimension will be tied.

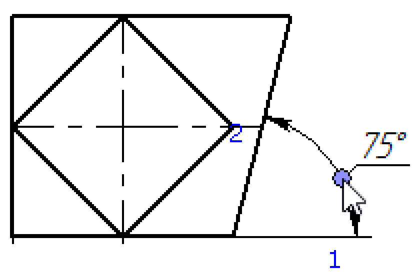
The following set of icons will be available in the automenu:

	<L>	Select Line
	<N>	Select Node
	<Esc>	Cancel selection

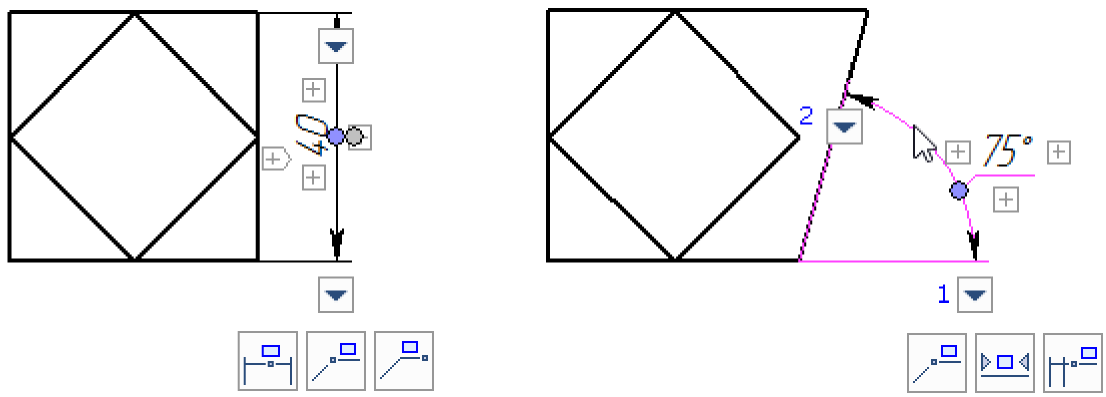
Obviously, the second dimension reference element can be selected using the common T-FLEX technique. One can click or type <L>, while pointing the mouse at a construction line. One can select a node using the option . This creates a dimension between the line and the node.

Once the second dimension reference element was selected, regardless of the selection technique, a dimension will start rubberbanding on the screen following the pointer movement. The start point and the end point of the dimension line will be marked by numbers: “1” – the start point of the dimension line, “2” – the end point of the dimension line.

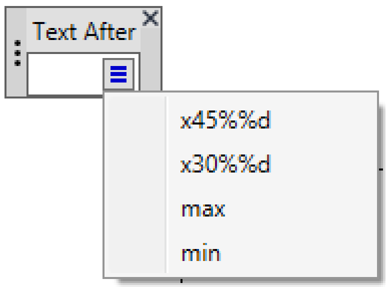




You may use special floating toolbar with options - icons and markers that allow changing dimension position, arrows style and dimension text.

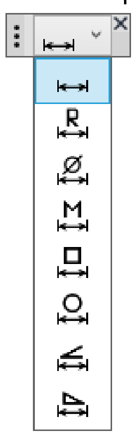


You can enter text **Before**, **After** and **Under** on the toolbars that appear after clicking on one of the icons. Drop-down lists includes standard texts **Before**, **After** and **Under.** Context menu appears, when you press right mouse button in the text field.

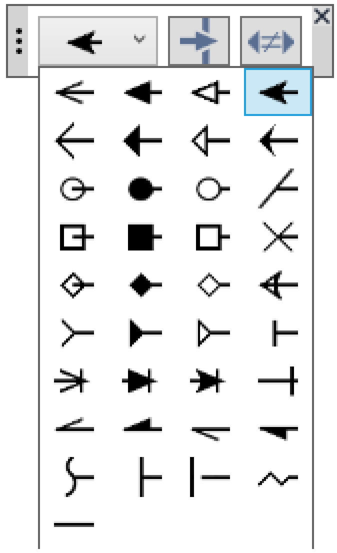


You can apply **Before** and **After** text insertion by pressing <Enter>. You can cancel text insertion using <Esc>.

You can select sign for the dimension value from the drop-down list after clicking icon.



You can set parameters for the arrows in the special toolbar that appears after clicking icon under the arrow.

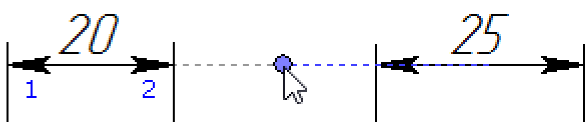


Option allows to clear background. Option enables/disables the same arrows for the dimension.

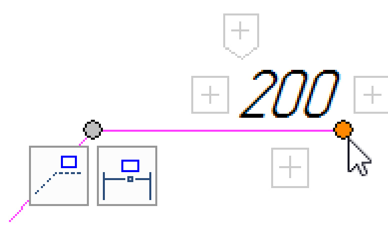
You can move toolbars. Place cursor on icon and move cursor, holding button.

Dynamic toolbar with **Place leader**, **Center dimension text**, **Dimension with jog** options appears under the dimension, when you place cursor on it.

To specify dimension location and to align it according to another dimension you can use the special control marker.



If dimension is on a jog, you can specify the jog length using special control marker.



Options **Hide jog** and **Remove jog** are available for the dimension with jog.

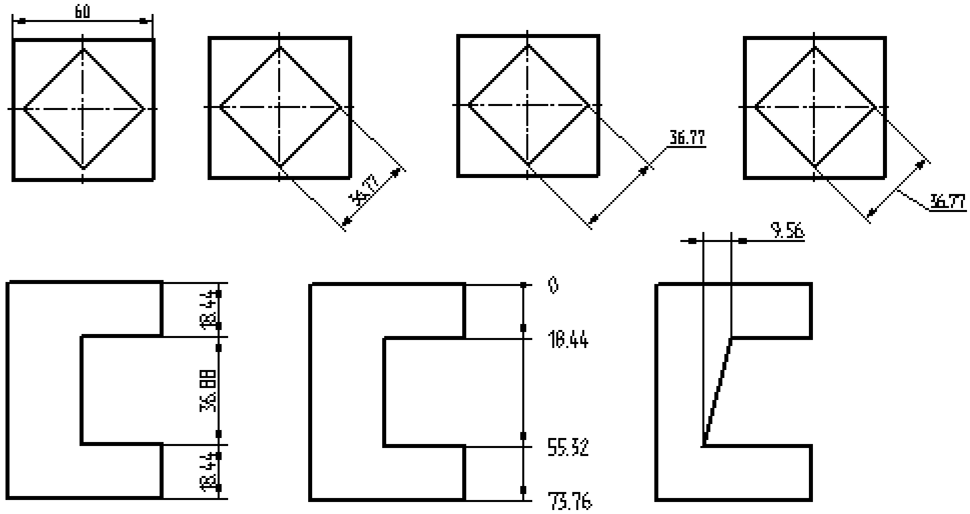
Option **Remove leader** is available for dimension on leader.

The new options that appear in the automenu, hint of further possible actions. This relates to either the linear or the angular dimension (in the case when the two selected lines intersect).

The available options after specifying the dimension references are:

	<P>	Set Dimension parameters
	<Alt+P>	Copy Properties from Existing Element
	<Shift+D>	Select Linked Dimension
	<Z>	Change leader line jog orientation
	<Z>	Change Dimension Orientation (for angular dimension)
	<Spacebar>	Place Dimension in absolute coordinates
	<T>	Tie Dimension to Node
	<J>	Center Dimension Text
	<D>	Change Sign
	<N>	Select insertion Node
	<M>	Change Dimension Type
	<K>	Break relations (available if dimension is tied to the node)
	<Esc>	Cancel selection

After selecting two lines to be dimensioned, one can click , while pointing the mouse at the desired position of the dimension. Before clicking, one can define parameters of the given dimension by calling the option , as well as specify position of strokes of the dimension entity. This can be done primarily with the options <Spacebar> and <T>. Presented below are several examples of dimensions that can be created between two lines or a line and a node. Parameter definition is described below, after the description of other dimension types.



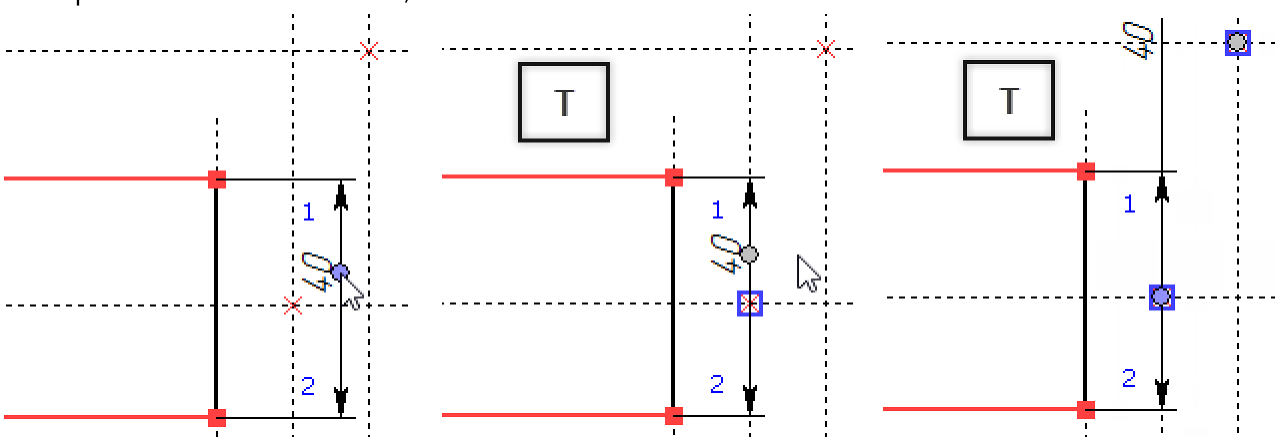
Let’s review the process of tying the dimension witness lines using the options <Spacebar> and <T>. This process is optional and necessary only when the dimension entity strokes must be strictly tied to the construction elements in the drawing.

Example of option **Tie Dimension to Node** usage:

Let's create a dimension by two lines and press <T> once. The dimension will be fixed in the position next to the node. After that we move cursor up and create a leader line extension.

> The **Center Dimension Text** option should be disabled.

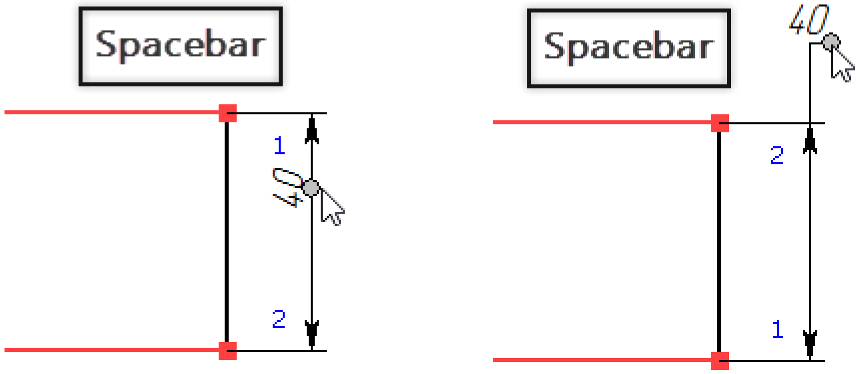
If we press <T> one more time, the leader will be tied to the nearest node.



Example of option **Place Dimension in absolute coordinates** usage:

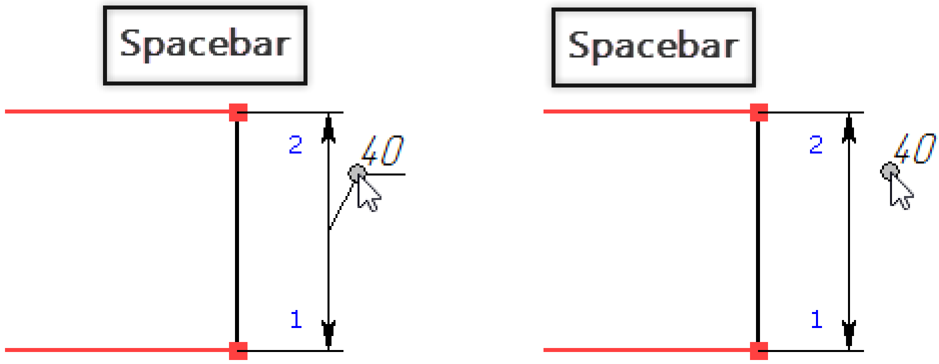
When you type <Spacebar> for the first time the dimension will be fixed at the specified distance from the object. If you move cursor up, create a leader line extension and press <Spacebar> again, a leader jog will be created.

> If the **Center Dimension Text** option is enabled, the second spacebar pressing will create a leader line extension.



The third subsequent use of the option <Spacebar> causes creation of the leader jog attached to the center of the leader line.

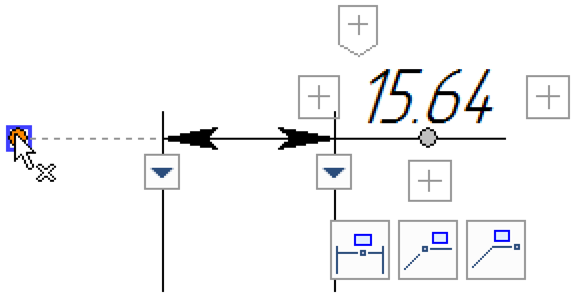
The fourth subsequent use of <Spacebar> will switch dimension into “invisible leader” mode. There will be no leader lines but text will remain and will behave like if it is located on a leader. This mode allows locating text in arbitrary place.

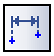


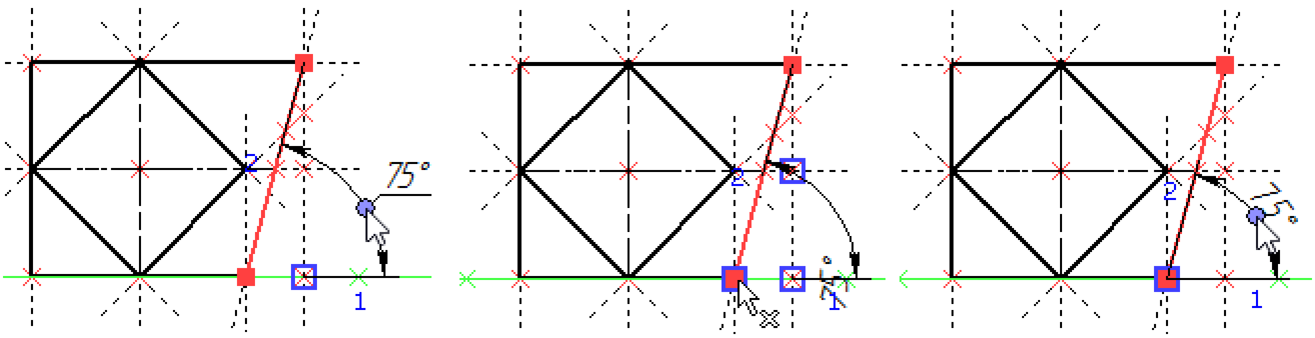
The fifth subsequent use of <Spacebar> reverts the dimension to the original state.

Note, that use of options <Spacebar> and <T> is recommended, when you want to strictly define the dimension position through parametric modifications of the drawing. When modifying positions of the nodes to which the dimension strokes are tied, the dimension position will be changing accordingly.

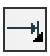
To break a tie to a node, use the option . If you click on the marker that indicates node to which the dimension is tied, the relation will be broken automatically and the option will not appear.



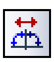
The option  allows explicitly assigning the tying nodes for dimension strokes created on the construction lines (by default, the system selects the node on the selected line nearest to the dimension position). Suppose, the lower witness line in the first drawing is tied to the default node. Let's specify another tying node using the option <N>. The modified dimension is shown on the right hand side diagram.

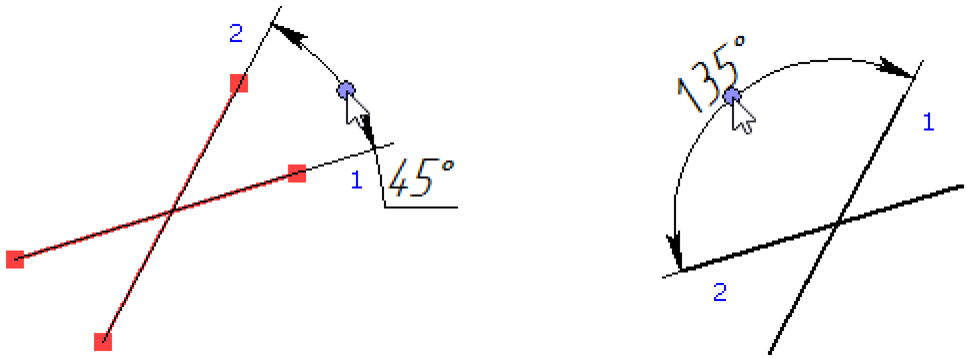


With the option  turned on, the dimension text will be centered between the witness lines. Without the option, the dimension text is positioned wherever the mouse click occurred.

The option  allows quickly changing the dimension prefix symbol (“R”, “∅”, “M”, “□”, “O”), without calling the dimension parameters dialog box.

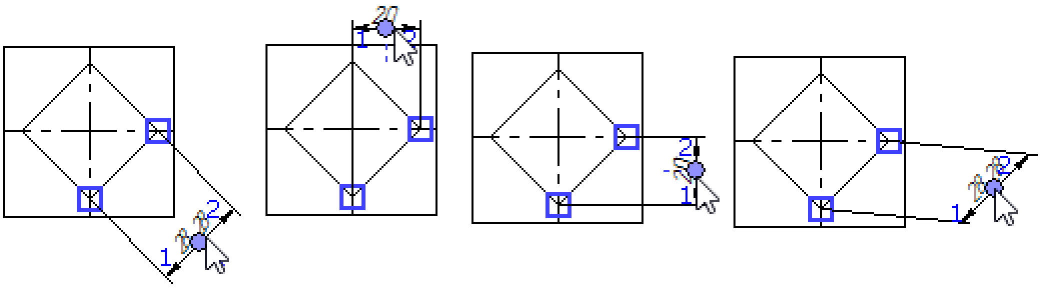
When creating a dimension with a jog, the option  (<Z>) will help changing the shape and position of the jog.

Pressing  (<Z>) results in a change of a quarter in which the dimension is drawn. In this case, the mouse should be pointing to the quarter where the dimension should be placed.

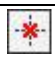


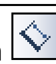
Dimensions between Two Nodes

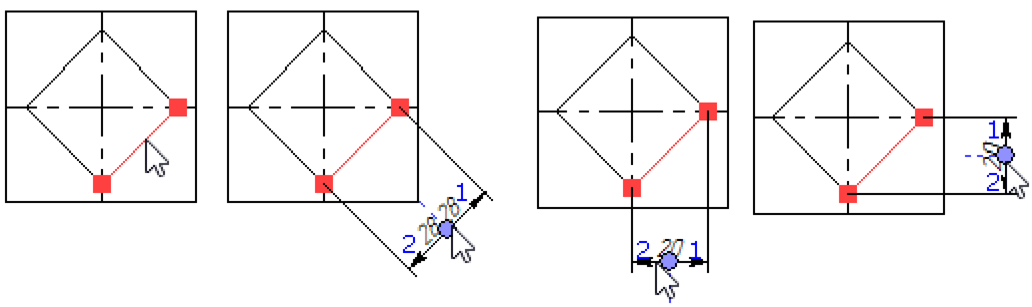
Creating a dimension between two nodes is mostly similar to creating a dimension between two lines. The exception is multiple possibilities for the witness line positions. These possibilities are shown on the diagram below.



To select nodes, use the option:

	<N>	Select Node

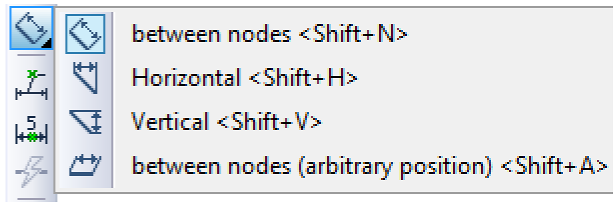
If two nodes to be dimensioned are connected by a graphic line, the option  is used. Upon calling the option, select the desired segment, and its end nodes will be automatically selected for dimension creation.



To toggle through the various types of dimensions between two nodes, the option <M> is provided in the automenu:

	<M>	Change Dimension Type

This option toggles the various dimension types. Besides, the desired dimension type can be selected from the pull-down list.

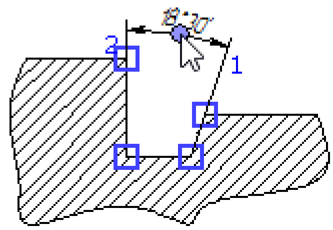


The parameters of the dimension being created can be defined using the option:

	<P>	Set Dimension parameters

Angular Dimension by Four Nodes

Placing a dimension by four nodes represents itself basically a variation of the angular dimension between two segments (lines). The lines between which the dimension will be drawn are defined by the end nodes.



For drawing the dimension by four nodes the following option is used:

	<F>	Angular Dimensions by 4 Points

After choosing this option in the automenu, the option for selecting the nodes appears:

	<N>	Select Node

Tooltips in the status bar show the order in which nodes are selected. The dimension will be tied to the first nodes of both lines.

After specifying all nodes, it is necessary to indicate location of the created dimension with the help of . While doing it, the following options will be available in the automenu:

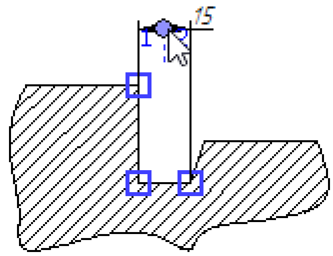
	<P>	Set dimension Parameters
	<Alt+P>	Copy Properties from Existing Element
	<Z>	Change leader line jog orientation
	<Shift+D>	Select Linked Dimension
	<Z>	Change Dimension Orientation
	<Spacebar >	Place Dimension in absolute coordinates
	<T>	Tie Dimension to Node
	<J>	Center Dimension Text

	<K>	Break relations (available if dimension is tied to the node)
	<Esc>	Cancel selection

The use of these options has been described above.

Linear Dimension by Three Nodes

Linear dimension by three nodes represents itself a variation of the linear dimension between a line (a segment) and a point. This means that the dimension will be drawn between the line defined by first two nodes (end points of the segment of the line) and the third node selected.



For drawing dimension by three points, the following option is used:

	<T>	Linear Dimension by Three Nodes

After choosing this option in the automenu, the option for picking the nodes appears:

	<N>	Select Node

Tooltips in the status bar indicate the node selection order. After specifying all nodes, it is necessary to indicate the location of the dimension being created with the help of . While doing it, the following options are available in the automenu:

	<P>	Set dimension Parameters
	<Alt+P>	Copy Properties from Existing Element
	<Shift+D>	Select Linked Dimension
	<D>	Change Sign
	<Spacebar >	Place Dimension in absolute coordinates
	<T>	Tie Dimension to Node
	<Z>	Change leader line jog orientation
	<J>	Center Dimension Text
	<K>	Break relations (available if dimension is tied to the node)
	<Esc>	Cancel selection

The use of these options was described above in the section “Dimensions Between Two Straight Lines or Between a Line and a Node”.

Creating Arc Length Dimension

To dimension the length of a circular arc, use the option . The following items become available in the automenu after selecting the option:

	<N>	Select Node
	<Esc>	Cancel selection

Dimensioning a whole arc starts with selecting an appropriate graphic entity.

In the case of dimensioning a portion of an arc or circle between two nodes, subsequently do the following:

1. Select the start node of the arc being dimensioned.
2. Select the end node of the arc being dimensioned.
3. Select the arc or circle passing through these nodes. An additional option is provided in the automenu:

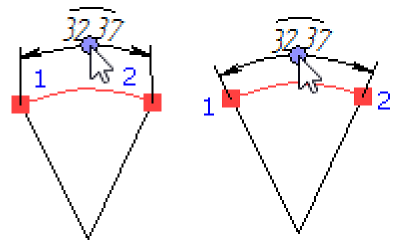
	<C>	Select Circle

After selecting the arc by any means, the automenu provides the options:

	<P>	Set Dimension parameters
	<Alt+P>	Copy Properties from Existing Element
	<Shift+D>	Select Linked Dimension
	<Z>	Change leader line jog orientation
	<Spacebar >	Place Dimension in the absolute coordinates
	<J>	Center Dimension Text
	<T>	Tie Dimension to Node
	<K>	Break relations (available if dimension is tied to the node)
/	<H>	Parallel/Radial Dimension Lines
	<Esc>	Cancel selection

As in the case of creating a dimension between two lines, the option <Spacebar> allows specifying the positions of dimension strokes. The option is used for tying the dimension to a node, while the option - for breaking the tie. To set parameters of the dimension being created, use the option . The option allows flipping the orientation of the dimension leader line jog.

With the option turned on, the dimension text will be centered between the witness lines. With the centering off, the dimension text is position wherever the mouse click occurred.



Additionally, one can choose between the types of the dimension witness lines (parallel or radial) by using the options / .

To complete a dimension creation, click its position by . If the dimension is already tied to a node, clicking simply confirms its creation.

Creating Dimension by Cone

A dimension by cone is created based on two non-parallel graphic lines. The selected lines are considered silhouette edges of a cone (cone projection). In this case, the dimension measures the distance between the ends of the selected lines, in the direction orthogonal to the cone axis. Thus, one can dimension a cone base on a projection view without creating additional nodes at the line ends.

To create a dimension by cone, use the option . After selecting the option, the following items are available in the automenu:

	<G>	Select Line
	<Esc>	Cancel selection

The dimension creation starts with subsequent selection of two non-parallel graphic lines. Once the lines are selected, the following options appear in the automenu:

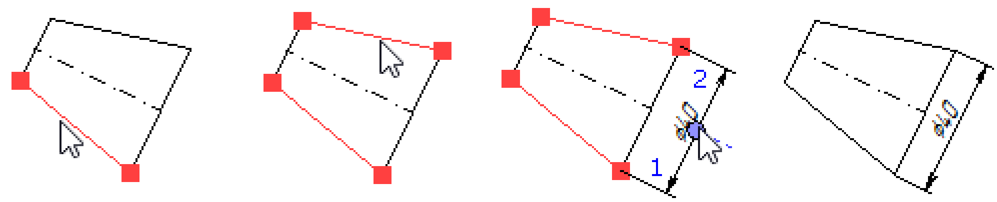
	<P>	Set Dimension parameters
	<Alt+P>	Copy Properties from Existing Element
	<Shift+D>	Select Linked Dimension
	<Z>	Change leader line jog orientation
	<Spacebar>	Place Dimension in the absolute coordinates
	<J>	Center Dimension Text
	<D>	Change Dimension sign

	<M>	Change Dimension type
	<T>	Tie Dimension to Node
	<K>	Break relations (available if dimension is tied to the node)
	<Esc>	Cancel selection

Line ends selection for attaching a dimension is done automatically by the system. However, if necessary, this can be modified using the option <M> () that helps quickly toggle through all possible pairs of line ends.

As in the case of creating a dimension between two lines, the option <Spacebar> allows specifying the positions of strokes of a dimension. The option is used for tying a dimension to a node, while the option – for breaking such a tie. The option serves for specifying dimension parameters.

The option allows flipping the dimension leader jog orientation, and the option – quickly changing the dimension prefix (the default symbol is "∅"). Centering dimension text is set by the option .



Dimensioning a Single Graphic Entity

To dimension a single graphic entity (a line segment or an arc), you do not have to call the option or . You can simply select a line segment or an arc by clicking . Then, while still holding down the left mouse button, slightly move the pointer. The command will then automatically assume the mode of creating a line segment length or an arc dimension.

Creating Dimension Chains

The option allows creating dimension chains for a group of parallel lines, as well as appending dimensions to already existing chains. Upon selecting the option, the following icons become available in the automenu:

	<D>	Select Dimensions in Chain
	<L>	Select Line
	<N>	Select Node

	<Bkspace>	Cancel last element selection
	<Esc>	Cancel selection

To create a new dimension chain, select subsequently the construction lines, the graphic lines or the nodes to be dimensioned. Complete the entity sequence selection by pushing the option:

	<E>	End Dimension Chain input

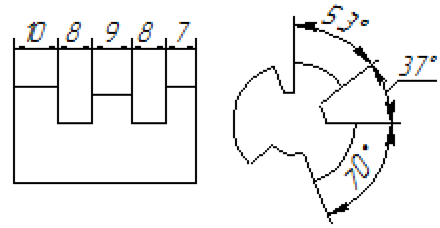
After that, the following options will appear in the automenu:

	<P>	Set Dimension parameters
	<Alt+P>	Copy Properties from Existing Element
	<T>	Tie Dimension to Node
	<K>	Break relations (available if dimension is tied to the node)
	<Esc>	Cancel selection

The option help tying dimension position to a node, while the option breaks this tie. To define parameters of a dimension being created, use the option .

To complete a dimension chain creation, specify its position by clicking . If the chain was tied to a node, then clicking simply confirms the creation.

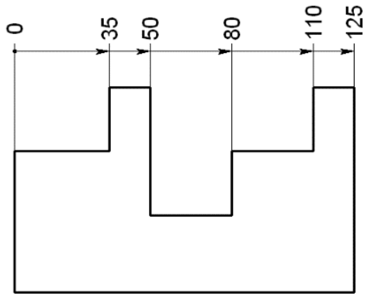
Note that each dimension in the chain referencing a pair of lines, is a separate entity and can have its own parameters.

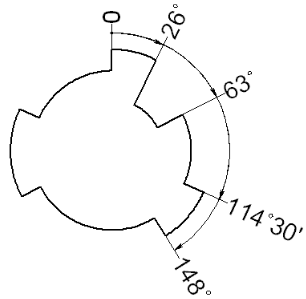


To add more dimensions to an existing chain of dimensions, upon calling the option select one of the dimensions in the chain to be appended. By doing so, you define the base for the dimensions to be created. The further steps will be similar to the previous case. The chains can have gaps. Should this be the case, select a line or node as the first entity of the chain continuation that is not referenced by the last existing dimension in the chain.

Creating Dimensions from One Baseline

The option serves for creating dimensions from one base. The creation procedure for this dimension type is mostly similar to the previous case (creating a chain of dimensions from the common base), except the number of additional features.



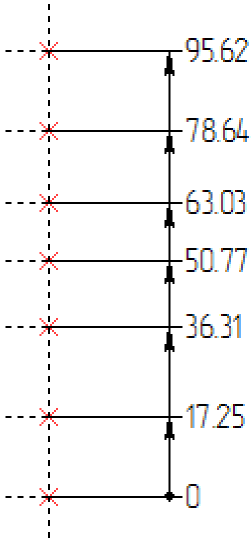


The following items are available in the automenu after selection of lines and nodes for the dimension creation and pressing :

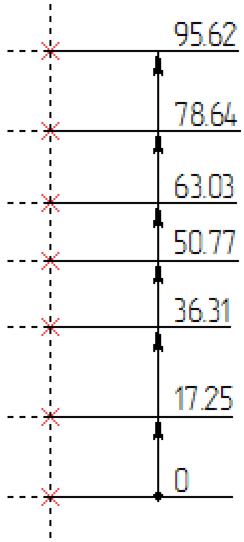
	<P>	Set Dimension parameters
	<Alt+P>	Copy Properties from Existing Element
	<Z>	Show/Hide zero at baseline
	<S>	Sign of dimensions direction
	<Space>	Leaders auto correction
	<M>	Change Dimension Type
/ /	<L>	Nominal near Jog/Nominal at Jog/Nominal at jog inside lines
	<T>	Tie Dimension to Node
	<K>	Break relations (available if dimension is tied to the node)
	<Esc>	Cancel selection

Option allows showing or hiding zero mark for the dimensions chain. When option is enabled, dimensions are created with negative nominal values. This option works only for dimensions that were created from top to bottom or from right to left.

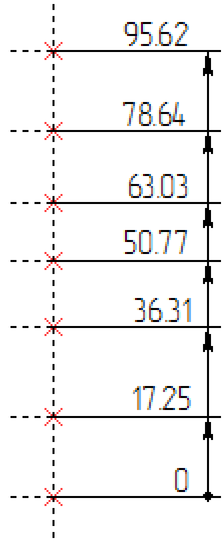
Option / / is toggled cyclically and allows to choose variant of placement for dimensional size: **nominal near jog/nominal at jog/nominal at jog inside lines.**



Nominal near Jog



Nominal at Jog

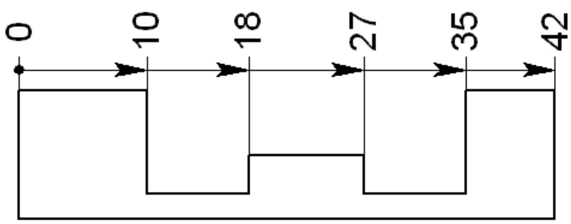


Nominal at jog inside lines

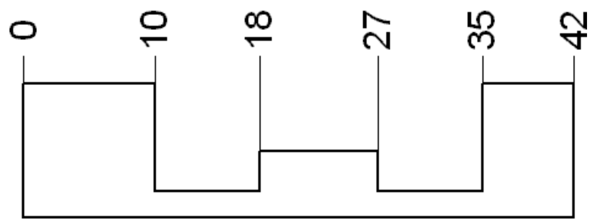
Dimensions from one base can be of two types: normal and ordinate. The dimension type can be modified either at the time of creation or at editing. This is done by the option:

	<M>	Change Dimension type

In the case of ordinate dimensions, only the witness lines are drawn, while the leader lines are not.



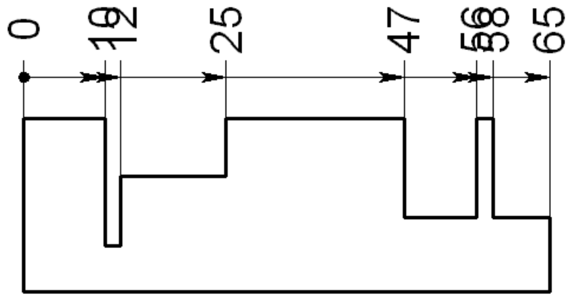
Normal dimension



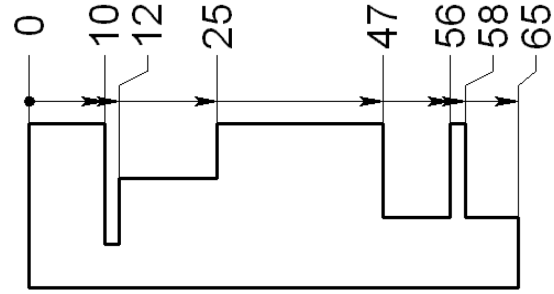
Ordinate dimension

	<Space>	Leaders auto correction

This option helps avoiding overlapping text of the dimensions from one base when those are placed near to each other. A jog is introduced in the dimensions that “creep” on the predecessors.

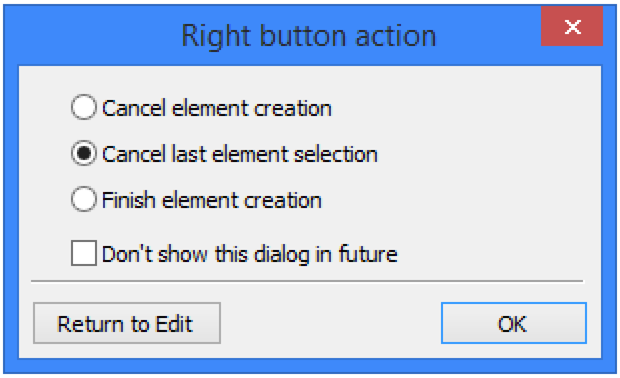


Autocorrection is off



Autocorrection is on

In the process of creating a dimension chain or base dimensions, pressing <Esc> or displays a dialog box for selecting the action to execute.

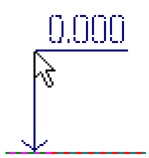


Creating Ordinate Dimensions (Level Markers)

To create a ordinate dimension, use the option . Upon calling the option, the following icons become available in the automenu:

	<D>	Select ordinate dimension
	<L>	Select Line
	<N>	Select Node
	<Esc>	Cancel selection

Creating a series of dimensions from the same base starts with selecting the base ("zero") dimension. To do this, upon calling the option simply select a horizontal ordinate or graphic line, or a node. As a result, a dimension starts rubberbanding on the screen, following the pointer. Meanwhile, the following options become available in the automenu:



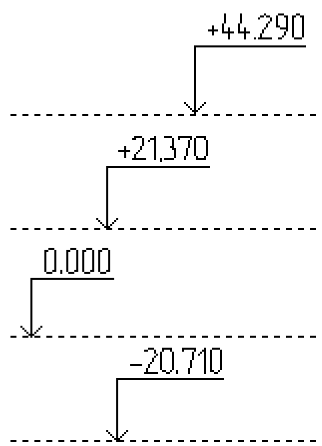
	<P>	Set Dimension parameters
	<Alt+P>	Copy Properties from Existing Element
	<Shift+D>	Select Linked Dimension

	<Z>	Change leader line jog orientation
	<T>	Tie Dimension to Node
	<K>	Break relations (available if dimension is tied to the node)
	<Esc>	Cancel selection

As in the case of a normal dimension creation, the option flips the orientation of the dimension leader jog. The option allows defining the positions of the dimension strokes by tying them to drawing nodes. In the letter case, use of the option <T> for the first time defines horizontal positioning, that is, the position of the vertical stroke of the dimension jog. The second use of the option <T> defines the dimension height, that is, the level of the horizontal stroke of the jog. To undo a tie, use the option . The dimension creation can be completed by pointing the mouse at the desired position and clicking .

The thus created dimension becomes the base dimension. Meanwhile, the option of creating ordinate dimensions stays active, with the automenu providing the options for a line and node selection. By selecting next horizontal line or node, you begin creation of another dimension relative to the base. It's creation steps are same as the described above.

To complete creation of a series of base dimensions, return from the option into the main command menu. On a subsequent call for the option, selecting a horizontal line or a node will create a new base, with the subsequently created dimensions referencing the new base.

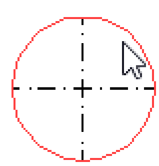


To create dimensions relative to an existing base, upon calling the option, select the desired base dimension, or a dimension referencing the base. All dimensions created thereafter will be referencing that base.

Dimensioning a Circle

When dimensioning a circle, there is only one reference, which is the circle being dimension.

After calling the command **D: Create dimension**, point the mouse at the desired circle and click or <C>. The circle will be highlighted, and a radius or diameter dimension will start rubberbanding with the pointer.

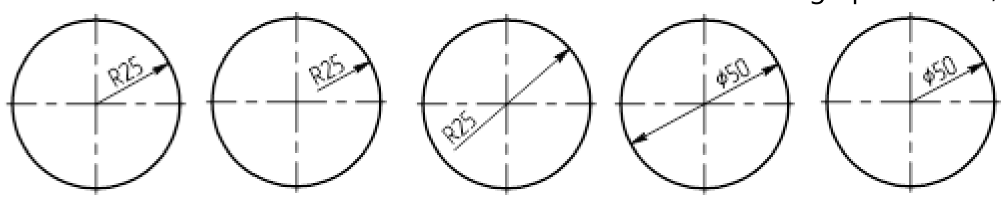


The following set of options will become available in the automenu:

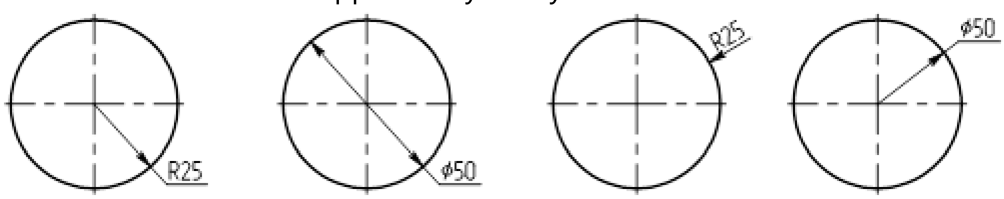
	<P>	Set Dimension parameters
	<Alt+P>	Copy Properties from Existing Element
	<Shift+D>	Select Linked Dimension
	<Z>	Change leader line jog orientation
	<Spacebar>	Place Dimension in absolute coordinates
	<T>	Tie Dimension to Node
	<D>	Change Dimension Symbol
/	<M>	Change Dimension Type
	<Esc>	Cancel selection

The type of the dimension being created is chosen in the drop-down list of the option **Change Dimension Symbol.** The following types are available , , , , .

The type of the dimension may be selected from the drop-down list **Change Dimension Type** , , . The list differ for the radial dimension and includes the following options: , , .



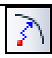
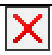
Once the necessary settings are defined, the created dimension will appear on the drawing. Follows are some types of dimensions on a circle supported by the system.



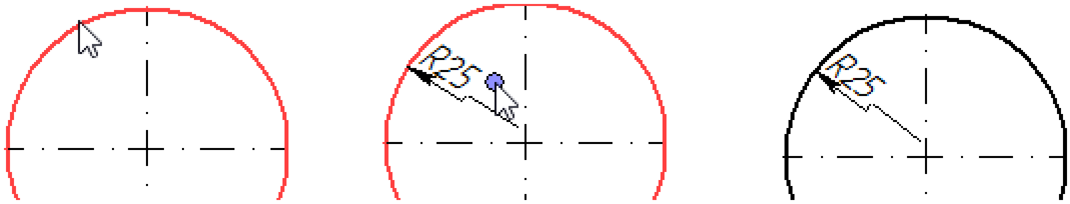
Creating Radial Dimension with Jog on Leader Line

For circles of a large radius, a radial dimension can be created with a jog on the leader line. To do this, use the option of the command top-level automenu. Upon calling the option, select the circle to be dimensioned. As a result, the select a circle will be highlighted and a dimension will start rubberbanding following the pointer. As the pointer moves, the end of the leader line and the whole dimension position will be adjusting accordingly. Meanwhile, the following options are available in the automenu:

	<P>	Set Dimension parameters

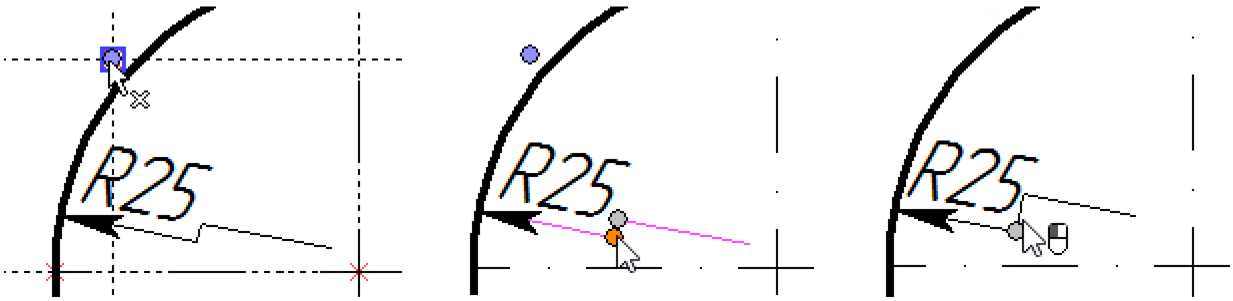
	<Alt+P>	Copy Properties from Existing Element
	<Shift+D>	Select Linked Dimension
	<K>	Break relations (available if dimension is tied to the node)
	<X>	Create Group of Radial Dimensions
	<Esc>	Cancel selection

To create the dimension, move the pointer to position the dimension as desired, and click . In this case, the default size and position of the dimension jog are set.



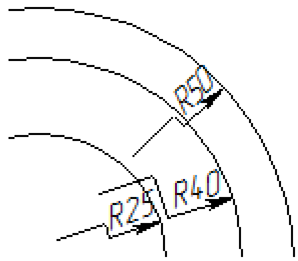
If necessary, the size and position of dimension components can be arbitrarily modified, including tying nodes. Use special markers for this purpose. Upon selecting the circle, a fixation marker appears. You need to select a node to which the dimension will be tied.

After the dimension position selection, you may change the dimension leader jog position. Select the special marker on the dimension leader jog, press and move the cursor. The dimension leader jog will follow the cursor along the dimension line.



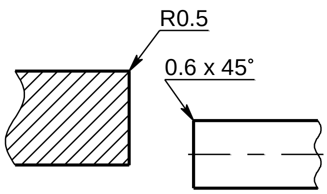
The additional option  turns on the mode of continuous radius dimension creation at a common fixing point. This mode can be used for creating radius dimensions, originating at the same point, on several concentric circles. With the option on, upon completing one dimension creation (the one on the first circle), the system waits for dimensioning another circle.

If the fixing point of the first circle dimension was tied to a 2D node, then the dimensions on all following circles are tied to this node automatically. However, if the first dimension was fixed arbitrarily, then a free node is created at that point, with all the rest of the dimensions tied to it.



Drawing Leader Dimension

For drawing dimension similar to one shown on the picture on the right, the option is used.



After calling this option, it is necessary to indicate the fixing point (2D node) for tying the dimension. In the automenu the following option for picking the node appears:

	<N>	Select Node

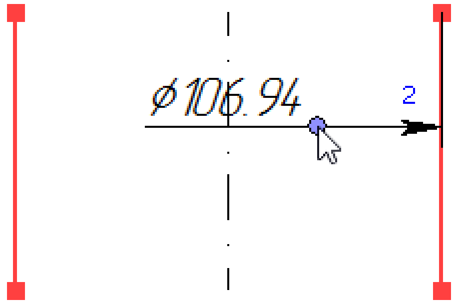
After specifying the fixing point, it is necessary to specify location of the leader of the dimension. Dimension parameters are specified manually in the dialog of the command's properties or with the help of the option .

In addition, the following options are available in the automenu:

	<P>	Set Dimension parameters
	<Alt+P>	Copy Properties from Existing Element
	<Shift+D>	Select Linked Dimension
	<Z>	Change leader line jog orientation
	<D>	Change Sign
	<T>	Tie Dimension to Node
	<Ctrl+T>	Link String to Node
	<Esc>	Cancel selection

Creating dimensions from Axis

Diameter dimensions, which are created from the axis, look like standard linear dimensions with diameter sign and doubled nominal values set in their parameters. There is no arrow near axis; it is replaced with the outline with length half of the font size.



The option is used for creation of such dimensions. After activation, you should define axis for the dimension creation. The following option is used for this purpose:

	<L>	Select Line

Straight construction or graphic line can be chosen as axis.

When axis was selected, you should define object for creation of the dimension. It can be a construction line, a graphic line or a 2D node. The following items become available in automenu:

	<L>	Select Line
	<N>	Select Node

After defining axis and object used for creation of dimension, it is necessary to set its location. Parameters of dimension can be set manually in the dialog box of the command properties or with the help of option

The following options are available in automenu:

	<P>	Set Dimension parameters
	<Alt+P>	Copy Properties from Existing Element
	<Shift+D>	Select Linked Dimension
	<Z>	Change leader line jog orientation
	<Space>	Place Dimension in absolute coordinates
	<J>	Center Dimension Text
	<D>	Change Sign (change prefix)
	<T>	Tie Dimension to Node
	<K>	Break relations (available if dimension is tied to the node)
	<Esc>	Cancel selection

Dimensions Between Two Circles

The command allows creating linear dimensions between two circles. After activation of the command, you need to select two circles between which the dimension will be created.

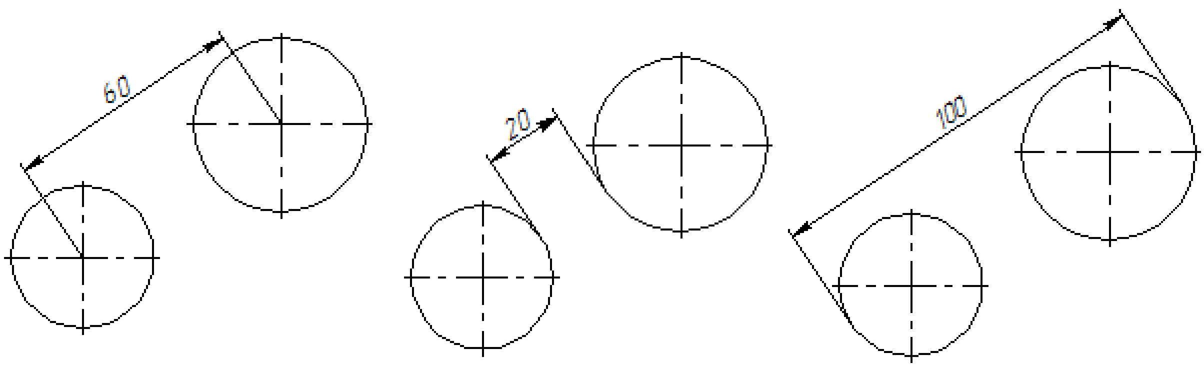
The following options are available in automenu:

	<P>	Set Dimension parameters
	<Alt+P>	Copy Properties from Existing Element
	<Shift+D>	Select Linked Dimension
	<Z>	Change leader line jog orientation
	<Space>	Place Dimension in absolute coordinates
	<J>	Center Dimension Text
	<D>	Change Sign (change prefix)
	<T>	Tie Dimension to Node
	<Esc>	Cancel selection

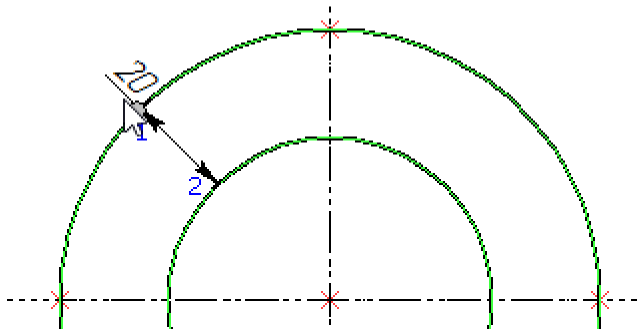
There are three dimension types for non-coaxial circles dimensions:

- ✓ Distance between centers
- ✓ Distance between the closest points of the circles
- ✓ Distance between the farthest points of the circles

Use option to change the dimension types.



Difference of the radii is measured for coaxial circles. You can use special control marker on the larger circle to move dimension.

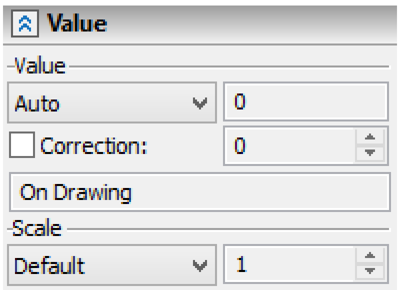


Dimension Parameters

Dimension parameters are defined in the command's properties window before finishing dimension creation (or editing).

Dimension parameters are arranged by several sections in the properties window according to the parameter type. Depending on the type of the created dimension (angular, linear, radius, ordinate, arc length), the sets of parameters in the sections may vary.

«Value» section



Value. This group parameters determine, how the nominal dimension value will be defined. You can select the following choices from the drop-down list:

Auto. The dimension value is calculated automatically based on drawing elements on which it is created. This allows a dimension to automatically change its value upon any modification to the drawing. The field on the right-hand side for the manual dimension value input is inaccessible.

Manual. The dimension value is defined by the user manually in the input field at the right of the drop-down list. Regardless of drawing modifications, such dimension value will stay unchanged. This option is used when you need to introduce a dimension value in the drawing, that does not match the calculated dimension value.

Manual with Corrections. The dimension value is specified manually by a user similar to the previous case. However, on the drawing the dimension value for the given dimension will be shown taking into consideration the specified scale and corrections (see below).

By Source Lines. This option is available for 2D dimensions, drawn on the elements of any associative copy. Upon selection of this option, the dimension value is determined by the original elements of the copy (if for such dimension the case "Auto" is selected, then the value of the dimension is evaluated on the basis of those elements of the copy to which the dimension is tied to).

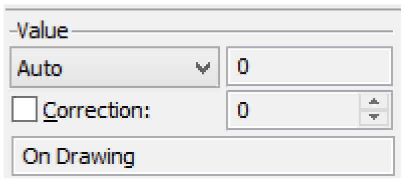
The option "By Source Lines" can be used, for example, for dimensioning on the lines of the associative copies (including the drawing views) without taking into account the scale of the copy.

None. The dimension value string does not show on the dimension.

From Lines. This option is available for dimensions created manually on the 2D projection lines (or their associative copies) corresponding to the thread (special 3D operation created with the help of the command **3AT: Create Thread**). Also, this option is available for 3D dimensions drawn on the threaded surfaces of the 3D model. When selecting this option, the text displayed on the dimension originates from the thread parameters (that is, the thread notation is displayed instead of the dimension value string).

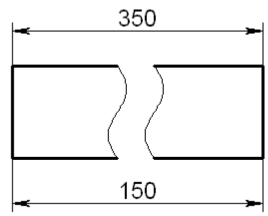
From 3D parent. This option is available only for dimensions created manually on the 2D projection lines (or their associative copies) and on condition that 3D dimensions are drawn on the corresponding to those lines faces, edges and vertexes of the original 3D model. Upon selection of this option, the text and parameters of the dimension will be inherited from the parameters of the 3D dimension.

The inaccessible for editing field found below the group "Value" is informative. It shows how the dimension has been created and how its value and parameters have been determined:



– *"On Drawing"* – standard 2D dimension drawn by nodes or by lines of a 2D drawing. The value of the dimension is evaluated by the geometry of the drawing or specified manually;
– *"On Drawing "* – standard 2D dimension drawn by nodes or by lines of a 2D drawing. The value of the dimension is evaluated by the geometry of the drawing or specified manually;
– *"On Copy: On Drawing"* – dimension is drawn on the elements of the associative copy of the nodes and the lines of a 2D drawing. Similar text string also appears if the dimension is drawn on the elements of the associative copy of the 2D projection lines, and for this dimension the field "Value" has an arbitrary value except "By Source Lines" (that is, the value of the dimension is determined either by the elements of the copy itself or specified manually);
– *"On Copy: On Projection"* – dimension is drawn on the elements of the associative copy of the 2D projection lines and for this dimension the field "Value" has the value "By Source Lines". In this case, the value of the dimension is evaluated by original objects of the copy, i.e. by the 2D projection lines;
– *"On Projection"* – dimension is drawn on the 2D projection lines. If on the faces of a 3D model corresponding to given projection lines there is a 3D dimension, and for the 2D dimension the field "Value" has the value "From 3D parent", then *"(*3D)"* is added to the information string;

– *"On Operation"* – this text string appears for 3D dimensions drawn on the vertexes, edges and faces of a 3D model (see the chapter "3D Annotations" of the manual on 3D modeling)
– *"Projected"* – this string appears for 2D dimensions created via the command of auto-dimensioning of 2D projections (see the chapter "3D Annotations" of the manual on 3D modeling).



Correction. This parameter defines the correction amount that will always be added to a dimension value. It is available only if the options **Auto** or **Manual with Correction** were chosen for specifying dimension value.

In the case when a scale is defined, the correction is added to the already scaled dimension value.

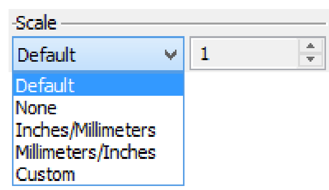
The info field at the end of the **Value** group indicates the position of the dimension placement: **On drawing** (for the dimensions created on common construction or graphic elements), **On projection** (for dimensions on 2D projections), **On operation** (for 3D dimensions).

The **Scale** group serves to define the value of the **scale factor**. The scale factor provides control over the dimension value. For example, suppose, some portion of a drawing was performed to a different scale. Since the units are the same throughout the drawing, you shall specify a scale factor for the dimensions in the said portion of the drawing. Then the dimension value will be displayed according to the specified scale (that is, multiplied by the specified scale factor). The value of the scale is not taken into account if the parameter "Value" was set the value "Manual".

You can select the desired option from the drop-down list:

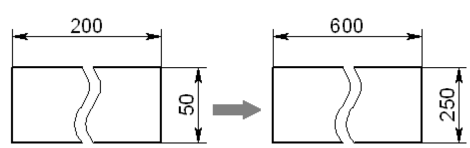
Default. The dimension value will be affected by the settings made on the **Dimensions** tab of the **ST: Set Document Parameters** command dialog.

None. The dimension doesn't have a scale factor.

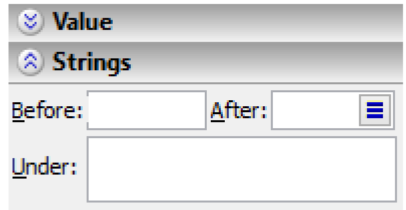


Inches/Millimeters, **Millimeters/Inches**. These are the standard scale factors introduced for the user convenience. When selecting one of those items, the appropriate scale factor is set automatically to make the conversion to other units.

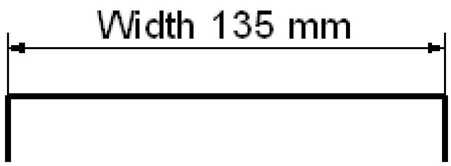
Custom. This option is used when it is necessary to set an arbitrary scale factor. The factor value is entered in the input field to the right of the drop-down list. For example, the case shown on the figure at the right has the, specified scale factor equal to "5" for both dimensions (the top dimension also having the correction of "100").



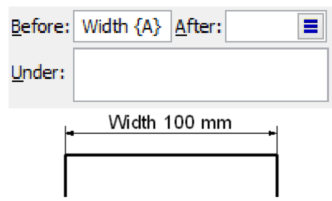
«Strings» section



This section of the properties window collects the parameters that serve to define the text that shall be placed before, after or under the dimension value string.



Those strings can be defined manually, or substituted by numerical or text variables. Besides that, one can use all capabilities of text line formatting described in the “Text” chapter.



To insert a variable into one of the dimension strings, you need to enter its name in the respective string, surrounded in braces. For example, if you need to make the “A” variable value appear in the dimension text on the drawing, enter *{A}* in the respective dimension string.

For example, in the string **“Before”** enter the desired text: “width {A} mm”

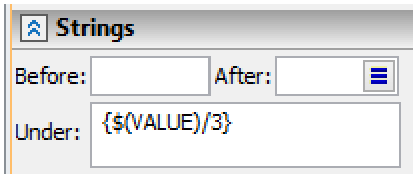
Note that the variable *“A”* is entered in braces. Suppose, its value is equal to 100. Set the parameter **“Value”** (described above) to the value “No”. As a result, the dimension on the drawing will appear as shown on the diagram at the right.

> For complete information on use of the variables, refer to the chapter “Variables”.

Besides, special symbols can be entered in the text strings via <Alt><F9>. To do this, click inside a text string input box and press <Alt><F9>. Then, select the desired symbol and hit <Enter>.

> Special symbols are various conventional textual and drawing notations. The symbols are represented by the textual font used in the parameter dialog box. On selecting a special symbol from the dialog box, it will be entered in the parameter string input box as a double percent character (%%) followed by the symbol code. However, upon placing the cursor into the parameter string which contains special symbols, a popup tooltip with a real image of the line contents will appear on the screen. On the drawing the special symbols will be also displayed precisely.
>
> Note that the special symbols can be used as part of any parameter, which is a textual string, for various system elements.

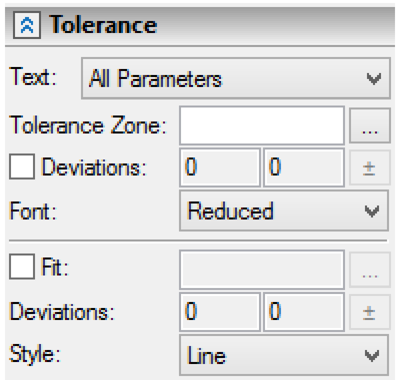
String parameters of the dimension may be set using other parameters of the same dimension. You need to use $(VALUE) record, where VALUE – name of the parameter (as it is called in the **PM: Measure** operation)



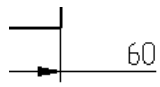
«Tolerance» section

This section of the properties window defines the tolerance zones and deviation limits.

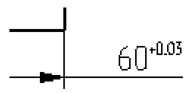
The "Text" parameter, which is coming first in the section, defines what parameters will be displayed together with the dimension nominal value:



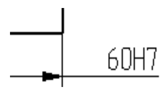
Nominal. Only the dimension value is displayed.



Nominal + Deviations. The deviation limits are displayed next to the dimension value string.

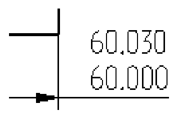


Nominal + Tolerance. The tolerance notation will be displayed next to the dimension value string.



All parameters. In this case, both the tolerance notation and the deviation limits are displayed.

Limits (ANSI standart). The dimension value string is composed of the two main values, each being the sum of the dimension value and the corresponding deviation limit.



Deviations can be defined manually or calculated automatically based on the specified tolerance zone. Automatic deviations calculation is used by default. The change from the manual setting to the automatic is done by toggling the **Deviations** flag (when the flag is off, the deviations are calculated automatically, when it is on – those are defined manually).

When using the automatic mode of calculating tolerances, you just need to specify the tolerance zone. Please note that the values of the calculated deviations depend on the measurement units defined in the

ST: Set Document Parameters command menu. Automatic calculation of deviations works only for the **millimeters** or **inches** settings.

When the dimension value is modified, the deviations populated from tolerances will be adjusted automatically.

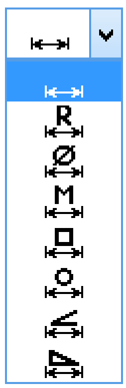
> A variable can be entered in place of a deviation value just like in any other numerical field.

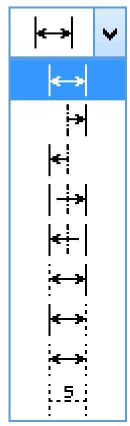
Various standards set the different requirements to the font size used to display value deviation limits in the drawing. Two choices are provided: **reduced height**, which is half the font size, and **full-height**. To choose the desired one, use the "Font" parameter.

Fit parameters are defined similar to tolerance parameters.

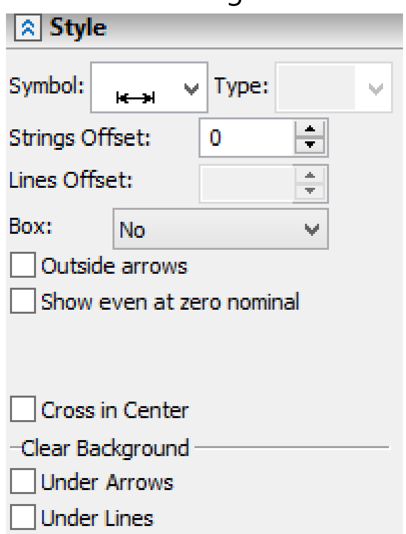
«Style» section

Sign. For a linear dimension, this parameter defines a special symbol to be displayed before the dimension value. This is necessary to create, for example, a radius, diameter or thread dimension.





This parameter is not available for the circular arc length dimension.

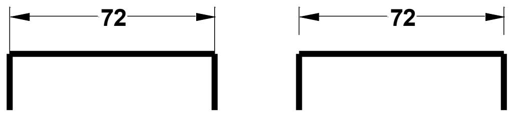


Type. This parameter is required if you need to have a linear dimension without extension (witness) lines (this is often used when creating dimensions from a centerline). This parameter can also be used to create one-sided dimensions (used for dimensioning on cut views or for dimensions on large-size parts).

Strings offset. This parameter defines the distance by which the dimension value string and the string beneath the dimension will be separated from the dimension line or from the leader jog (by default = 0).

Lines offset. This parameter defines the offset of extension (witness) lines of a dimension from the object when creating dimensions per the ANSI and AR_ANSI standards. Whenever this parameter is not defined, the offset amount will depend on the size of the dimension line arrows.

The diagrams at the right correspond to the values of this parameter equal to *"0"* and *"5"*.



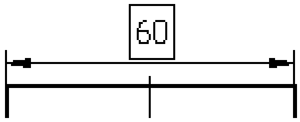
Box. You can choose one of the following options from the drop-down list.

No. The dimension text will be displayed without box.

Square. The dimension text is outlined by square box.

Rounded. The dimension text is outlined by rounded box.

You can outline only selected part of the dimension text using %%R.



Outside Arrows. When this flag is set, the dimension arrows are always drawn outside of the extension (witness) lines. When the flag is disabled (the default), the arrows position is determined automatically depending on the distance between the extension lines of the dimension. If arrows don't fit inside, then those will be automatically switched to outside.

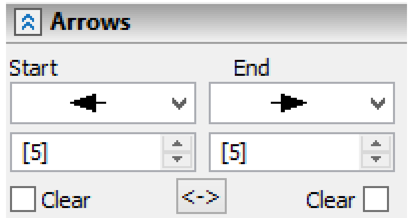
Show when zero value. When this parameter is enabled, dimension will be displayed on the drawing even when its value is zero.

The "**Clear Background**" group provides controls for the dimension display mode in which drawing portions are erased around the dimension strokes:

Under Arrows. Turning on this item erases the drawing portions under and around the leader lines and leader arrows (at the distance equal to the thickness of the main continuous line).

Under Lines. Turning on this item erases the drawing portions under and around the witness lines (at the distance equal to the thickness of the main continuous line). In the case of the radial dimensions, the background is also erased under the cross marking the circle center.

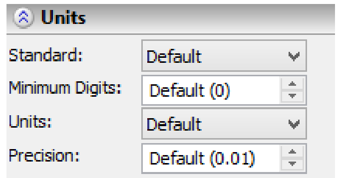
«Arrows» section



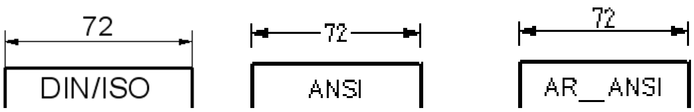
In this section you define the type and size of arrows on either end of a dimension line. The value in square brackets means the arrow size will be per the value defined on the **Lines** > **Arrow (End) Size** tab of the **ST: Set Document Parameters** command dialog.

Button <-> toggles start and end arrow head parameters.

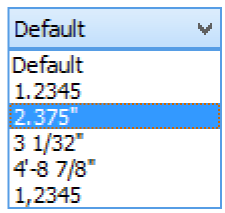
«Units» section



Standard. This parameter allows selecting the dimension display standard: ANSI, AR_ANSI, ISO or Default. When the "Default" value is used, the dimension standard is copied from the parameter value in **Dimensions** > **Standard** of the **ST: Set Document Parameters** command.



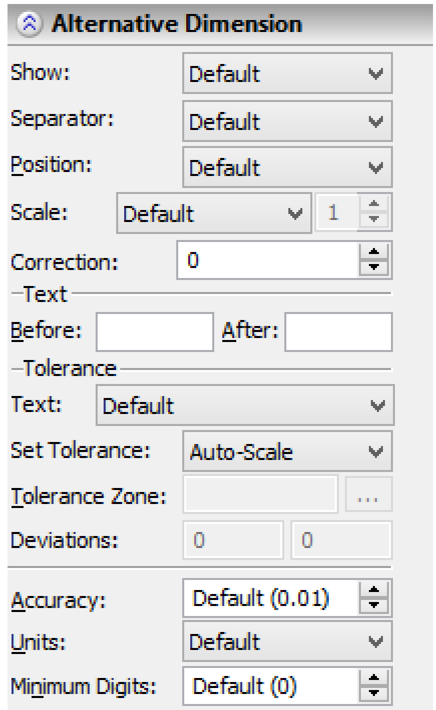
Minimum Digits. This parameter determines the minimum number of decimal digits displayed in a dimension. For example, if the value *"3"* is specified, then the dimension *28.5* will be displayed as *28.500*. If the "Default" is set, then the value is taken from the **Dimensions** tab of the **ST: Set Document Parameters** command.



Units. This defines the units in which the dimension value is displayed. Mostly, this item is important for inch dimensions. When the "Default" value is used, the dimension is displayed in the units defined in the **ST: Set Document Parameters** command.

Precision. Sets the rounding accuracy of dimension values. The accuracy "*0.01*" means the dimension values will be rounded to the second decimal digit. For example, if there is a dimension *28.4482*, and the accuracy is *0.01*, then the value *28.45* will be displayed in the drawing. If this item is set to **Default**, then its setting is defined on the **Dimensions** tab of the **ST: Set Document Parameters** command dialog.

«Alternative dimension» section



Show. This parameter determines the presence or absence of an alternative dimension in the drawing. When the parameter is set to **Default**, then its setting is defined on the **Alternative dimensions** tab of the **ST: Set Document Parameters** command dialog.

Separator. This parameter sets the appearance of separators that are used in the drawing to separate the alternative and the main dimension values:

Default. In this case, the separator assumes the type defined in the command **ST: Set Document Parameters.**

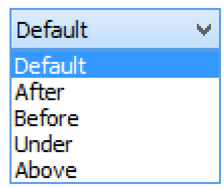
None. The alternative dimension text will not be separated.

[Brackets]. The alternative dimension text will appear in brackets.

{Braces}. The alternative dimension text will appear in braces.

(Parentheses). The alternative dimension text will appear in braces.

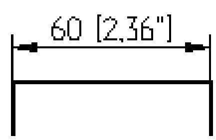
Position. This parameter defines the mode of displaying the alternative dimension value in the drawing at a relative location of the main dimension value. According to the choice made, the alternative dimension value will appear in the drawing "After", "Before", "Under" or "Above" the main dimension value string. If the parameter is defined as **Default**, then its value is assumed from the **Alternative dimensions** > **Location** setting of the command **ST: Set Document Parameters.**



Scale. this parameter defines the scale factor of an alternative dimension (which is fully analogous to the scale factor of the main dimension value).

Why are **Alternative scale** and **Scale factor,** respectively? Suppose, dimension values need to be created in two measurement systems at the same time, inches and metric. For this purpose, a special expression is introduced in the system - *#DIM#*. If this expression is used in any of the dimension lines (strings), then a dimension value will appear instead of it in the drawing, which will be multiplied by the alternative scale factor.

To follow the example on the right, set the **Alternative scale** to **Millimeters/Inches** and enter the following text in the **Text** > **After** field: *[#DIM#%%119]*.



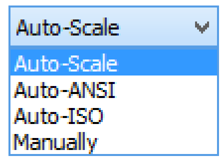
Correction. Parameter defines correction of the alternative dimension value compared to the calculated value. Correction is calculated similarly to the nominal value: displayed value = calculated value*scale factor + correction.

The **Text** group of parameters allows defining the text that will be displayed before or after the dimension value of an alternative dimension. Such strings can be entered manually or substituted by numerical or text variables. You can familiarize yourself with this capability in detail by reading the description of the counterpart parameter in the **String** section. If no parameter values are set in this group, then those are taken from the **Alternative dimensions** tab of the **ST: Set Document Parameters** command.

The **Tolerance** group of parameters allows defining tolerance zones (ranges) and value deviation limits of alternative dimensions:

Text. This parameter defines the included members of the dimension value string of an alternative dimension (the dimension value only, the dimension value with tolerance, etc.). A value can be selected from the list. The **Default** setting means the appearance of an alternative dimension will be determined by the **Text** parameter setting in the **Tolerance** section.

The tolerance zone is defined by the same-name parameter. The value deviation limits can be either defined manually (in the respective input fields) or calculated automatically. The method of defining deviations is controlled by the **Set Tolerance** parameter:



Auto-Scale. The deviation limit values are calculated by the tolerance zone of the main dimension (while accounting for the measurement units of the alternative dimension – millimeters or inches). The definition of the tolerance zone value of an alternative dimension does not affect the calculation of its value deviation limits.

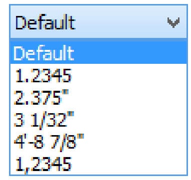
Auto-ANSI. Deviations are calculated by the defined tolerance zone of an alternative dimension per the ANSI standard.

Auto-ISO. Deviations are calculated by the defined tolerance zone of an alternative dimension per the ISO standard.

Manually. The deviation values are defined by the user. Variables can be used instead of deviation values.

When using the automatic modes of calculating deviations (Auto-Scale, Auto-ANSI or Auto-ISO), you just need to define the tolerance zone. The deviations will be calculated automatically.

Accuracy. Defines the rounding precision of the dimension values of the alternative linear dimensions. For example, the accuracy *0.01* means the dimension values will be rounded to the second decimal digit. Accuracy *0* means no rounding. If set to “Default”, the accuracy assumes the settings from the **Alternative dimensions** tab of the **ST: Set Document Parameters** command.

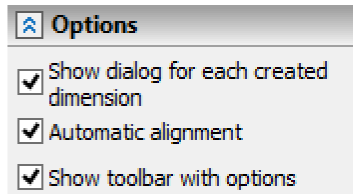


Units. Defines the way of displaying the dimension values of alternative linear dimensions. This item is primarily used when working in inches.

Just as in the previous case, the **Default** setting means the dimension will be displayed in the units defined on the **Alternative dimensions** tab of the **ST: Set Document Parameters** command.

Minimum Digits. This parameter determines the minimum number of decimal digits that is displayed for the alternative dimension value (similar to the "Minimum Digits" parameter of the main dimension value). If "Default" is set, then the value is taken from the **Alternative dimensions** tab of the **ST: Set Document Parameters** command.

«Options» section

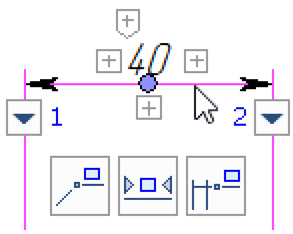


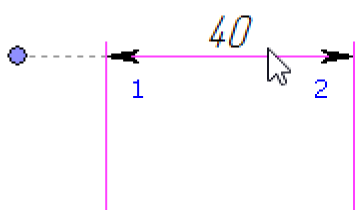
The section contains only one auxiliary parameter– **Show Dialog for each Created Dimension**. If this parameter is enabled, then the dimension parameters dialog will automatically appear after defining the dimension position in the dimension creation command (the option ).

This mode allows working in the same way as in previous versions of T-FLEX CAD – by specifying the dimension position in the drawing first, and then defining its parameters.

Automatic alignment. If the option is enabled, the system automatically places the dimension number and the arrows outside when the dimension size is reduced to certain values.

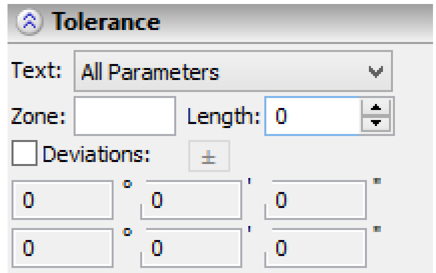
Show toolbar with options. If the option is enabled, additional markers and options for will appear near dimension when you put cursor on it.



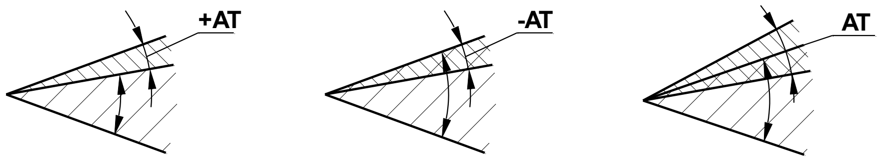


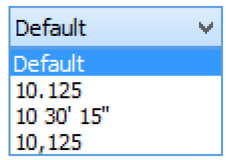
Special about Defining Angular Dimension Parameters

Linear and angular dimension parameters are mainly same. Described below are the existing differencies.



The set of tolerance parameters for angular dimensions is different from the respective parameters of other dimensions. Since angular dimension tolerance calculation depends on the length, the parameter **Length** is provided for angular dimensions. When calculating deviations automatically, the following formula of defining the tolerance zone is used: *+AT8*, *AT8* or *-AT8*, where *8* is the tolerance grade, while a signed AT defines the tolerance type.





The deviations are calculated in degrees, minutes and seconds, respectively. The angular dimensions use special units for displaying the dimension value.

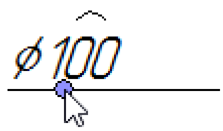
For angular dimensions you cannot specify an alternative scale, nor any other parameters related to an alternative dimension.

When you create angular dimensions, the **Text along the line** option appears. If the option is enabled, the text on the angular dimension will not be drawn along the arc, but along the tangent to it.

Special about Defining Parameters for Dimensions on Circles

In the **Style** section you can enable the display of a small cross at the center of the circle on which the dimension is created (the **Cross in Center** parameter). This is required by some standards. Otherwise, the parameters for circles correspond with the parameters for linear dimensions.

When you create dimension on the circle the **Arc Symblol** option appears.



Special about Defining Parameters of Ordinate Dimensions

Ordinate dimension parameters are mainly same as those of linear dimensions. Described below are the existing differencies.

The ordinate dimensions do not have the parameters to define tolerances and deviation limits.

In the **Style** section there are only the parameters, whose status is relevant to an ordinate dimension. Besides, there are the following additional parameters:

> **Leader Line**. Controls the leader extension creation between the dimension and the attachment node.

Show "**plus**". Controls the display of the "+" sign in dimensions with positive offset from the base dimension.

Just as for angular dimensions, there is no alternative scale for ordinate dimensions, nor any other parameters related to an alternative dimension.

Working with the dimension parameters dialog

You can also define dimension parameters in the parameters dialog called by the automenu option:

	<P>	Set Dimension Parameters

The parameters located on the tabs of this dialog duplicate the parameters in the properties window. Besides that, the parameters dialog has a number of additional settings. First of all, that's the system-wise parameters: level, layer, priority and color. The parameters dialog also has an additional tab that contains the font settings. This tab is standard for various system elements (dimensions, annotation leaders, roughness symbols, GD&T formlimits). It allows defining all necessary font settings used to display the dimension value string. If the value of some of the parameters on the tab is set as "Default", then it will be taken from the **Font** tab of the **ST: Set Document Parameters** command dialog.

Parameters for New Dimensions (Default Parameters)

The default parameters that will be applied to all newly created dimensions can be defined in a number of ways.

First of all, those can be defined within the parameters dialog (the option). To do this, call the dialog before creating a dimension. The parameters defined for the new dimensions will be copied to the parameters set of each created dimension.

You can also save the parameters defined when creating (or editing) a dimension, as the default parameters, by clicking the button in the command's properties window.

Note, that the tab **Dimensions** of the command **ST: Set Document Parameters** defines only those of the described parameters, that have the default option. As a rule, all dimensions should appear consistently. Therefore, a good strategy is defining their appearance in the command **ST: Set Document Parameters**, while using the default settings for the parameters of a particular dimension. This allows instantly changing appearance of all dimensions, if necessary.

> Besides the described parameters, the command **ST: Set Document Parameters** defines two more parameters. **Tolerance Grade** defines the threshold precision, up to which the dimension values are displayed on the drawing. This means, for dimensions, whose tolerance grade is less or equal to the specified, only the nominal values will be displayed.

The **Symbols** in the **ST: Set Document Parameters** command dialog provides for defining codes of the selected special font, corresponding to the symbols diameter, degree, and "±" sign. This can be helpful when exporting files, and when using fonts that use different codes for these symbols.

Copying Parameters from Existing Dimensions

Parameters of the dimension being created can be quickly copied from an already existing dimension. To do this, use the option:

	<Alt+P>	Copy Properties from Existing Element

> This option is available in the command automenu prior to creating the dimension or during the creation process (before selecting the dimension placement on the drawing).

After calling the option, simply pick the dimension whose parameters are to be transferred on the new dimension. The parameters will be copied that are common for both the selected dimension and the dimension being created.

To make the copied parameter values assigned to all newly created dimensions, before selecting the source dimension activate an additional option:

	<S>	Set Properties as Default

With the option active, the copied parameters will be saved as default parameters.

> This option simplifies creation of dimensions with identical parameters. However, it does not allow to copy specific parameters or parameters from an object of a different type. In such cases, more convenient could be using the general mechanism of editing element parameters in the property window.

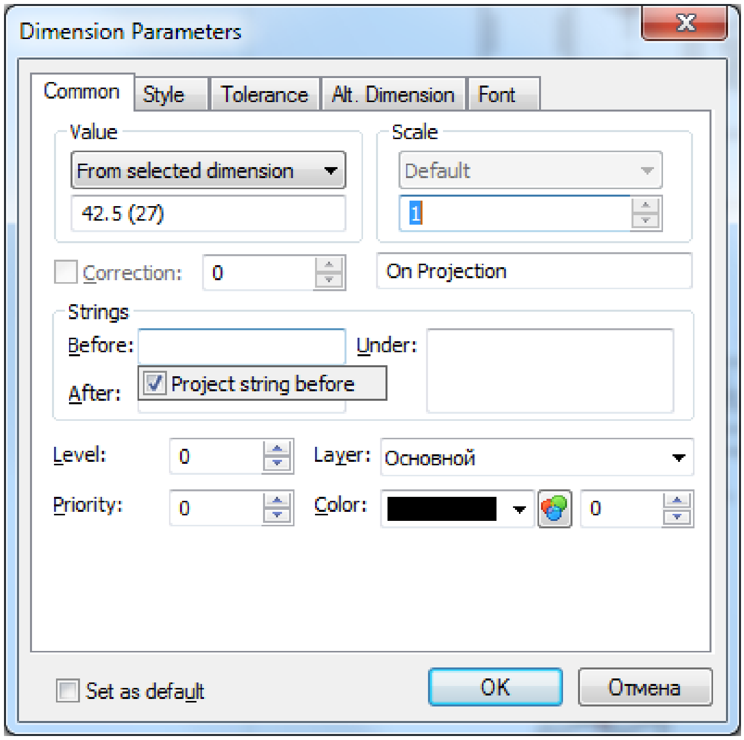
Setting link of dimension with another dimension

For any type of dimension you create, it is possible to link it with the other already existing dimension by means of the following automenu option:

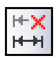
	<Shift+D>	Select linked dimension

After calling this option, you must specify the dimension which parameters will be used for the dimension being created.

When setting such link, parameters of "linked" dimension, for example, the nominal value, other display options, will be taken from the original dimension. List of "linked" parameters can be controlled in the properties dialog by selecting corresponding values for the nominal mode - “Auto”, “Manually”, “Manually+Corrections”, “From selected dimension”. If you do not want to display the content of "Before", "After" and "Under" strings of the linked dimension, it is necessary to remove the flag to "Project string ...". The flag appears when you put the mouse cursor over a field of the corresponding parameter.



To break the link between the dimensions you can use option:

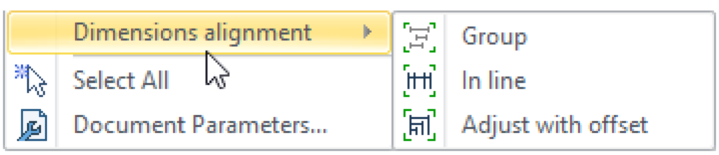
	<Shift+K>	Break link with dimension

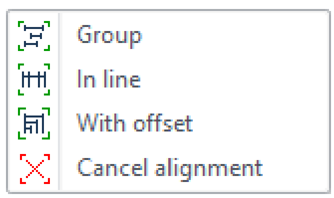
Option is available only for the linked dimensions.

This functionality can be used, for example, to display dimensions on a simplified view with the same nominal values and other parameters as on the original accurate view.

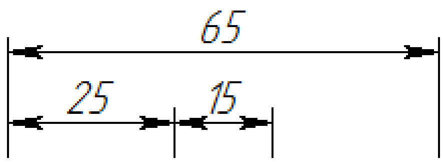
Dimensions Alignment

The options group **Dimensions Alignment** is available in the context menu when you select several dimensions. The options allow associating the selected dimensions to move them together on the drawing.





The **Group** option creates group of dimensions. When you move one of the dimensions, the rest dimensions also move, maintaining its position, retaining their original position relative to each other.



The **Delete Alignment** option appears for grouped dimensions. The option cancels the alignment of all dimensions in the group.

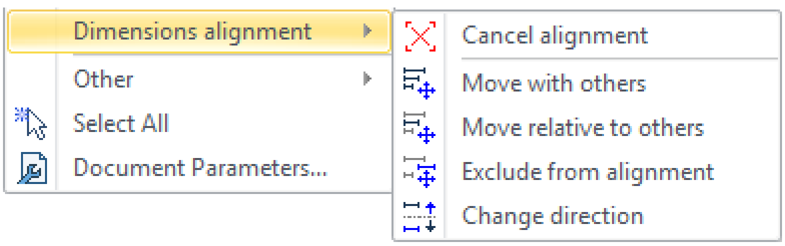
The **In Line** option aligns all dimensions in one line. The option allows you to quickly build a chain of dimensions.

The **Adjust with offset** option allows you to align the dimensions vertically in accordance with the specified spacing.

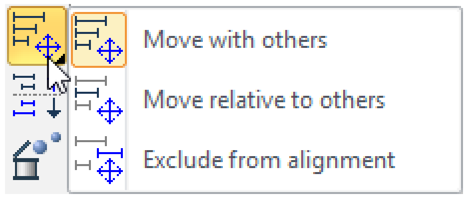
The spacing between dimensions is specified in the **ST: Set Document Parameters** on the **Dimensions** tab using **Alignment spacing** parameter. The parameter specifies the spacing for newly created groups of dimensions and does not affect spaces of the already created groups.

> When you create a new dimension above or under another dimension, a snap in accordance with a predetermined alignment spacing is active.

The following options are available in the context menu for each of the dimensions in the group:



The same options are in the automenu of the **Edit** command for the dimension.

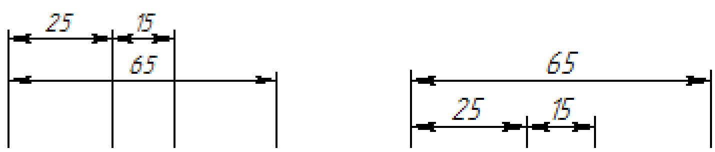


Move with others. The selected dimension is moved with the other dimensions in the group. The option is enabled by default.

Move relative to others. After activation of this option, the selected dimension can be moved relative to other dimensions in the group. After that, the dimension will move with other dimensions again, but the new position will be preserved.

Exclude from alignment. The selected dimension is excluded from the alignment.

Change Direction. Changes the position of the dimensions relative to the selected dimension. If the remaining dimensions in the group are located above the selected dimension, then they will be located below after activation of the option and vice versa.



Editing Dimensions

Dimension editing is done via the command **ED: Edit Dimension**:

Keyboard	Textual Menu	Icon
<ED>	Edit > Draw > Dimension	

Upon calling the command, the following options become available:

	<Enter>	Select dimension
	<*>	Select All Elements
	<Esc>	Exit command

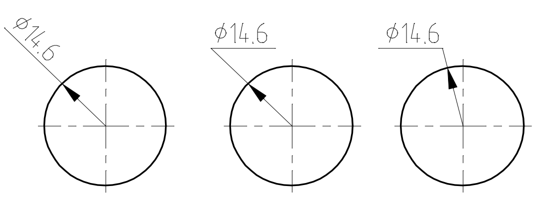
Select a dimension for editing by pointing and clicking the mouse. That highlights the dimension. This dimension parameters will be displayed in the properties window. Meanwhile, the automenu offers the following options:

	<P>	Set selected Element(s) parameters
	<Alt+P>	Copy Properties from Existing Element
	<O>	Create Name for Selected Element
	<Shift+D>	Select Linked Dimension

	<Shift+K>	Break Link with Dimension
	<Z>	Change leader line jog orientation
	<Z>	Change Dimension orientation (for angular dimension)
	<Spacebar >	Place Dimension in the absolute coordinates
	<J>	Center Dimension Text
	<D>	Change Dimension Symbol
	<T>	Tie Dimension to Node
	<Ctrl+T>	Link String to Node
	<N>	Select insertion Node
	<W>	Move dimension
, /	<M>	Change Dimension type
/	<H>	Change dimension symbol
	<K>	Break relations
	<D>	Change dimension type
	<I>	Select Other Element
	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

Availability of some of the above options in the automenu depends on the ways of creation and the type of the selected dimension.

The selected dimension can be moved, tied to other drawing elements or have its parameters modified, with respect to the original settings. To do this, select the appropriate option in the automenu.

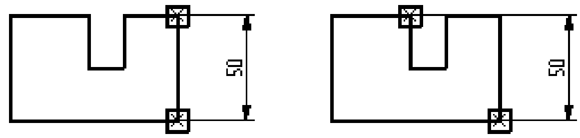


The option allows you to modify orientation of the leader extension of the dimension text.

The option changes orientation of angular dimension (i.e., the quarter of an angle on which the dimension is drawn).

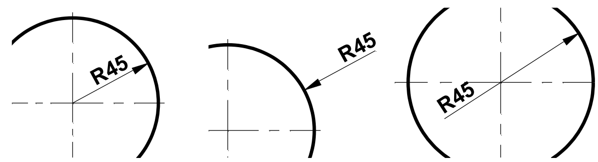
The option sets the mode of centering the dimension text. When pushed, the dimension text will be automatically centered between the witness lines.

The option helps quickly changing the dimension value prefix (“R”, “∅”, “M”,“□”, “O”), without calling the dimension parameters dialog box. The option serves for changing the reference elements (lines, nodes) of the dimension being edited. The fixing position of the dimension created using the option , can be changed by selecting two nodes.



Sometimes, it is necessary to change the witness line attachment point. To do this, select the dimension at the point of the intended origin node of the witness line, and pick the option .

If a wrong dimension was selected, alter the selection using the option .



The option or / , depending on dimension type, allows changing its type without altering its references.

Note that if the dimension was tied to a node using the option , then to modify its fixing condition, first use the option .

Option is used to assign a name to the selected dimension. The name is unique and allows you to uniquely identify this dimension.

A selected dimension can be deleted using the option . Deleting chain dimensions or dimensions from one base can be done separately for each dimension. Deleting the parent dimension (which is the dimension between the first two lines) causes deletion of the whole dimension group. The same rule is used for ordinate dimensions, created on one base: any dimension in the series, except the base one, is deleted as a separate entity; when deleting the base dimension, all dependent dimensions are also deleted.

As in the case of other elements, multiple selection is done by the option , or using box selection or clicking while holding down the <Shift> key (for adding to the list of selected) or <Ctrl> (for excluding from the list of selected).

For editing parameters of a group of selected dimensions, use the option:

	<P>	Set selected Element(s) parameters

First, select the set of parameters to be modified, in the dialog box that comes up on the screen. The standard dimension parameter dialog box will follow, allowing to define new parameter values. To define color, later, level and priority, one can also use the system toolbar. The option  helps copying parameters from another existing dimension.

Remember, that a number of dimension settings are defined by default, which can be changed in the command **ST: Set Document Paremeters.**

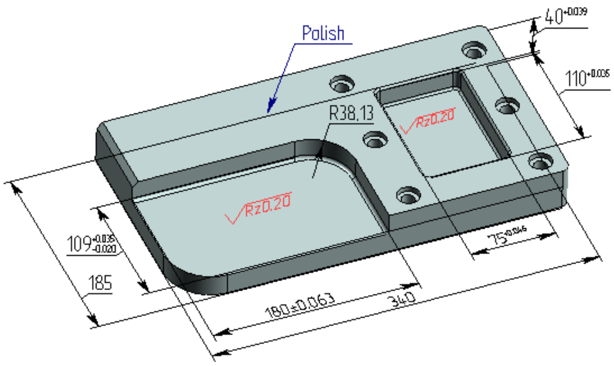
One can also enter the dimension editing command directly from the command **D: Create dimension**, using the option:

	<F4>	Edit

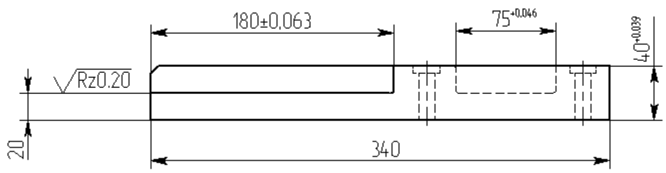
The third way of selecting a dimension for editing is available, when the system is in the command-waiting mode. Move the pointer over the dimension to be modified, and click . As a result, editing of the selected dimension begins. Besides, one can select the dimension and right click . The coming up context menu provides the commands for editing, deleting and modifying properties of the selected dimension.

Working with dimensions in the 3D window

When using the 3D version of the system, you can create dimensions, just as well as leader notes and roughness symbols, in the 3D window, on the faces of the 3D model. This allows you to create fully functional three-dimensional drawings.



The parameters of such dimensions are automatically passed to the dimensions created in the 2D window on the respective lines of a 2D projection of a given model.



3D dimensions can serve as driving ("control") dimensions for 3D operations or construction elements. That means, when a dimension's nominal value is modified, the respective parameters of the operation or 3D construction element adjust to the change automatically.

You can also use the dimensions corresponding to the driving ones that are created on 2D projections, in order to modify the values of operation parameters. The system will match such pairs of dimensions automatically.

> The detailed description of creating 3D dimensions and dimensions on projections is provided in the chapters “3D Annotations” and “2D Projections. Creating Drawings from 3D Models”.

TEXT

By learning various text handling techniques presented in this chapter, you will gain the command of a wide range of tools for handling text in T-FLEX CAD environment. At the time of decorating a drawing, you can insert standalone notes consisting of one or more lines of text. These lines are positioned on the drawing according to the attachment point and are controlled by a number of parameters that define the font, size, rotation angle, etc. (the section **String Text**). It is also possible to insert large amounts of text, including several paragraphs with different formatting (the sections **Paragraph Text and Multiline Text**). Table creation is also supported (the section **Table**). Any text may include variables created in the drawing and text excerpts from the dictionary (the section **Working with dictionary**). If necessary, a text can be imported or exported.

Creating Text

To create a text, the command is used **TE: Create Text**:

Icon	Ribbon
	Draw → Title Block → Text
Keyboard	Textual Menu
<TE>	Construct > Text

Upon calling the command, the following options are available in the automenu:

	<M>	Create Multiline Text
	<R>	Create paragraph text
	<B>	Create Table
	<D>	Create string text
	<P>	Set Text Parameters
	<Alt+P>	Copy Properties from Existing Element
	<A>	Set absolute coordinates
	<N>	Set relation with Node
	<L>	Set relation with Line (available for The string text only)
	<C>	Place Text around Circle (available for The string text only)
	<F4>	Execute Edit Text command

	<Esc>	Exit command

The first four options serve for selecting the type of the text to be created (multiline, paragraph, string text and table). The creation and handling techniques will be reviewed in details for each type in the respective sections of this chapter.

> Note that on subsequent calls to the command the type option will be turned on that was used in the previous command session. The default type is .

The text being created can be positioned either in absolute coordinates (the option), or attached to an existing node (the option) to have its position adjust together with the position of the specified nodes. A string text can also be positioned at the specified vertical and horizontal offsets with respect to the attachment node.

Besides that, the following additional options will be available in the automenu for a string text in order to relate it to construction entities (lines and circles). Those allow defining the rotation angle and the shape of the text according to the position and shape of the construction entities.

Text of any type can also be bound to the joint points between graphic lines belonging to 2D fragments or 2D projections (a 2D node is automatically created when selecting such a point).

Before you begin creating a text, you can set default parameters for all newly created text by using the option . Upon calling this option, the text parameters dialog box appears.

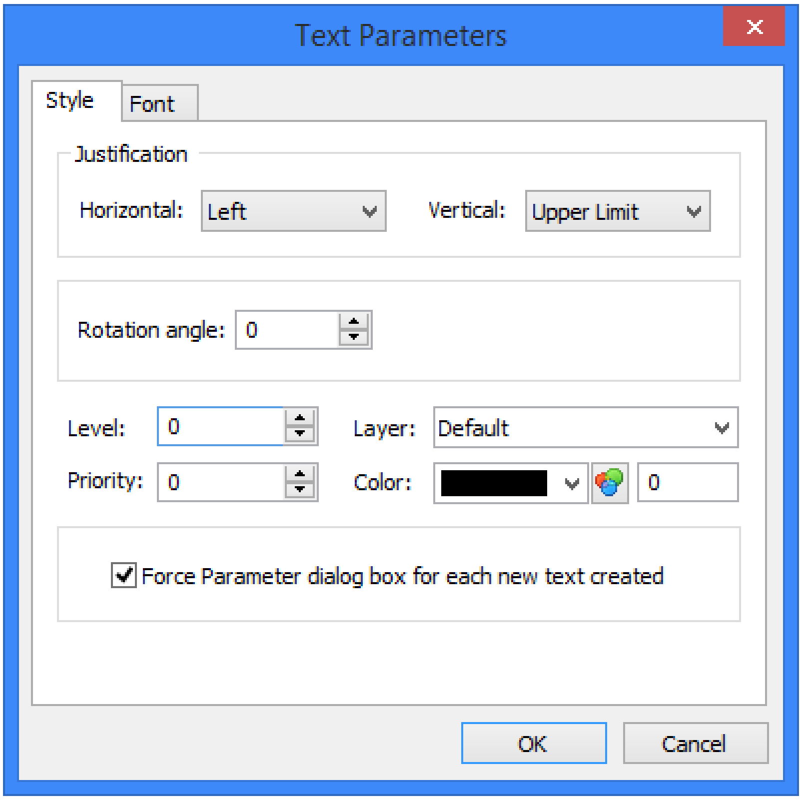
Default Text Parameters

“Style” Tab

The first group of parameters – **“Justification”** - defines the element position with respect to its attachment point, as well is justification modes of the element contents with respect to its boundaries. The effect of these parameters is different for different types of text (see the respective sections for details).

Justification - Horizontal. This parameter can take five values: **Left**, **Center**, **Right**, **Left&Center**, **Right&Center**.

Generally, this parameter defines the way of positioning the text with respect to the attachment point, as well as the horizontal justification mode of the text contents. The combination values of the parameter, such as “Left&Center”, simultaneously define the text contents justification (the first value, along the left margin), and the attachment mode (centered with respect to the attachment element).



The simple parameter value entries, such as "Left", simultaneously set the justification of the text contents and the same attachment mode.

When working with a paragraph text, the attachment mode is ignored, since this element is attached by two points. In the case of "Table"-type text, the text contents justification mode defined in this dialog box is ignored.

Justification - Vertical. In the case of the paragraph text, this parameter defines the vertical justification of the text contents. In all other cases, it defines the way of positioning the text with respect to the attachment point. However, this parameter affects various types of text in the ways specific to each type.

This parameter can have five values:

Lower Limit - The string text is positioned above the attachment point at the distance defined by the font size; multiline text and table are attached at their lower boundary; in the case of the paragraph text, this defines the vertical justification of the text contents along the bottom margin;

Lower Base - The string text will be positioned immediately above the attachment point; for the rest of text types this parameter is equivalent to the previous one;

Center - The string, multiline text and table are centered with respect to the attachment point; the contents of the paragraph text are vertically centered;

Upper Base - The string text is positioned immediately under the attachment point; multiline text and table are attached at the upper boundary; the contents of the paragraph text are top-aligned vertically;

Upper Limit - The string text is positioned under the attachment point at the distance defined by the font size; for the rest of text types this parameter is equivalent to the previous one.

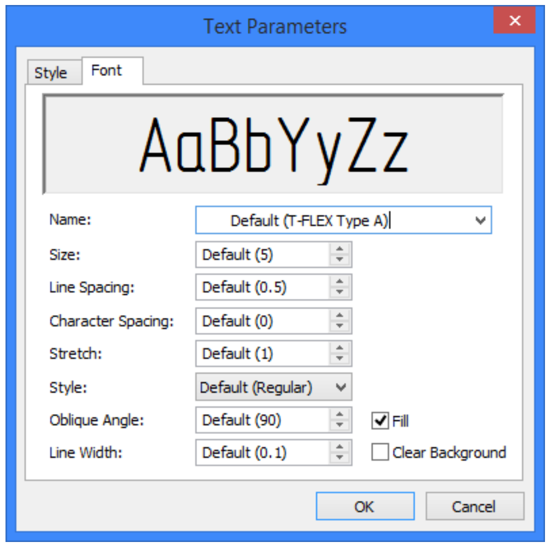
Rotation angle. Defines the Rotation angle of the text with respect to the horizontal coordinate axis in degrees. Positive rotation is counterclockwise.

Symmetric. This parameter defines "mirror" text mode.

General system parameters Color, **Level**, **Layer**, **Priority**.

Force Parameter dialog box for each new text created. This parameter makes sense only for the string text. If it is set, then the contents of the string text being created can be defined on the additional tab "Contents" of the parameters dialog box. Otherwise, the special text editor is invoked.

"Font" Tab



Name. This parameter defines the font name and type. T-FLEX CAD supports use of the two types of fonts: the TrueType () fonts that are the Windows standard, and the vector font format .SHX (). The fonts of the TrueType and SHX formats are distinguished in the font menu by the respective icons before the font name.

Size. Defines the vertical size of capital letters (for example, the height of the character "A").

Oblique Angle. Defines the oblique angle of the font. The normal (vertical) font has the slant angle equal to 90°. The slant angle of 75° makes the font Italic. This parameter is of higher priority than the parameter "Style".

Line Spacing. Defines the spacing between the neighboring lines of a multiline text. Line interval is defined in relative units. To calculate the absolute value of the line spacing, multiply this parameter by the font height.

Character Spacing. Defines additional spacing between the two neighboring characters in a line. The value of this parameter is also relative. To calculate the absolute value of the character spacing, multiply this parameter by the font height.

Clear background. This parameter is used for padding the text outline box with the drawing background color. Clearing background may be convenient when displaying text over hatches, fills, etc.

Stretch. Defines the scale factor for the width of the font symbol. Any stretch value can be specified except 0.

Style. This is a standard parameter for fonts TrueType (it does not affect SHX-fonts). It is selected from the list (normal, bold, italic, bold italic).

The following two options affect only SHX-fonts:

Line Width. Defines the width of the text contour lines for the fonts of the formats *.SHX.

Fill. Note that not all fonts can be filled. A font can be filled, if a file is present in the T-FLEX system folder with the extension CHD and the same name as the respective SHX font name. The file can be empty. In this case, every font character will be filled.

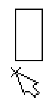
Once the font type and its parameters are defined, the result is displayed in the preview pane.

> Any parameter value can be defined by a variable. A textual variable can be used for the font name in this case.

Creating String Text

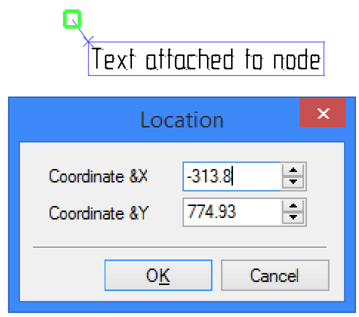
To create a string text, call the command **TE: Create Text**, and then select the option in the automenu. The necessary text parameters can be defined before creating the text (the option). These parameters will be applied to all types of the newly created text by default.

Upon calling the option , the graphic pointer will be displayed as a box and a crossing. The box height corresponds with the height of the text font.



The position of the crossing indicates the position of the text attachment point. The position of the box with respect to the crossing indicates the scheme of the text justification and positioning with respect to the attachment point, defined in the text parameters dialog box. The height of the box corresponds to the size of the text font.

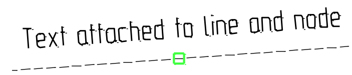
You can define the text position by clicking , and then invoke the text editor for inputting its contents. Besides, the text can be attached to a node, line or circle in order to have its position adjust together with the drawing parameter modifications.



If attached to a node, the offsets of the text attachment point with respect to the node are maintained constant. If you want to impose such a relation, use the key <N> to select the desired node before clicking for positioning the text.

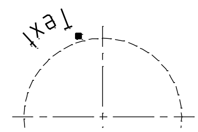
If you want to specify the exact horizontal or vertical offset of the text from a node, use the option <A> for entering the offset values. The same option used without selecting a node will allow you entering the exact text position with respect to the coordinates of the drawing.

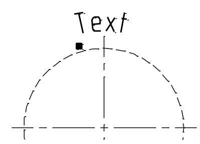
An attachment line defines the Rotation angle of the text. The text can be positioned parallel to the line or at some angle (the angle being defined among the text parameters). Type <L> for tying the text to a line.



Attaching a text to a node and to a line can be combined by using the options <L> and <N> subsequently. This allows, for example, attaching the text in such a way that it will adequately adjust as the image rotates. The example of the right hand side shows the result of selecting a construction line and a node. The option <A> was used for selecting the node with the offset values *"0,0"*.

If you want to wrap the text on a circle, select a circle using the option <C>. You can flip such text (rotate by 180°) by using the justification parameters for defining the position of the attachment point.





The diagram on the right hand side shows a text wrapped on a circle. The following justification parameters were used for creating this text: horizontal – left, vertical – upper limit. By selecting this text for editing one can see a small square displayed in the upper left corner of this text, indicating the position of the text attachment point.

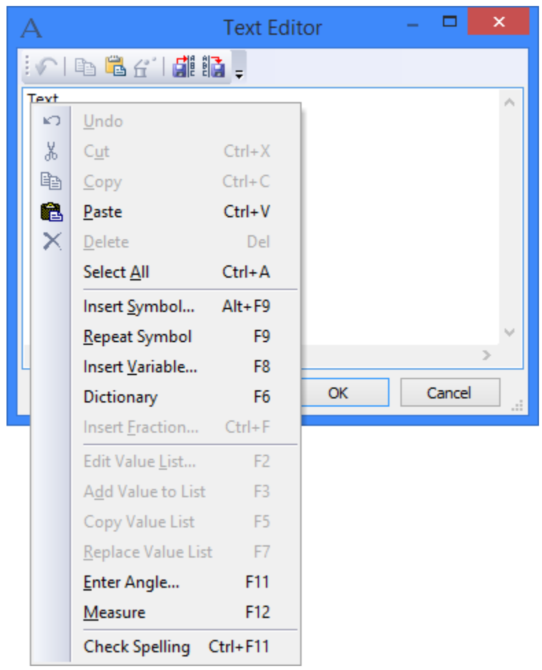
Modifying the justification parameters of this text as follows: horizontal – left, vertical – lower limit - flips the text by 180°.

> With the snapping turned on, the construction elements suitable for the text attachment references are pre-highlighted as the pointer approaches. To attach the text to those, you can simply click . Construction line intersections can also be selected as the attachment node, the actual node being created automatically in this case.

Upon defining the text position, a window will be displayed for inputting the text contents. The text may contain variables, subscripts and superscripts, as well as special symbols. If the flag "Force Parameter dialog box for each new text created" is set among the text parameters, then the parameters dialog box appears with an additional tab "Contents", where the desired text can be input.

Otherwise, a special text editor will be displayed for inputting the text contents. **Text Editor** is provided for inputting and editing the contents of string text. The text editor supports all functions of a common Windows text editor, including importing/exporting text files.

The following commands are provided in the context menu of both the text editor and the parameters dialog of the string text: Insert Symbol (by selecting it from the symbol table); Repeat Symbol inserted before; Insert Variable; Insert text from Dictionary (see below); call Measure command.



Subscript and Superscript Text. Use of Variables in Text

The string text allows insertion of textual and numerical variables, subscripts and superscripts, as well as special symbols. These symbols are displayed using the same font as the main text. You can use the context menu commands for inserting, while the pointer is within the text editor or the text contents input pane in the parameters dialog.

For example, to insert a variable in a text, you can use the context menu item "Insert Variable..." or use the function key <F8>. The standard dialog box "Insert Variable" will appear on the screen. Upon selecting a variable, the reference to the variable will be inserted in the text contents in the following format: *{<variable name>}*. In the drawing, the reference to the variable will be replaced by its value.

The string text also allows "manual" insertion of variables and various symbols in the text. To insert variable values in a text, use the following syntax:

{<variable name>} or {<format>,<variable name>}

Example of using variables:

Create the text with the following contents:

Diameter of cylinder is {D} millimeters

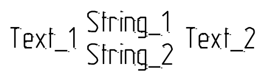
Assigning the variable *"D"* the value *"10"* will make the text appear on the drawing as shown on the diagram.

The following is an example of using formatted representation of variables:

Today is {"%lg",DAY}, {"%s",$MONTH}, {YEAR}

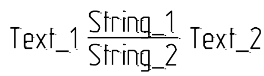
The format structure used by T-FLEX variables corresponds to the syntax of the input/output formats of the "C" programming language. Use of formats helps you control the appearance of the variable as displayed on the screen (for example, the number of decimal digits, or justification of the output value).

To insert a subscript or superscript at any position in a line of text, surround the intended block in double brackets:



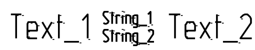
Text_1 [[String_1^String_2]] Text_2

Using the symbol "~" instead of "^", divides the *String_1* and *String_2* by a horizontal line:



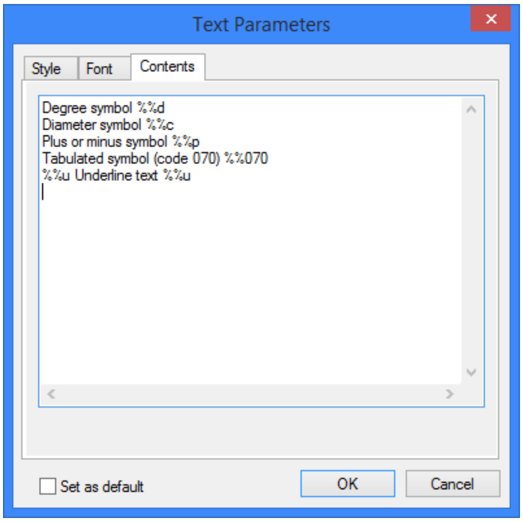
Text_1 [[String_1~String_2]] Text_2

Using angular brackets instead of square ones makes *String_1* and *String_2* displayed in a two times smaller font than the rest of the text:



Text_1 <<String_1^String_2>> Text_2

String text can be framed by prefixing it with the symbol "~". This feature affects the whole text.



Special symbols and underlining can be used in text.

Degree symbol °
Diameter symbol ⌀
Plus or minus symbol ±
Tabulated symbol (code 070) △
Underlined text

Text supports symbols in the Unicode format. Those are entered as *"\U+FFFF"*, where *"FFFF"* – is a four-position hexadecimal describing the symbol code. For example, use of the symbol *"\U+03A9"* in combination with the "Arial" font is displayed as the Greek character *"Ω"*.

> To insert such symbols, one can use the standard Windows symbol table (Character Map). In it, you can find out the symbol code and insert it in the string text editor.

Editing String Text

To edit a text, start the command **ET: Edit Text**:

Keyboard	Textual Menu	Icon
<ET>	Edit > Draw > Text	

Upon calling the command, the following options become available in the automenu:

	<*>	Select All Elements
	<R>	Select element from list (for named elements only)
	<Esc>	Exit command

Selection, editing the position and attachment, and modifying text parameters are similar to editing other system elements.

Selection of several text, as well as multiple selection of other system elements, can be done by box or by using the options (selection of all text) and (selection from the list of the named elements). Besides that, the string text allow subsequent selection of elements by using with the <Shift> key depressed. Use of in combination with the depressed key <Ctrl> excludes the text from the list of selected for editing.

If the selected text is attached to some construction element, this element will be highlighted.

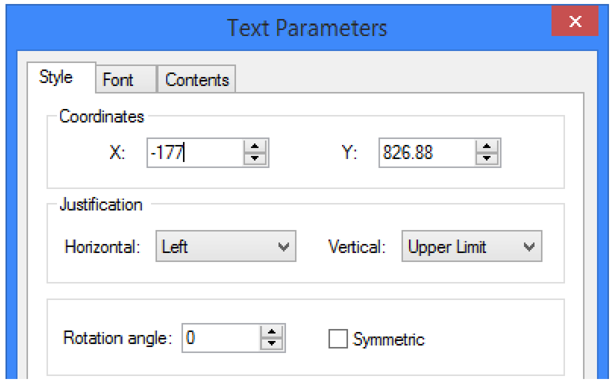
After selecting several text, the following options become available:

	<P>	Set selected Element(s) parameters
	<Alt+P>	Copy Properties from Existing Element
	<N>	Set relation with Node
	<K>	Break (kill) relations
	<J>	Merge Text
	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

Upon selecting a specific element, the following options are available:

	<E>	Edit selected Text
	<P>	Set selected Element(s) parameters
	<Alt+P>	Copy Properties from Existing Element
	<Y>	Create Name for selected Element
	<N>	Set relation with Node
	<L>	Set relation with Line
	<C>	Place Text around Circle
	<K>	Break (kill) relations
	<I>	Select Other Element
	<X>	Explode Text
	<D>	Duplicate Text
	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

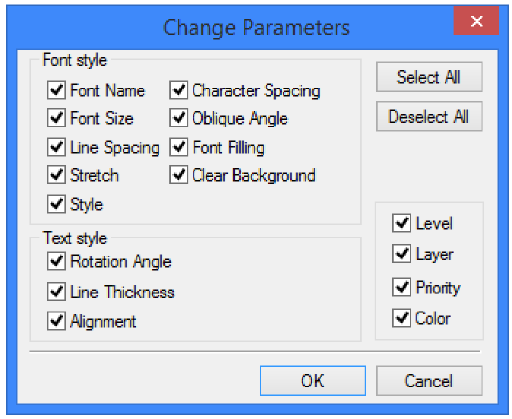
The option is used for editing the selected text contents. When the option is called, the text editor window appears on the screen.



To attach the selected text to a node, line or circle, the same options , , are used as when creating a text. To break and attachment, use the option .

The option allows breaking a text containing several lines into a number of separate elements. In this case, each line of the original text will be converted into a separate “String Text” element.

The option allows creating a copy of the selected text.



Modification of text parameters is bound to the option . The parameters dialog box appears in the screen. Besides accessing the text parameters and its contents, you can define the text position in the absolute coordinates or by an offset from the attachment node, if the text was attached to a node.

When calling the option while several elements are selected, you will have to first specify the parameters to be modified in the dialog box “Change Parameters”. By default, all parameters of the selected elements are subject to editing. Upon specifying the parameters to edit, this standard text parameters dialog box appears.

Creating Paragraph Text

Paragraph Text is a text located in a specified rectangular area. The lines of such text are wrapped automatically upon reaching the area boundary. Various formatting functionalities can be used for paragraph text creation that are applicable to any text fragment.

To create a paragraph text, use the command **TE: Create Text.**

When creating a text, the parameters are automatically used that were set as the default parameters (the option). Originally, these parameters use the settings “from document”. The text parameters will be applied to the whole content of the given text.

When editing the contents of the text being created, you can assign specific parameters to its separate elements, for example, to an isolated word or sentence. This capability is described in the topic **Standard formatting options**.

Defining Text Position and Size

In the automenu, select the option:

	<R>	Create paragraph text

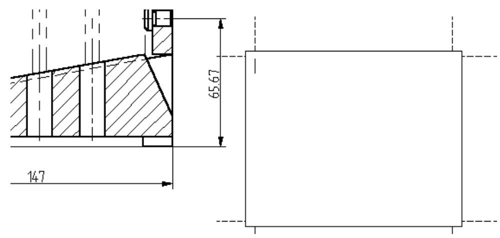
To create a paragraph text, first define the position and boundaries of the rectangular area that will hold the text (you can subsequently define several such areas with their boundaries). To do this, subsequently select the two bounding points for attachment. This can be freely done by clicking or specified by selecting existing nodes. Upon defining the first attachment point, a rectangle starts rubberbanding following the pointer, indicating the size and position of the text being created.

One can subsequently input the boundaries of several such rectangles. This, however, creates only one element of the type “paragraph text”. This means, as the text being input fully occupies the first rectangle, the input automatically continues in the second rectangular area, etc. in the order of rectangles creation.

Next, click inside the defined area or the icon (<End>) to proceed with entering the text contents. At this moment, the rectangle will appear as follows, depending on the way of attaching the text:

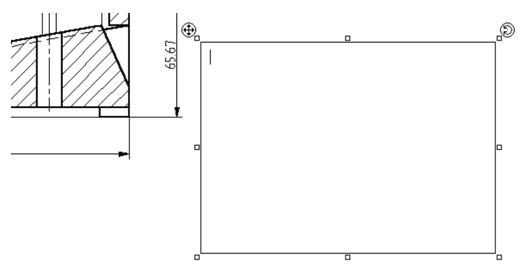
If the paragraph text is attached to nodes

In this case, the rectangle can be modified or moved using the construction lines to which the paragraph text is attached.



If the paragraph text is not constrained

In this case, the rectangle can be moved, rotated or resized by using the provided handles while in the text editing mode. Move the rectangle using the handle located in the upper left corner of the rectangular area. Move the pointer to the handle (the pointer appearance will change), depress and drag the mouse to the desired position. The rectangular area of the text will follow the pointer.



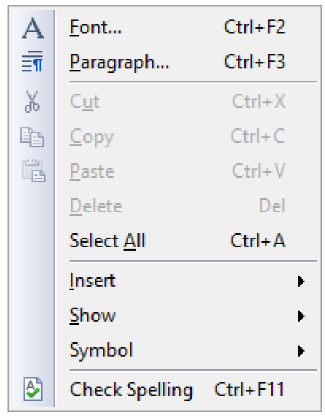
Meanwhile, the coordinates of the upper left corner of the rectangular area will be displayed in the auxiliary fields of the status bar: X=-235.375 Y=819.505 Angle = 338.25. To locate the text, move the pointer to the handle located in the up or right corner. The pointer appearance will change accordingly. Hold down the left mouse button and rotate the text rectangle in the desired direction. Rotation will be about the text centerpoint with snapping at each 15°. Rotation without snapping is done by depressing the <Ctrl> key. Rotation angle will also be displayed in the auxiliary field of the status bar.

To resize the rectangle, move the pointer over one of the small squares located at midpoints of each side and at each corner of the rectangle. The pointer will assume the shape corresponding to the vertical, horizontal or diagonal resizing. Hold the down and move the pointer in the desired direction.

Inputting Text Contents

When inputting the contents of a paragraph text, the user is provided with various tools for formatting the text being input. To call the necessary options, you can use the system toolbar, the context menu or the command automenu.

Selection of a text fragment (as for its formatting) is done by dragging the mouse with the depressed. To select the whole contents of the paragraph text, one can use the key combination <Ctrl><A> or the context menu command **Select All**.



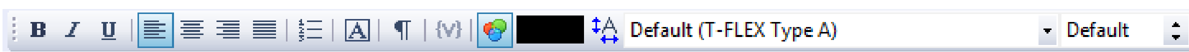
The command **Copy** allows copying a highlighted text fragment or table into the clipboard for further pasting into another "Text" element or into another application.

The command **Paste** is used for pasting a text or table from the clipboard. For example, you can copy a table from Microsoft Word and paste it in T-FLEX CAD.

When working with variables, as well as with various objects inserted in the text, additional commands are provided in the context menu for defining and modifying the object parameters (see below).

System toolbar options for handling text

As you enter the mode of creating (editing) text contents, the system toolbar appearance changes. Various text-handling options become available. Those work on all elements of the paragraph text.



Bold font <Ctrl+B>.

Italic font <Ctrl+I>.

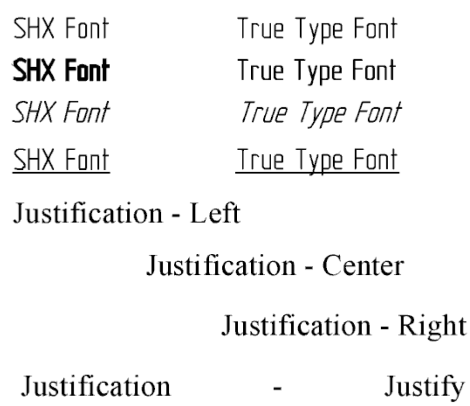
Underline <Ctrl+U>.

Left Text Alignment <Ctrl+L>.

Center Text Alignment <Ctrl+T>.

Right Text Alignment <Ctrl+H>.

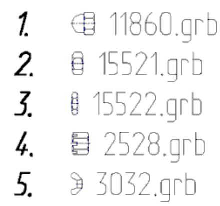
Text Justification <Ctrl+J>.



Text numbering <Ctrl+M>. Turns on and off automatic numbering of parameters.

Once turned on, the paragraph numbering begins from number 1. The subsequent paragraphs are numbered automatically until the command is turned off.

To access additional settings of text numbering, lunch the command **Format Paragraph** (see the description below).



Box. With this option turned on, the selected text will be framed by a box.

Show Unprintable characters <Ctrl+F2>. Toggles the display of the formatting marks. When viewing or editing a document, various formatting marks can be displayed, such as tabulation characters, spaces and paragraph marks that do not appear in printouts. For example, tabs are marked by arrows, spaces - by dots. This allows identifying, for instance, extra spaces between the words, spaces used instead of tabulation, etc.

Show Variable names <Ctrl+F3>. Defines whether the values or names of the variables are displayed on the drawing.

Besides the above, the system toolbar allows defining the color, type and size of the text being created or selected portion of an existing text. Initially, these parameters are set "By default", that is, their values are taken from the text properties defined before the text creation.

To set a color different from the default color, turn off the button and select the color from the list (the button turns off automatically as you select a color from the list). To use the color set "By default", push the button .

Automenu options for handling text

When inputting a text, the following automenu options are available to the user:

Standard Windows options:

Cut selected text. <Ctrl><X>

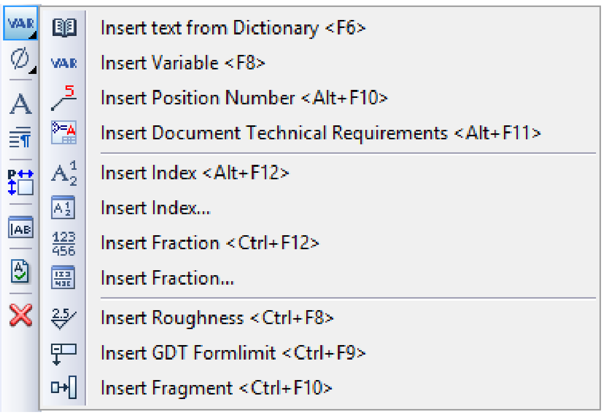
Copy selected text to Clipboard. <Ctrl><C>

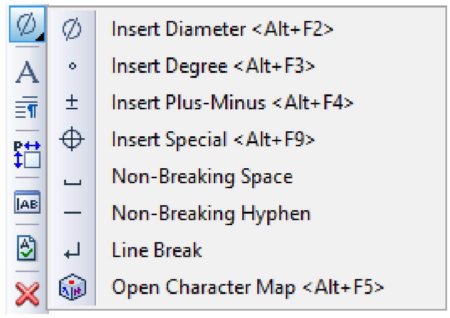
Paste selected text from Clipboard. <Ctrl><V>

> When pasting a text from the Clipboard, the system checks the type of the clipboard contents against the current variable, preventing, for instance, insertion of a character string into a numerical variable.

Insertion options:

A black triangle in the lower right corner of an icon indicates the presence of several enclosed options behind this icon. Holding the button 🖱 depressed a bit longer over such an icon opens up a menu with additional options.

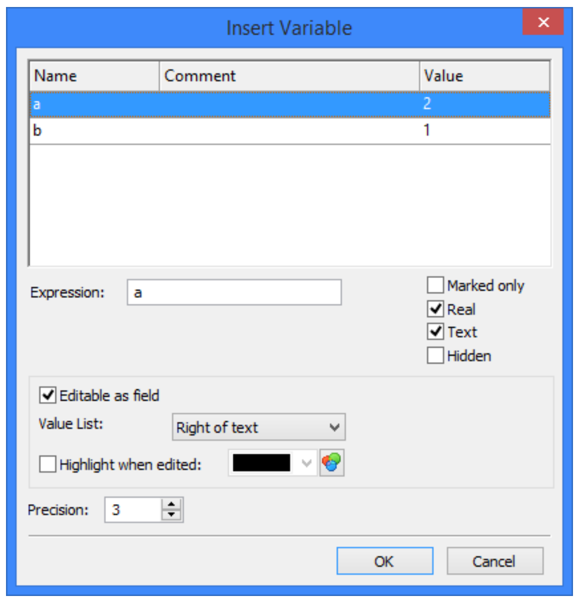




Any of the enclosed options can be displayed by the automenu. Usually, it is the icon of the option used last in this command.

 **Insert text from Dictionary** <F6>. Opens the dictionary (see the section “Working with dictionary” below).

 **Insert Variable** <F8>. If variables exist in the drawing at the time of the text creation, those can be inserted in the text using this option. Upon calling the command, the “Insert Variable” dialog box appears on the screen.



This dialog displays the list of variables created in the current drawing. This list can be sorted by the attributes of the variables when displayed: **Marked only** – the variables that were checkmarked in the variable editor (external); **Real** – the variables with numerical value; **Text** – the textual string variables.

Next, select a variable from the list to be inserted. The variable name will then be automatically entered in the **Expression** input box. You can create a new variable by entering its name manually.

Besides a variable name, you can insert an expression – in this case, the result of its evaluation will be displayed on the drawing. To make a variable available for editing directly in the text, set the flag **Editable as field**.

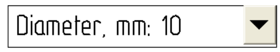
To edit an expression inserted in a text, call the “Insert variable” dialog box again. To do this, while editing the text, point the mouse to the expression/variable and right click, and then select the **Parameters…** item in the context menu.

You can control the displayed number of decimal digits of real variables inserted in the text by the parameter **Precision**.

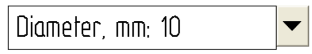
If the variable has a list of predefined values, then you can specify the position of the list access button that will be displayed in the variable editing mode:

Right of text – the button will be located on the right hand side and immediately after the variable.

Left of text border – the button will be located at the end of the line within the rectangular text area.



Right of text border – next to the end of the line outside the rectangular text area.

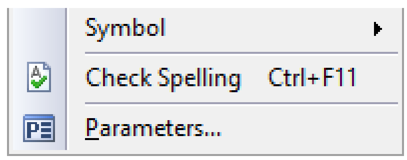


None – the list access button is not displayed.

If a variable with a predefined list is followed in the text by another element (text or variable), then the option *Right of text* will be used for the list access button of this variable.

The editable variables can be highlighted among the given paragraph text for easy selection while in the editing mode (see the section “ Editing paragraph text”). You can define their highlighting color. To do this, check the flag **Highlight when edited** and select the highlighting color from the list.

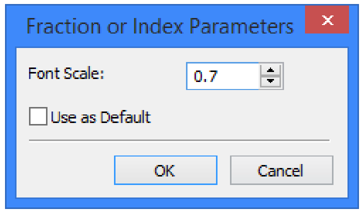
To modify an expression inserted in a text, call the Insert variable dialog box again. To do this, while in the text editing mode, point the mouse to the area occupied by the expression and right click, and then select the item **Parameters…** in the context menu.



Insert position numbers <Alt+F10>. This command allows us to insert the BOM position number into the text. When invoking this command the “BOM record selection” window appears, in which it is required to specify the desired BOM object. When the position number of the selected object in the BOM is changed, it will also be updated in the text.

Insert document’s technical requirements <Alt+F11>. This option allows us to insert into the text the technical requirements of the document that are specified in the **Format > Technical Requirements > Document’s Tech Requirements…** command.

Insert Index <Ctrl+F11> and **Insert Index…**. These options allow inserting subscripts and superscripts in a text.



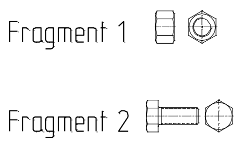
Upon selecting the option , two input boxes appear by the cursor position – the subscript and the superscript one. Use the arrow keys or the mouse for navigation. When selecting the option the dialog box appears on the screen in which the user can specify the font scale for the inserted indexes. After closing the dialog box, the fields for entering indexes appear at the cursor's position.

Insert Fraction <Ctrl+F12> and **Insert Fraction…**. Are similar to the options **Insert Index** (and). The subscript and superscript fields are divided by a horizontal line in this case.

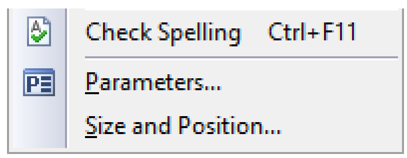
The options **Insert Index** and **Insert Fraction** allow unlimited nesting, which means that any subscript/superscript or fraction can contain an unlimited number of its own subscripts/superscripts or fractions.

Insert Roughness <Ctrl+F8>. Allows inserting the roughness symbol in the text. Upon calling the command, the standard "Roughness Symbol Parameters" dialog box appears. The same options are available in this case as when inserting the roughness symbol directly in the drawing.

Insert GDT Formlimit<Ctrl+F9>. This command is similar to the previous one. Upon calling the command, the standard "GD&T Symbol Parameters" dialog box appears.



Insert Fragment <Ctrl+F10>. Sometimes, you may need to insert, let's say, a symbol, that is not present in any table. In this case, first you can create its 2D drawing, and then insert it in the text as a 2D fragment. When inserting a fragment in a text, the standard "Insert Fragment" dialog box appears.

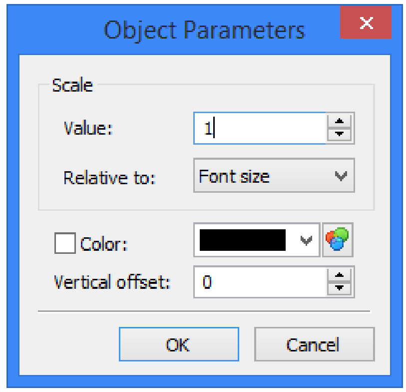


The images of the roughness or tolerance symbol or a fragment can be scaled in two ways. The scaling can be with respect to the font height or with respect to the object itself.

To alter the way of scaling of the inserted object (the roughness, tolerance or a fragment), while in the text editing (creation) mode, right click over the element, and then select the “Size and Position...” item in the context menu.

In the coming up “Object Parameters” dialog box, specify the scale value and the way of scale (with respect to “Object size” or “Font size”).

The element properties can be modified by selecting the “Parameters...” item.



Insert Diameter <Alt+F2>, **Insert Degree** <Alt+F3>, **Insert** “**Plus Minus**” <Alt+F4> - the shortcuts are provided for inserting these frequently used symbols.

Insert Special <Alt+F9>. Inserts a symbol from the table of special symbols.

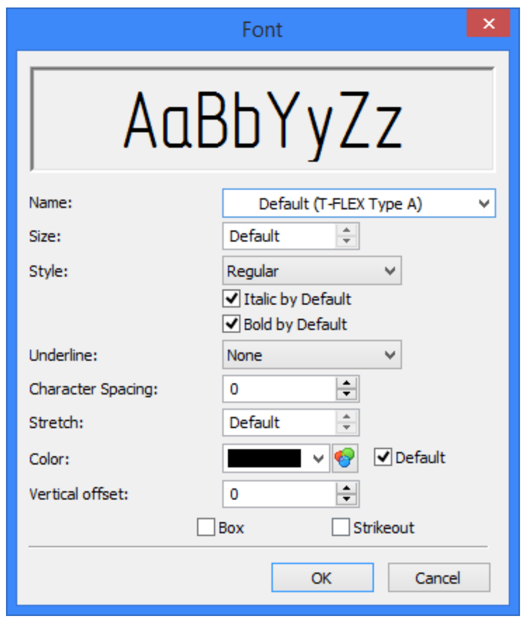
Non-Breaking Space <Shift+Ctrl+Spacebar> and **Non-Breaking Hyphen** <Shift+Ctrl+"-">. Allows creating a phrase without wrapping to the new line.

Line Break <Shift+Enter>. Continues the text on the new line without creating a new paragraph.

Open Character Map <Alt+F5>. This command brings up the standard Windows character map table, allowing to pick a symbol or a set of symbols into the clipboard.

Standard formatting options

Format Font <F9>. This option allows setting the special parameters for a text contents fragment, such as a separate word or sentence. Calling the option brings up the “Font” dialog box. The following parameters can be defined in this dialog:



Name and **Size**. Modify the font appearance and size of the selected text element. If set to “Default”, the font name and size assume the overall text parameters.

Style and **Italic by Default**. When the flag **Italic by Default** is set, the overall text italic style setting is used. When the flag is cleared, the edited text fragment is assigned the style, specified by the parameter **Style** (**Regular**, **Bold**, **Italic** and **Bold Italic**).

Underline. This parameter allows defining the way of underlining a text: "None" – without underlining, "Single" – with underlining (<u>This is the underlined text</u>).

Character spacing. Defines spacing between neighboring characters in a line.

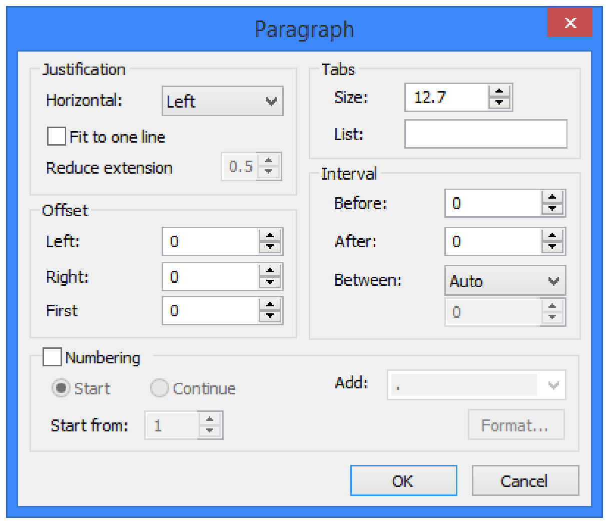
Color. This parameter sets the font color. If the "**Default**" flag is turned on, the text color is used from the general settings of the overall text.

Vertical offset. Defines the vertical text offset with respect to the bottom edge of the line. For positive offsets, the shift is upwards, for negative – downwards.

Box. With this flag turned on, the text will appear in a box. text in a box

Strikeout. Setting this flag will apply the Strikeout style on the text ~~Strikeout text~~

Format Paragraph <F10>. By calling this command, you can set the parameters of the current or the selected paragraphs in the dialog box:



Justification - Horizontal. This option controls the text horizontal justification. Four modes are supported: **Left**, **Right**, **Center**, **Justify**.

Fit to one line. Select this option when you need to fit the whole paragraph in one line. In this case, the font characters are first scaled by the width to the **minimum extension (width) factor**. If that's not enough, then the font height is reduced.

Reduce extension (width) to. Defines the minimum width factor.

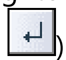
Tab size. Defines the distance at which the cursor moves after pressing the <Tab> key.

Tab list. You can enter several numbers in the **list**, separated by commas, that will define the tab sizes. The numbers are automatically sorted in the ascending order. These numbers are the distances from the text left margin to the text position. For example, if you enter the set of numbers 20,40,50, then after pressing the <Tab> key first time the cursor will move by 20 measurement units from the left margin; on pressing the second time – by 40, on the third time – by 50. Thereafter, the tabulation is done according to the specified size (the default is 12.7).

Offset. Defines the distance from the boundary of the rectangular area to the text – left, right, new line indent.

Interval. Defines the spacing between the lines. **Before** – sets the height of the first line in the paragraph, **After** – sets the spacing between the last line of the current paragraph in the first line of the next one. **Between** – sets the spacing between the lines in a paragraph. With the **Auto** setting, the spacing is set automatically, depending on the maximum font height (by the printed characters). **Minimum** - sets the numerical value of the minimum line spacing. If a printed character doesn't fit in the line by height, the spacing is increased automatically. **Exactly** - defines a strictly fixed numerical value of the line spacing. **Factor** – defines the line spacing as the product of the font height (by the printed characters) and the input factor.

Numbering. This option allows turning on the automatic numbering of the paragraphs. You can specify a number to start a new numbering or to continue an existing numbering with. Additionally, you can define the format of the font for the digits of the current numbering.

A new paragraph is automatically created by pressing the <Enter> key. However, you can wrap a textual string to the new line without creating a new paragraph. To do this, press <Shift>+<Enter> (the icon ).

Options to control text input and text editing

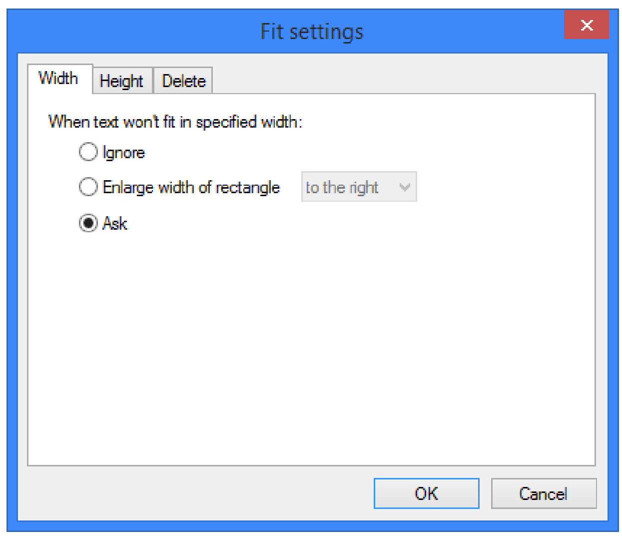
 **Parameters of changing Paragraph size** <Ctrl+F5>. This option is provided for the paragraph text only. It calls the dialog box for defining the coordinates of the window and the system actions.

The **Width** tab defines the system action in the case when the text doesn't fit into the box by width:

Ignore. The text part that doesn't fit in the box will be saved but not displayed on the screen.

Enlarge width of rectangle – in the specified direction by the specified amount (the respective numerical value is displayed in the dialog which automatically appears in the process of text input).

Ask. This sets automatic display of the query dialog in the case when the text does not fit in the box. If this parameter is not set, then the previously specified action will be executed without displaying the dialog. This parameter is On by default.

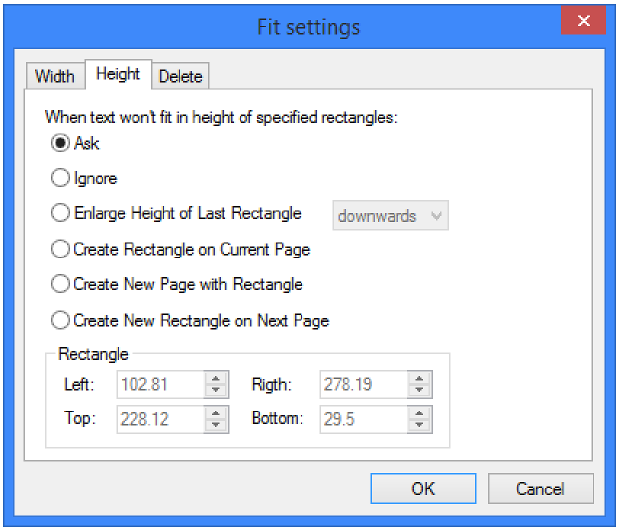


The **Height** tab defines the system actions in the case when the text doesn't fit in the box by height. It contains the same options as the **Width** tab, and, additionally:

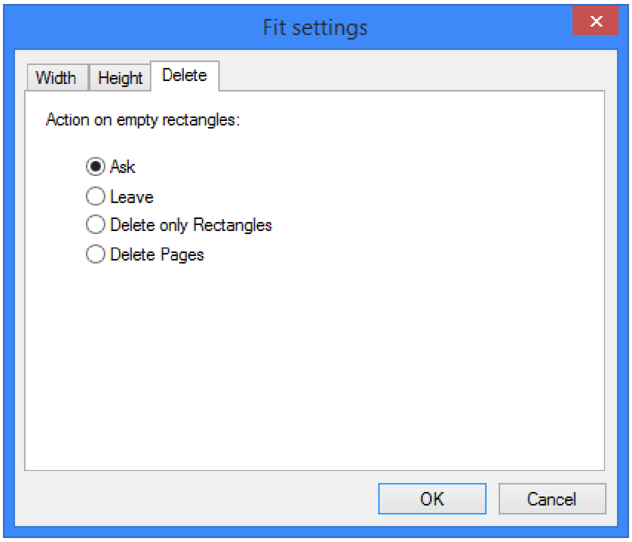
Create new rectangle on current/new/next

page. This option allows to create a new box with the specified coordinates on the respective page of the drawing. If necessary, the new page will be created automatically.

The **Deletion** tab serves to define the system actions in the case when the text shrinks so much that the last created box becomes empty. One can select one of the following options:



The **Deletion** tab serves to define the system actions in the case when the text shrinks so much that the last created box becomes empty. One can select one of the following options:



Ask. This parameter is similar to the same-name parameters on other types of this dialog. It sets the automatic display of the query dialog when the text shrinks. If this parameter is not set, then the previously specified action will be executed without displaying the dialog.

Leave. The paragraph text is left as is, with boxes remaining empty.

Delete only Rectangles. This means an emptied rectangle area (box) will be automatically deleted. Meanwhile, the document page, on which it was located, will still exist.

Delete Pages. An emptied rectangular area (box) of the paragraph text is automatically deleted. In addition, the document page, on which it was located, will also be deleted (provided that this text area was the only object on the page).

If the "Ask" option is set on any of the tabs of this dialog's window, and one of the above-described situations takes place, then an abridged dialog window will be automatically displayed. The contents of such dialog will be similar to the contents of the respective tab of the "Fit Settings" dialog. In addition, the dialog will have another parameter:

Don't ask this question again: ("For this text", "For this session"). If set, then the action defined in this dialog box will be performed in the future without displaying the dialog box. It is defined separately for the width and the height of the rectangle.

Edit in separate window <F11>. Allows editing the text in the text editor (see the section "Edit paragraph text").

Check Spelling <F7>. This option invokes the command for checking the spelling of the contents of the current text.

Editing Paragraph Text

To edit a paragraph text, use the command **ET: Edit Text**:

Keyboard	Textual Menu	Icon
<ET>	Edit > Draw > Text	

Selection of several paragraph text, just like multiple selection of other system elements, can be done by box or using the options (selects all text) and (selection from the list of the named elements). When selected, all rectangular areas of the specified paragraph text are highlighted.

Upon multiple text selection, the following options become available:

	<P>	Set selected Element(s) parameters
	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

Upon selecting a single element, the following options are available:

	<E>	Edit selected Text
	<P>	Set selected Element(s) parameters
	<D>	Add Rectangle
	<Y>	Create Name for selected Element
	<N>	Set relation with Node
	<I>	Select Other Element
	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

Text contents editing mode

To begin editing, click inside the rectangular area or select the option .

There is yet another way to select a text for editing. When the system is in the command waiting mode, point the mouse to the text to be edited and right click. In the coming up context menu, select the item **Edit text**.

Besides that, if the parameter "**Transparent**" **Text editing** is set among the drawing settings (the command **Customize** > **Set document parameters**, the tab "Preferences"), then you can start the text editing mode by clicking the text (the pointer in this case turns into a textual cursor).

Modifying parameters of selected elements

To modify parameters of selected text, use the option .

If the whole paragraph text was selected, then after calling the option a dialog box appears with general parameters of the paragraph text:

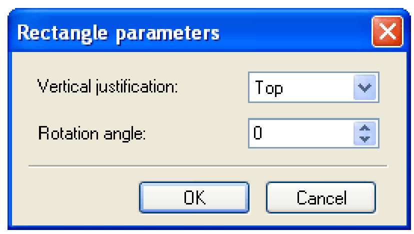
Level, layer, priority, color – the standard parameters for all T-FLEX CAD elements.

Symmetric. This parameter defines "mirror" text mode. **Account for italic font while formatting**. Enabling this parameter allows us to take into consideration the text in italic when placing the elements of fractions, indices and also when formatting the paragraph in the "fit into one line" mode.

By default this parameter is disabled for texts created in T-FLEX CAD of versions 11 and lower, and enabled for the newly created texts.

The **Font** tab contains several standard parameters.

If a rectangular area is selected (see the topic "Editing rectangle" for selection tips), then the option call brings up the dialog box **Rectangle parameters:**



Vertical Justification - sets the vertical text justification inside the selected rectangle. Three justification options are available –**Top**, **Center**, **and Bottom**.

Rotation angle – allows rotating the rectangular area by an arbitrary angle, in degrees.

In the case of selecting several elements, upon calling the option <P>, the dialog box appears for selecting the parameters to be edited. The checkmarked parameters will be available for editing in the general text parameters dialog box that follows.

Adding rectangle

The following option adds a rectangle at the end of the list of the existing rectangles:

	<D>	Add Rectangle

If an existing area of the paragraph text being edited was selected before calling the option, then the new rectangle will be inserted in the list before the selected area.

The new rectangle (box) may be automatically created when the content of a paragraph text is edited.

Editing rectangle

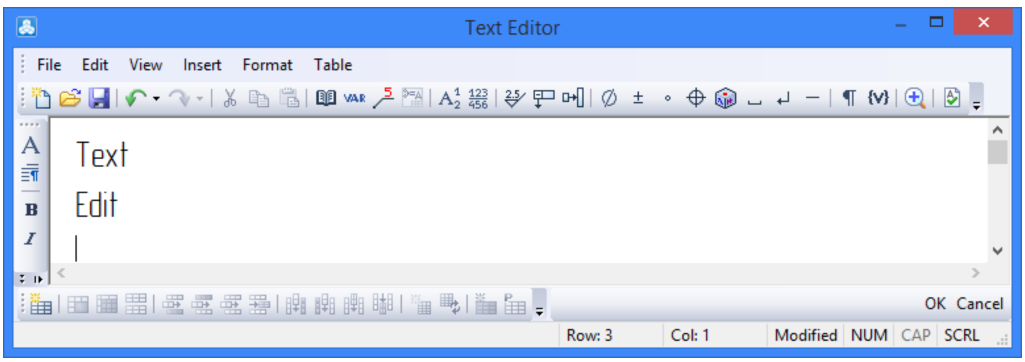
To resize an arbitrary rectangular area of a paragraph text, first you need to select it. To do this, click on the outer side of the rectangle. Now, you can resize it by the mouse.

Automatic deletion of rectangle

It is possible that, as you edit a paragraph text that spans several boxes, the text shrinks so much that the last created box becomes empty. In this case, the system could maintain the empty box, or delete it from the page, or even delete it together with the page on which it is located, provided that this text box is the only object on that page. The specific system actions depend on the settings made in the dialog of the “Parameters of changing Paragraph size” option (the icon) in the text content editing mode (see the topic “Entering text content”).

Editing text in separate window

Besides the way of editing a paragraph text directly on the drawing, there is a provision for working in the text editor. To start the editor in the text contents editing mode, call the option . The text editor window will be displayed.



The text editor provides all the text handling options described above (formatting and insertion), plus the option:

Zoom. The displayed text can be zoomed for convenience.

The text editor also allows working with various text document files. The following formats are supported: T-FLEX Paragraph Text (*.tft), Rich Text Format (*.rtf), Text files (*.txt), DOS Text files (*.txt).

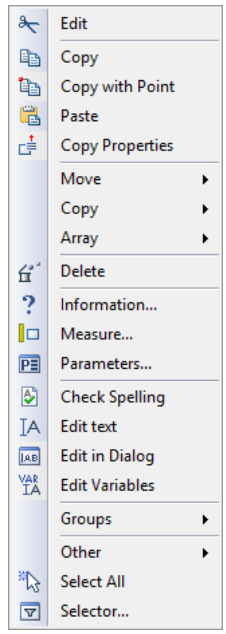
The following options are used for handling files:

New Text <Ctrl+N>. Creates a new document.

Open <Ctrl+O>. Opens a text document.

Save as <Ctrl+S>. Saves changes in a file.

These options support exchange of text with other text editors. Suppose, for example, that you need to import a Microsoft Word document into a T-FLEX CAD drawing. To do this, use the **Save as** command in Microsoft Word and save a copy of the document in the RTF format. Then, in the T-FLEX CAD text editor open the RTF document using the command **Open file**. Thus, you open a copy of the original Word document with all the formatting preserved. You can export text in a similar way. Besides, you can carry over a text from one editor to another by using the clipboard via the commands **Put selection on Clipboard, Paste Clipboard contents.**



The commands for editing paragraph text can be also called from the context menu by right clicking over an appropriate element:

- "Edit" - calls the command **ET: Edit Text**;
- "Delete" - deletes the selected text;
- "Parameters" - lunches the parameters editing of the selected text;
- "Edit text" (for multiline and paragraph text) - executes the command of editing the paragraph text in the text contents editing mode;
- "Edit in Dialog" - calls the text editor for editing the contents of the selected text;
- "Edit Variables" (if variables exist in the selected text) - starts **variable editing mode**.

Variable editing mode

To modify the values of the variables inserted in a paragraph text, you do not have to start **variable editor**. Modifications can be done directly in the text, which is convenient.

> This way of editing variables is applicable only to the variables inserted in the text with the flag "Editable as field" checked.

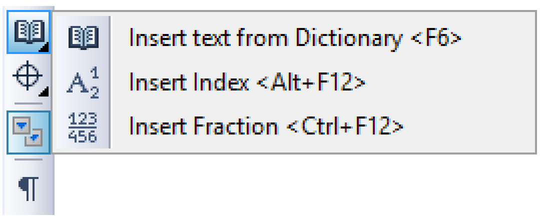
To start **variable editing mode**, simply point the mouse to the inserted variable and click. Besides, this mode can be invoked from the context menu by right clicking over a paragraph text.

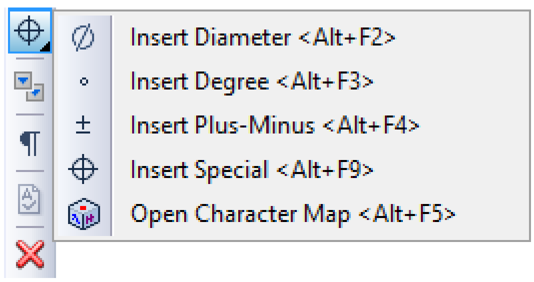
The paragraph text containing the chosen variable will be selected as if in the text contents editing mode. However, available for editing will be only the variables inserted in the text and flagged as "Editable as field".

At this moment, the following icons will appear in the automenu:

	<Ctrl+ Enter>	Finish Variable change
	<Ctrl><X>	Cut
	<Ctrl><C>	Copy
	<Ctrl><V>	Paste
	<F6>	Text...
	<Alt+F2>	Special...
	<Ctrl><F3>	Show "Select from List" buttons
	<Ctrl><F2>	Show non-printing symbols
	<Esc>	Cancel Variable change

The options and allow the user to insert the text from the dictionary, indexes, fractions and special symbols into the expression of the edited variable. Both options contain the lists of embedded icons. Any of them can be shown in the automenu. Usually it is the icon of the option which was invoked the most recently.





When using the options of insertion into the expression of the edited variable, the special symbols of formatting by rules adopted for the string text are automatically added (see section "Subscript and Superscript Text. Use of Variables in Text").

The options , , are provided for handling the selected portion of the current variable value. As the contents of the clipboard are being pasted, the system checks the correspondence between the types of the clipboard contents and the current variable. Therefore, it is impossible to insert a character string into a numerical variable.

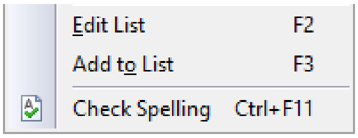
Besides, the system checks for presence of the tab character dividers in the text being inserted. If the text being inserted into the field of a textual variable contains the tabulation characters, it will be processed as follows. The portion of the text before the first tabulation character will be inserted in the current variable. If there are more textual variables in this paragraph text, then the next portion of the clipboard text will be automatically entered in the field of the next variable, and so on until all variables or the clipboard contents are used up.

The option toggles the display of the graphic buttons for selecting from the predefined list of values.

The option toggles the display of the formatting marks (tab, new line, etc.) in the text being edited.

To confirm the changes you have made, press the option or click the mouse outside the text area. The option allows quitting the mode without saving the entered changes.

A provision is made for the variables with a predefined list of values stored in a file for editing or adding values to the list without entering the variable editor. Simply right click and select the respective item in the coming up context menu.



Creating and Editing Multiline Text

The Multiline Text, as opposed to the paragraph text, is always located in one rectangular area, whose boundaries extend as the text is being input until the user presses "Enter" for wrapping the text to the new line.

To create a multiline text, use the command **TE: Create Text.**

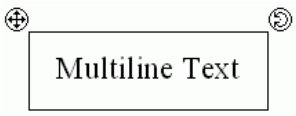
When creating a text, the parameters are automatically used that were set as the default parameters (the option). Originally, these parameters use the settings “from document”. The text parameters will be applied to the whole content of the given text.

When editing the contents of the text being created, you can assign specific parameters to its separate elements, for example, to an isolated word or sentence.

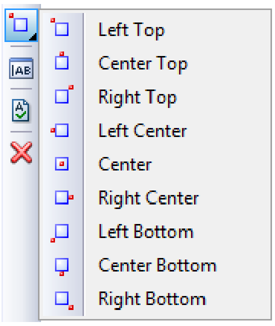
To create a multiline text, select the option:

	<M>	Create Multiline Text

Next, you need to specify the text attachment point. This can be done in three ways: by pointing the mouse anywhere in the document and clicking , or by specifying the absolute coordinates using the option , or else by selecting a node for attaching the text. A rectangular area will appear on the screen with a blinking cursor. You can now start inputting the text.



The options for creating a multiline text are similar to the options for the paragraph text creation. An exception is the additional icons for defining the ways of vertical positioning of the text with respect to the attachment point.

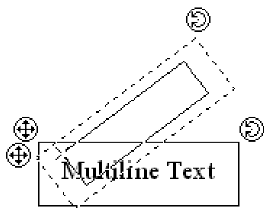


When creating a new text, an icon is always present in the automenu that indicates the default attachment. (This attachment is defined in the text parameters before the text creation.) Usually, this is the attachment at the upper left corner. The list of the enclosed icons will appear if you press and hold for a short while.

Selecting one of these icons will define the position of the text being created or edited with respect to the attachment point as follows. Selecting one of the **top** attachment settings positions the text below the attachment point; selecting one of the **bottom** settings positions the text above the attachment point; one of the **center** settings centers the text with respect to the attachment point.

The selected icon will be displayed in the automenu. The icon displayed in the automenu will be applied by default to newly created multiline text. When editing a multiline text, the icon is displayed in the automenu that corresponds to the attachment type of the text being edited. An attachment type for the text can be set without scrolling through the list of the enclosed icons. You can simply keep pressing the option in the automenu. That will rotate the icons and, therefore, the attachment type of the text with respect to the attachment point.

There is another difference of the multiline text from the paragraph text. Moving and rotating the rectangular text area can be done with respect to the text attachment point, which is marked by a cross when rotating.



Multiline text editing is similar to editing the paragraph text, except for the options provided for editing the rectangular text area.

Creating and Editing Tables

Table creation is based on the multiline text. The command used for this purpose is **TE: Create Text.**

Editing tables is none different from its creation. The table editing options are similar to the options of the multiline and paragraph text, except for the options for editing the rectangular text area.

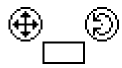
A table can be defined in two ways. One way is by inserting the table in a text (the paragraph text or multiline text). In this case, the table will be placed inside the text. Alternatively, you can create a stand-alone table. In the latter case, you cannot type outside the table borders. Select the option:

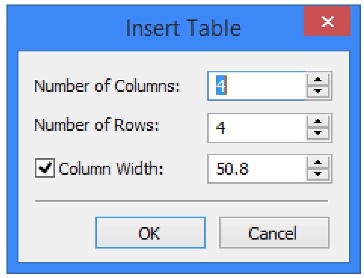
	<B>	Create Table

Before creating the table, you can set the default parameters (the icon) that will be applied to all newly created tables. The default color defines the font color for typing in the table and the table border color.

Next, specify the table attachment point. This can be done in three ways. One way is to click the mouse at an arbitrary place within the document. The other way is to specify the absolute coordinates using the option .

Yet another way is to select a node for attaching the table. An empty rectangular text area will then be displayed, along with a dialog box, where you can specify some parameters of the table being created. The buttons set “Text” will be displayed on the main toolbar (if it is not locked).





In the dialog box, set the desired number of columns and rows, as well as the combined width of all columns (individual column width can later be defined among the table properties). If the parameter **Column Width** is not set, then the system will automatically set the minimal column width. Upon confirming the specified parameters, the table being created will be drawn in the rectangle. If necessary, you can resize the cells by dragging their borders with the mouse.

Special handles are provided for moving and rotating the table just like the multiline text.

To start typing in the table, place the textual cursor in one of the table cells, and then enter the text.

The text creation options are similar to those for creating the paragraph text. The options defining the table attachment are similar to the attachment options for the multiline text. A table, just like a paragraph text or a multiline text, can be edited in a separate window.

> A table's contents can be created by copying the contents of an already populated table or a Word table via the text clipboard. Vice versa: a T-FLEX CAD table can be copied into MS Word.

The following manipulations can be done with the help of the options on the main toolbar (button set “Text”):

Insert Table <F12>. Inserts a table in a text (the paragraph text or multiline text). A dialog box appears, in which you can define the number of columns and rows in the table (see above). This option is available only if the table is inserted in a text or if the parameter is turned off, "Disable text input outside of Table" (see table properties/the tab “Table”).

Split Cells. Splits the selected cells (or the cell with the cursor) into the specified number of rows and columns.

Merge Cells. Merges the selected cells into one.

 **Split Table**. Splits the table into two separate tables. The split is made above the current row, if this is not the first row in the table. This option is available only when the table was inserted in a text or the parameter “Disable text input outside of Table” was turned off when creating the standalone table (see table properties/the tab "Table").

 **Insert row before current**. Inserts an empty row before the current row.

 **Insert row after current**. Inserts an empty row after the current row.

 **Insert rows**. Inserts a specified number of empty rows before or after the current row.

 **Delete rows**. Deletes the selected rows (or the row of the cursor).

 **Insert Column Before**. Inserts an empty column at the left of the current column.

 **Insert Column After**. Inserts an empty column at the right of the current column.

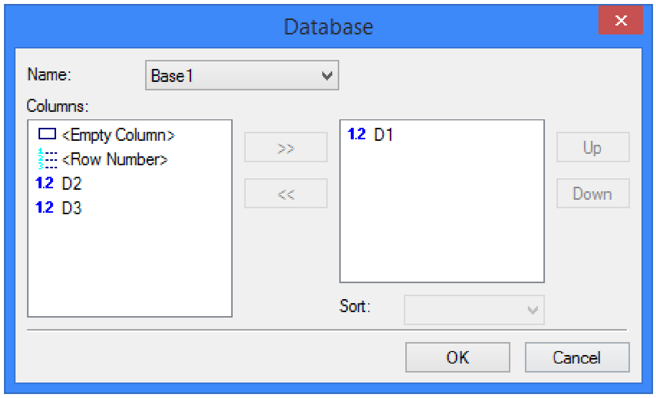
 **Insert Columns**. Inserts a specified number of empty columns before or after the current column.

 **Delete Columns**. Deletes the selected columns (or the current column).

 **Database**. Inserts the contents of the internal database or a reference to a database into the table. Upon calling this option, a dialog box appears on the screen that allows selecting the name of one of the existing databases. Then, the list of the database fields appears in the “Columns” pane of the database.

The data type (integer, real, text) is marked left of the field name. To enter the contents of a database field into a table cell, select the field name and press the graphic button **[> >]**. The field name will then be carried over into the right pane to prevent a repeated selection (except for the field keeping the ID number of the database record).

The number of the selected database fields may not be greater than the number of the table columns. The order of the field names in the list of selected corresponds to the order of filling in the table columns (the first field contents is entered in the first column, and so on). To delete data from a column, select the respective field name and press the graphic button [< <]. To modify the order of the data in the table, use the graphic buttons [Up], [Down]. A sorting rule can be assigned to any particular database field, except for the record number (none, ascending, descending). Upon confirming the selected fields by pressing the [OK] graphic button, the table will be filled with the respective values from the database.



If the database contents were changed, the table contents can be refreshed using the option:

 **Refresh from Database**. Refreshes the table contents according to the changes in the database.

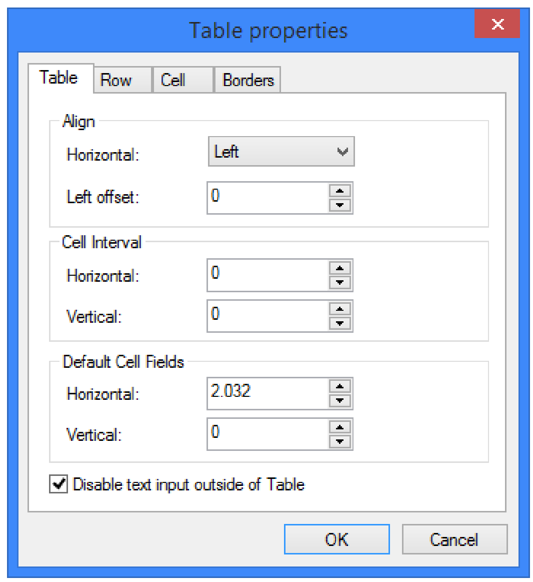
 **Select Table**. This option selects all cells in a table.

 **Table Properties**. Calls the table properties dialog box.

Table Properties

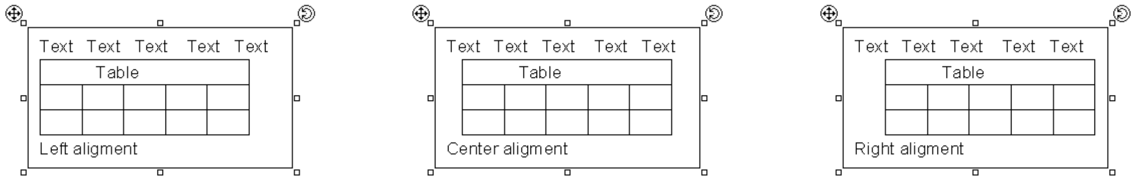
"Table" tab

This tab allows defining the parameters that affect the whole table.

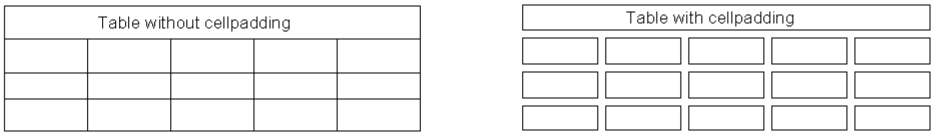


Align - Horizontal. Aligns the table horizontally within the text rectangular area. The alignment options are **Left**, **Center**, **Right**. The "left" alignment option enables the **offset** parameter that defines the distance from the left side of the text rectangle to the table.

Since standalone tables are created based on the multiline text, the dimensions of the text rectangular area depend on the input data. The values of this group of parameters are meaningful (and displayed) only in the case when some text was entered outside the table, thus increasing the size of the text rectangular area as compared to the table size. (See the description of the parameter "Disable text input outside of Table".)



Cell interval. This parameter defines horizontal or vertical padding of the cells.



Default Cell Fields. Sets the amount of padding between the text being input and the cell boundaries. This parameter affects all table cells except those whose padding was defined separately on the tab "Cell".

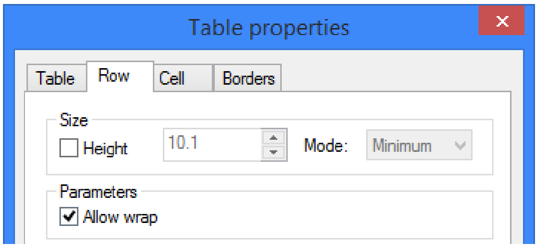
Disable text input outside of Table. This parameter allows creating standalone tables. If not set, then the system is in the multiline text creation mode that allows inputting text outside the table. In this way, the table being created will be inside a multiline text. Once at least one character is typed outside the table, this parameter becomes inaccessible. If the table appears in the beginning of a text and you need to type a text before the table, place the text cursor in the beginning of the first table cell and press <Enter>. To return to creating a standalone table, delete all character typed outside the table, leaving outside just one empty line, and then set this parameter. (The tables that you insert in a text when creating a multiline or paragraph text, have this parameter turned off.)

"Row" tab

The **Size** group of parameters defines the height of the table rows:

Height. Sets the height value of the selected rows in the table. If this parameter is not set then the height of the rows is defined automatically depending on the maximum font height (by the printed characters), also accounting for the maximum amount of padding across the cells in a row.

The parameter **Height** is not set for the rows of a newly created table and for rows that were added or created as a result of splitting cells.



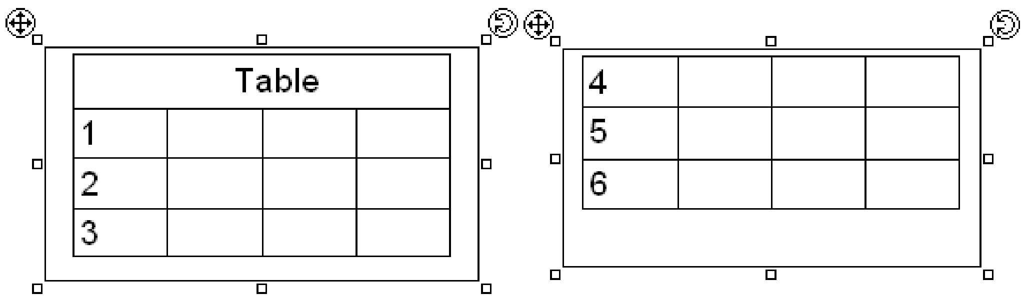
Mode. This parameter controls the specified value of the row height depending on the kind of text being input:

Minimum. Sets the minimum value of the row height. If the text doesn't fit by the height of the row, the row height is automatically increased.

Exact. Sets a fixed value of the row height. The text that does not fit by height is not displayed and will be visible only if you increase the height of the row.

Multiple. As you input the text, the height of the row will always be a multiple of a specified value.

Allow wrap. Allows carrying the rows of the table over from one text rectangle to the next one, and, consequently, from one page to the next one (in the case of creating or using an existing table within a paragraph text).



This parameter is set by default for all rows of the newly created tables, as well as the rows being added or created as a result of splitting cells. A row, for which this parameter is not set, will be kept together with the next row. As the table rows are carried over from one text rectangle into the next one, these rows will be carried over together.

"Cell" tab

Column width. Sets the width value of the selected columns.

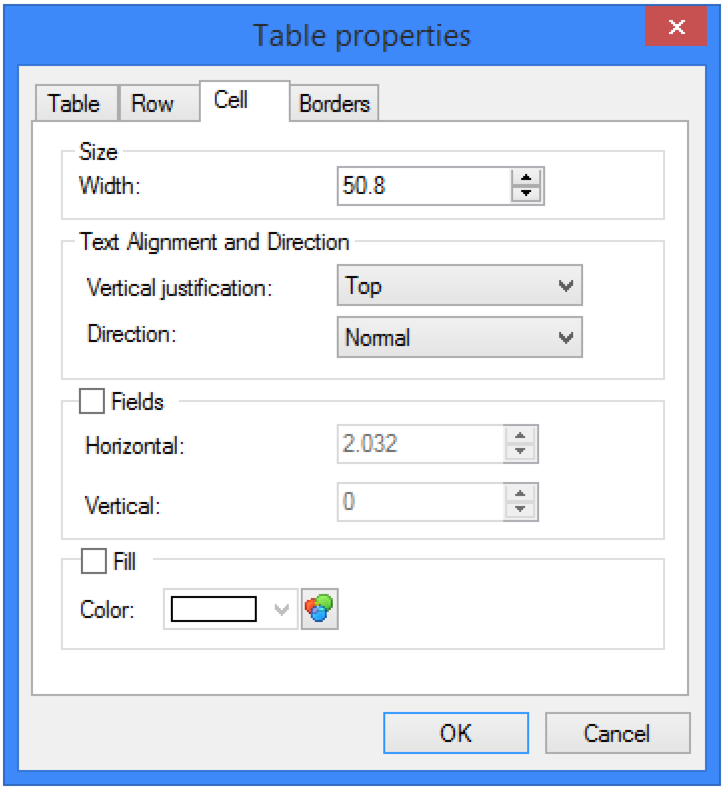
The group of parameters **Text Alignment and Direction** defines text alignment and direction inside each cell:

Vertical justification. Defines the vertical alignment of the text inside the selected cell. There are three alignment options – **Top**, **Center**, **Bottom**.

Direction. Defines the text direction inside the selected cell. Four direction options are available – **Normal** (from left to right), **From Bottom to Top**, **From Top to Bottom**, **From Right to Left**.

Fields. Defines the text padding (**Horizontal**, **Vertical**) inside the selected cells. If the parameter is not set, the padding values are used that were set for all table cells on the tab “Table”.

Fill. Defines the fill color of the selected cells.

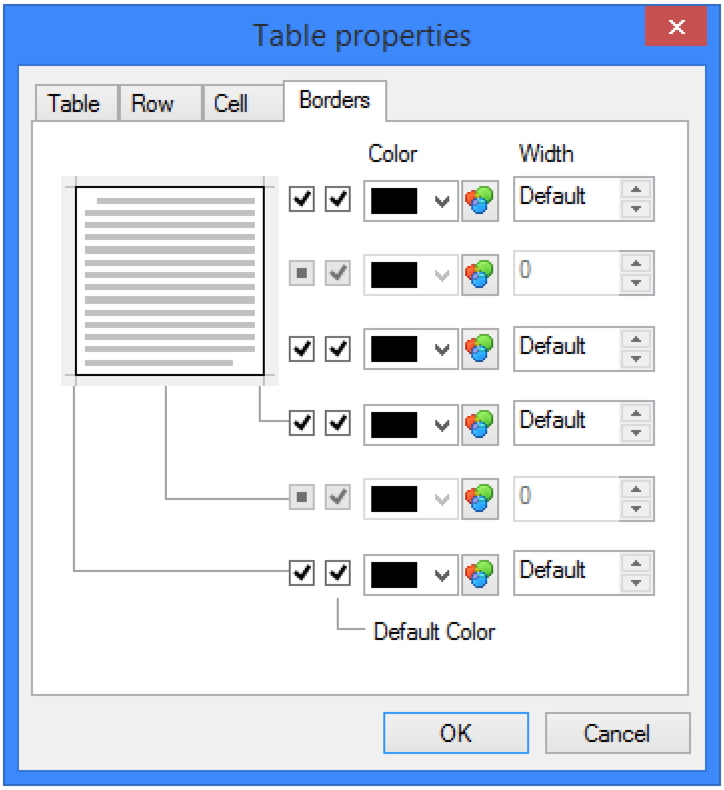


"Borders" tab

This tab allows modifying the borders of the selected rows, columns, cells or the whole table.

Each border has a group of attributes that toggle the border display altogether and define the color and the borderline width. The attributes are placed next to the respective borders or pointed at by the witness lines.

The first check box toggles the display of the border. The second one sets the border color to the default defined among the text parameters before the table creation. To set a border color different from the default, uncheck the item and select the desired color from the color list (the check is cleared automatically when selecting a color from the list). To restore the default border color, simply check the box again.



The parameter **Width** defines the value of the borderline thickness. Its default value is copied from the drawing parameters **ST: Set docement parameters** on the **Lines** tab **Other Lines** parameter.

A preview pane is provided on the tab that reflects on the border parameter changes as the textual cursor is replaced from one cell into another.

Working with Dictionary

When creating drawings, you often need to add text elements that require multiple entries or repeat from a drawing to a drawing (such as, for example, technical specifications). T-FLEX CAD makes a provision for this case by including in the installation a standard dictionary with a set of technical requirements. The dictionary is based on a database complying with the Microsoft Access (*.MDB) format, represented by the file *...T-FLEX CAD\Program\TFDict.mdb*. The user can create custom dictionaries or modify this standard dictionary by manually adding or deleting elements from it. To create a new dictionary file, make a copy of the standard dictionary and save it under a different name, and then modify as desired.

The text elements from the dictionary can be inserted in all types of the text (the string, multiline, paragraph text, table).

The dictionary can be opened while within the text creation and editing commands. To do this, engage the option **Insert text from Dictionary <F6>** with the respective automenu icon  or select the context menu item **Insert/Text...** accessible by right clicking. The dictionary window is always on top of other windows.

The following commands are available on the toolbar in various modes*:

Open. Opens the dictionary file. You can open the file for editing or in the "Read Only" mode. The mode is set in the File Open dialog box.

Add. Creates a folder in the structure pane or a new line of text. A folder or text can be created from the context menu.

Delete. Deletes the selected objects from the dictionary.

Rename. Renames the selected objects (the names of folders or lines of text). You can activate renaming from the context menu or by clicking on the selected object.

Sort. Sorts the current list alphabetically.

Move Up. Moves a line of text up the list.

Move Down. Moves a line of text down the list.

Keep visible. Allows continuously inserting several elements. The dictionary window is not closed after inserting a text. You can work simultaneously with the text and the dictionary.

Always on Top. While the option is active, dictionary window will be on top of all other windows.

Insert into Editor. Inserts the selected text in the drawing (see the description below).

Copy.

Paste.

Create formatted text. Calls the text editor (see the description above), where you can specify formatting parameters of the existing text or input new one. The formatted text cannot be edited in the dictionary dialog box. Instead, use the following command:

Edit formatted text. Calls the text editor window where you can edit formatted text. You can start the text editing mode by double-clicking in the editing pane located in the lower right corner.

Delete formatted text. Deletes all text formatting. Such text can be edited in the dictionary dialog box.

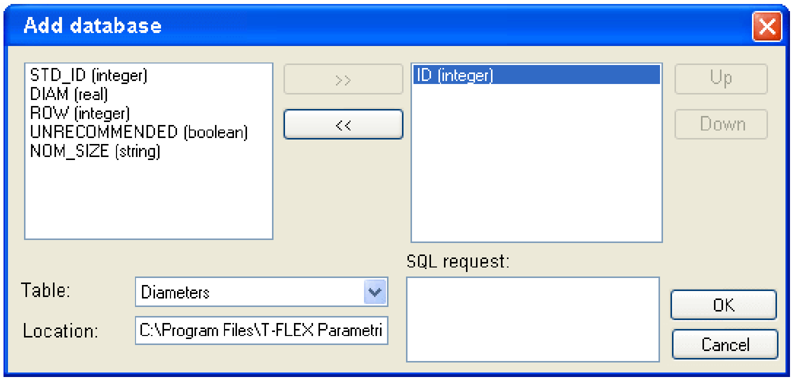
Add reference Database. This option allows including in the dictionary the contents of a database fields (*.mdb). After calling the option, a file browser appears on the screen for selecting the desired

* If the dictionary file is open in the " Read Only" mode, then the editing commands are inaccessible.

database. Note that adding a group with the selected database is done on the current level of the dictionary structure. Upon the creation, the database reference group can be renamed (the default name is composed of the path and the database filename) or moved to another level. Unlike other dictionary folders, the contents of a database by reference cannot be edited or converted into the formatted text. The next step will be defining the database parameters (at creation, this dialog box appears automatically).

Edit database parameters. The dialog box appears as shown on the diagram.

The left portion of the dialog contains the list of all fields of the selected table in the database (the table is selected from the list in the item "Table"). In the right hand side pane of the dialog the user places the fields to be used in the dictionary. To add a field, select it in the left pane and press the graphic button [>>].



To delete a field selected in the right thing and press the graphic button **[<<]**. The graphic buttons **[Add All]**, **[Delete All]** do this with all elements of the field list. The selected database fields will be placed in the dictionary window in the order they were in the right pane of the dialog. To modify a field position, select it in the right pane of the dialog and move within the list using the buttons **[Up]**, **[Down]**. The SQL Request field is provided for specifying the selection condition for the fields and records in the database.

Besides, you can use the commands from the main dialog menu:

File/Add from… Allows merging the database file you've specified into the current dictionary database file.

File/Save as… Allows saving the current dictionary file under a different name.

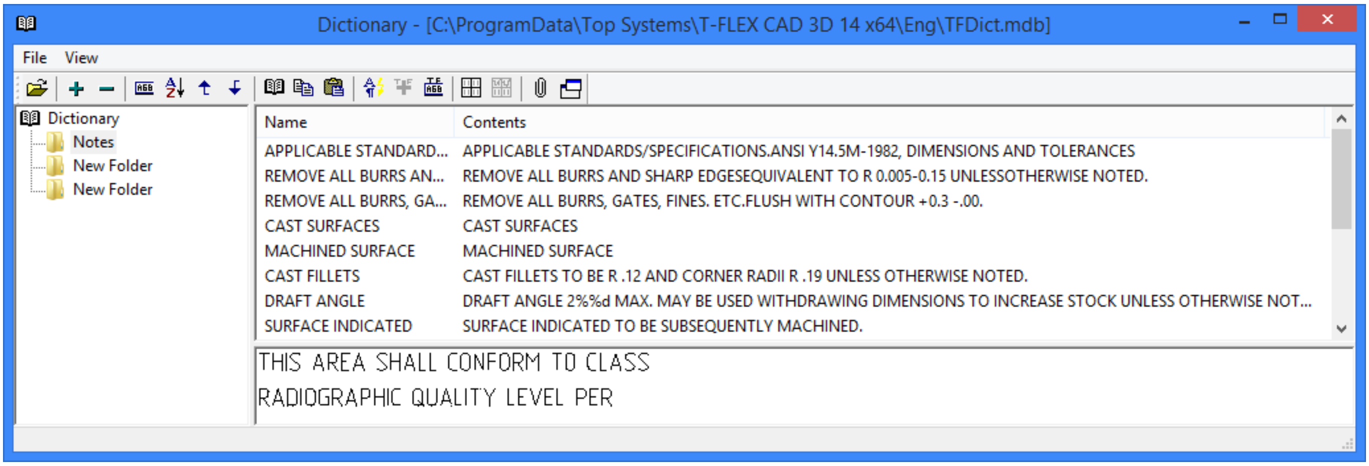
File/Close. Closes the current dictionary file.

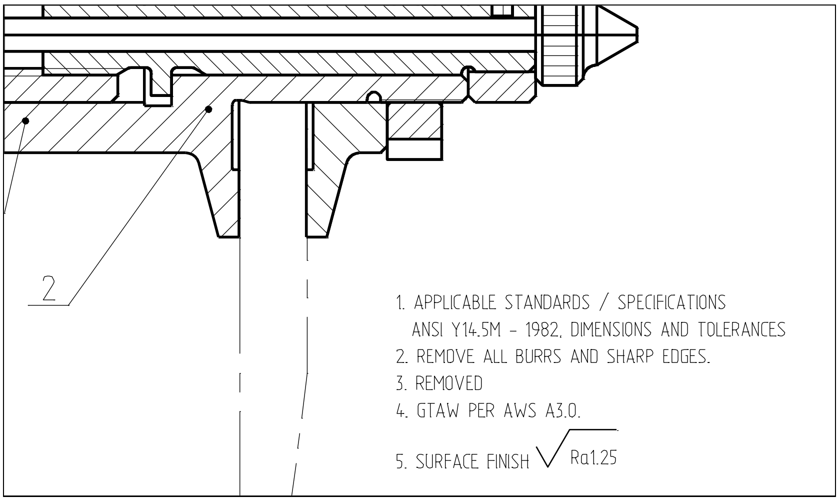
File/Exit. Closes the dictionary dialog box.

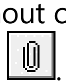
The dictionary supports "Drag & Drop" mode for copying and moving records or folders. Point the mouse to a record or folder, and then depress the left mouse button and hold while dragging the selected element to the new place.

Inserting Text from Dictionary

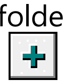
In the dictionary structure pane, select a group folder. The contents pane will then display the list of text elements, each one being named. The text elements are selected by their names. You can modify the element contents in the editing pane (the lower right pane). Next, select the desired line from the list by double-clicking or press for inserting the selected line. At this moment the text is inserted in the drawing, while the dictionary window closes. To keep inserting the same text, you can copy it on the clipboard by pressing <Ctrl+V>. If only the name is defined for an element, while the contents are empty, then the element name will be inserted in the drawing text.

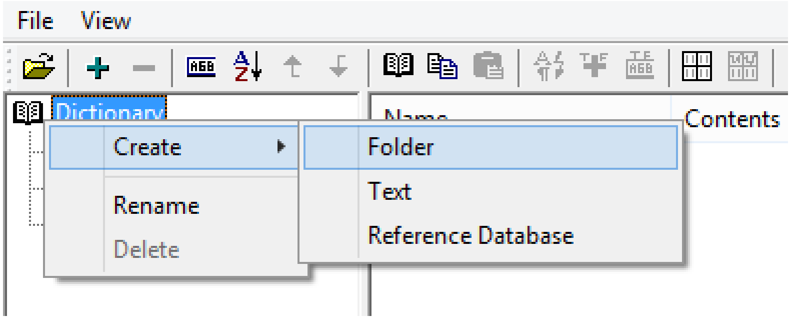




If you need to work with the text without closing the dictionary window, use the option "**Continuous text input**", or press the respective button .

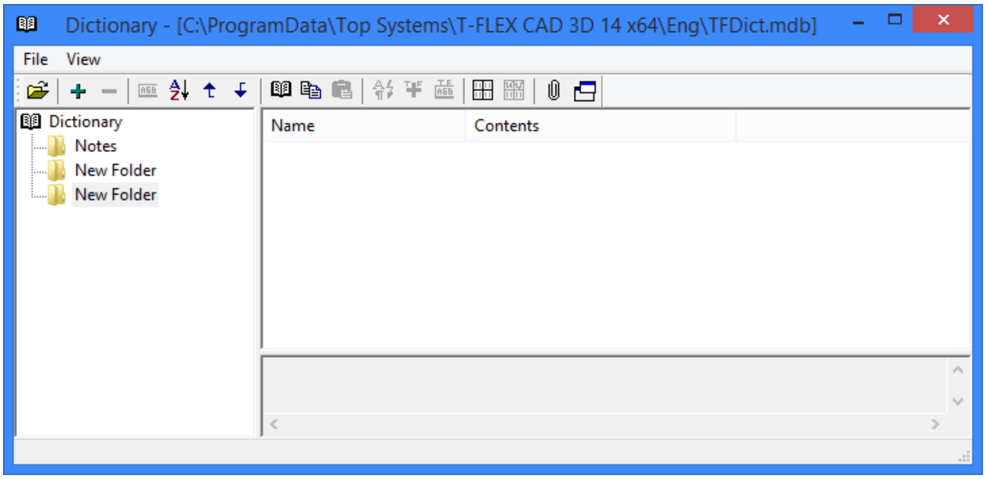
Creating New Records in the Dictionary

In the left pane, select a header in which you want to create a new folder or text. Select, for example, the folder "Dictionary". To add a subfolder to this folder, press the icon  or select the respective context menu item, invoked by the right mouse click (Create/folder).

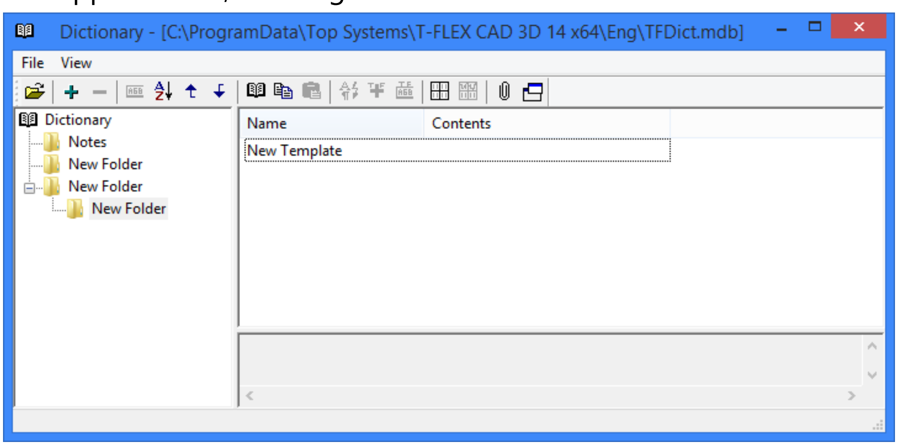


As a result, a new subfolder will be created in the folder "Dictionary".

The subfolder name will be highlighted for editing, so that you can enter its desired name.



To enter a text in the folder, first select the folder. In the context menu accessed by right clicking, select the item ("Create|Text"). Alternatively, upon selecting the folder, click in the upper right pane of the dialog and press the icon. As a result, a new line of text will appear in the contents pane, that can now be named. To enter the contents in this text line, move the pointer to the lower pane and click. The textual cursor will appear there, allowing to enter the desired text.

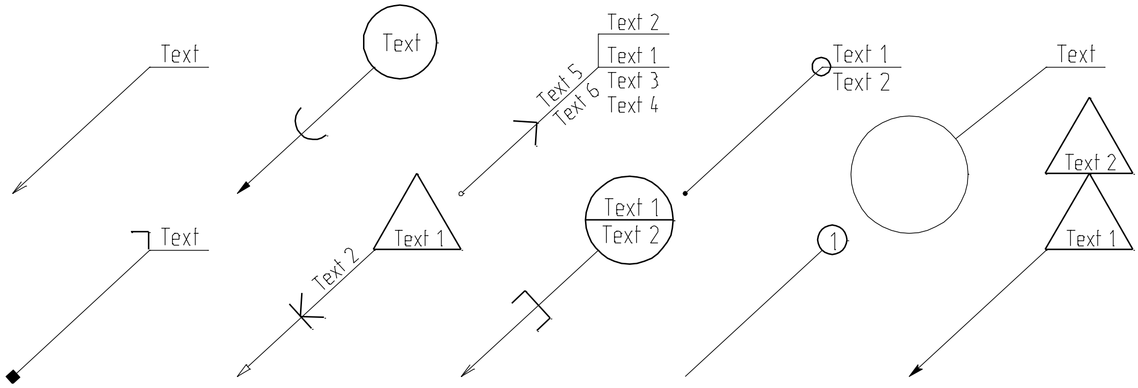


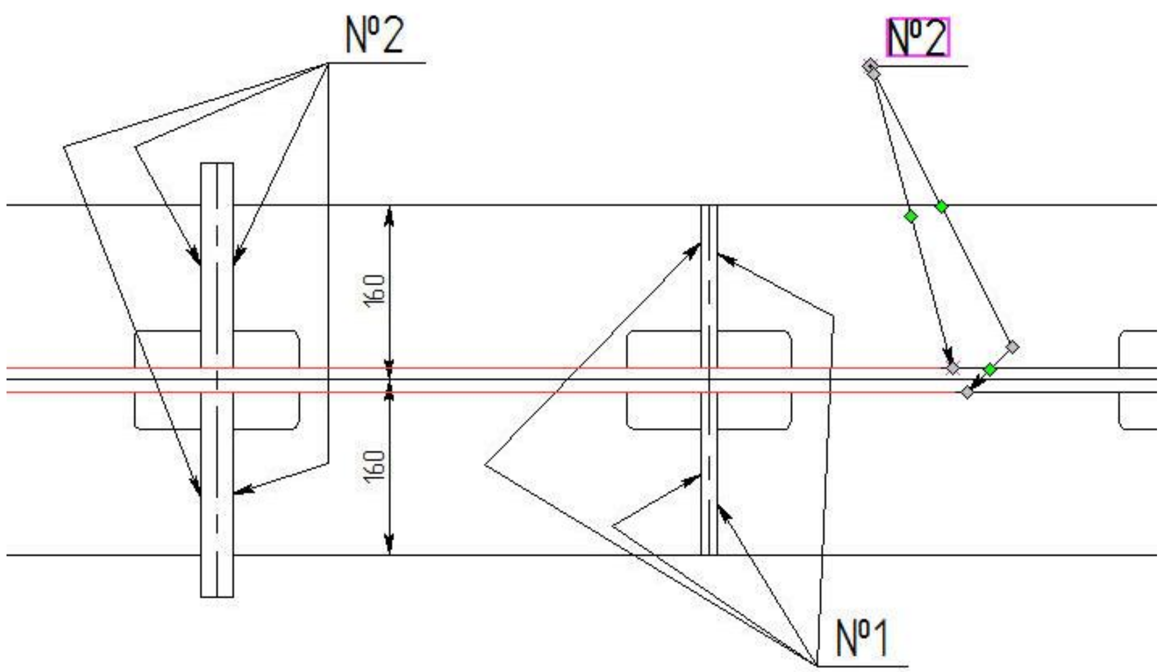
Leader Notes

Leader notes are used for decorating various drawing elements. Those mark BOM items, specification of BOM positions points to the locations of brand marks, part codes, etc. A leader note is composed of two parts: the witness line (arrow) and the jog (the leader).

To apply a leader note, you need to specify the position of both parts. Consequently, a leader note has two attachment points. If necessary, it is possible to create leader notes that contain several extension-arrows including bent ones.

Depending on the specified parameters, you can get various detailing elements.





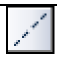
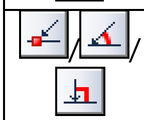
Creating Leader Notes

Leader node creation is done in the command **IN: Create Leader Note**

Icon	Ribbon
	Draw → Title Block → Leader Note
Keyboard	Textual Menu
<IN>	Construct > Leader Note

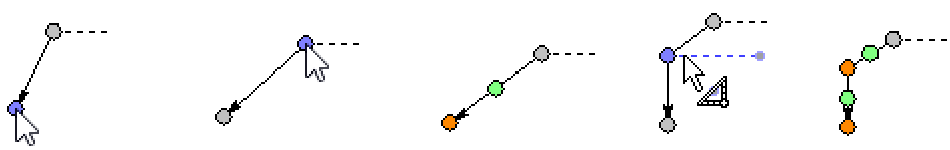
Creating Simple Leader Note

After launching the command of the leader note creation the following set of options is available in the automenu:

	<Enter>	Set a leader note attachment point at the position of the pointer
	<Alt+P>	Copy Properties from Existing Element
	<P>	Set Leader Note Parameters
	<L>	Set relation with Line
	<U>	Fixing to Point/Angle Fixing/Orthogonal fixing
	<N>	Select Node
	<Z>	Change Leader's Orientation
	<F4>	Edit Leader Note
	<Esc>	Exit command

To create a leader note, you need to specify subsequently the positions of two points. The first point determines the position of the witness line - the leader arrow, the second - the position of the jog.

The third point appears after the two points positions are specified. The point allows to specify a break. You can create several breaks for the leader note.

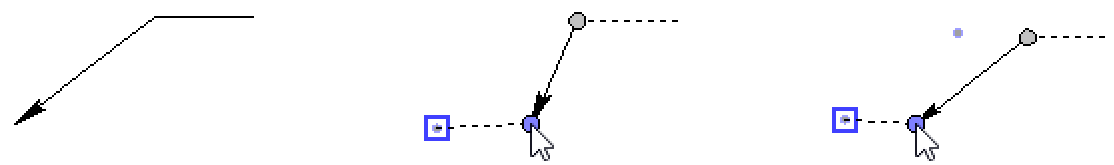


When creating the leader note, a dynamic view is fixed to the cursor that entirely shows the future leader note's view.

After specifying the first snapping point, additional option will appear in the automenu:

	<F>	Tie to Arrow

The position of the second point can be defined relative to the first point of the leader note attachment (relative to the arrow) or in the absolute coordinates. To select the desired mode, use the option . With the option turned on, the position of the jog is defined relative to the leader note arrow, otherwise – in the absolute coordinates. Later on when translating the first point of the leader note, created with the enabled option , the entire leader note will be translated, and for the leader note created with the disabled option , the leader will remain in the initial position



Option disabled

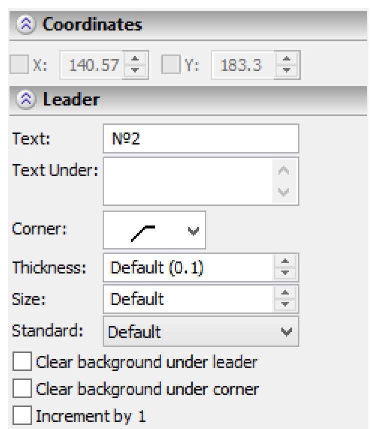
The second point don't move

Option enabled

The second point moves with the first

Leader note parameters are specified in the command's properties window before completing the leader note creation with the help of the option

Note that after specifying the last point of the leader note's arrow the focus is automatically shifted to the text editor. If we press <Enter> in the text input editor - leader note's creation will be completed.



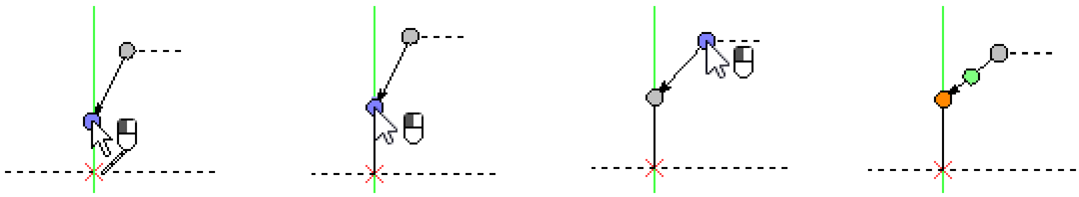
Leader note's parameters include the contents of the text lines, style of the leader note lines, font style parameters etc. In addition, in the properties window you can specify precise location of the leader note's snapping points (by specifying the absolute coordinates of the points or their shifts with respect to the selected snapping elements).

As with creation of the dimensions, leader note's parameters can be copied from already existing leader note-element with the help of the option. This option is described in more detail in the "Dimensions" chapter.

Snapping Leader Note to Drawing's Elements

Both leader note points can reference a node or a graphic or construction entity. You can select the desired element with the help of directly in the drawing's window. To select lines and nodes it is also possible to use the and options. When using this mechanism, keep in mind that attaching the second point to a construction line or node is possible only when the option is turned off. To cancel of the active attachment-defining mode, press <Esc> or right click.

To attach a leader note point to a construction entity, first select the desired entity (by clicking or using the respective option). The entity will be highlighted, and a leader note will start rubberbanding along the entity, following the pointer. Now, specify the position of the leader note attachment point on the entity by clicking.

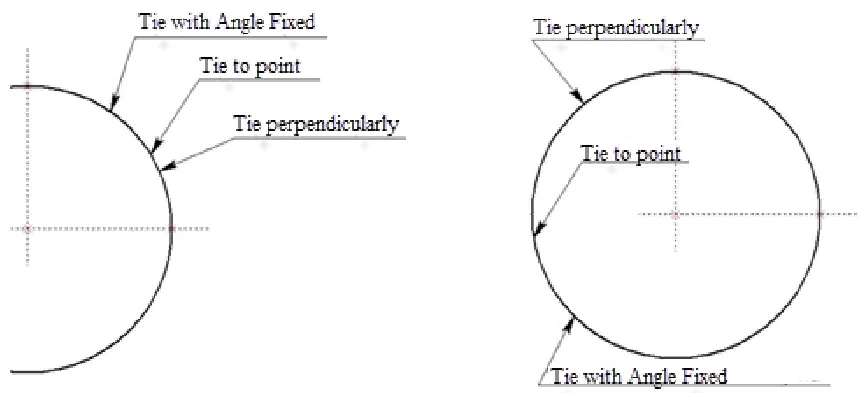


When attaching a leader note to a straight line, at least one node must exist on the line.

Attaching to a graphic entity is similar to attaching to a construction entity. In the case when the leader note point is beyond the limits of the graphic entity, it will be positioned on the continuation of this graphic entity.

The , , options define the rules of changing location of leader and leader arrow when location of the snapping point is modified. These modes affect only the leader notes that are snapped to lines. When snapping to points there will be no effect if these modes are selected.

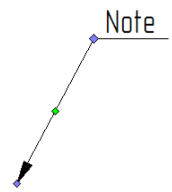
If the leader note is created with the use of the option, then upon translation of the leader note along the line it is translated "rigidly" (without changing angles). The option allows us to keep the arrow perpendicular to the selected snapping line. The option allows us to keep the angle that the arrow makes with respect to the selected snapping line.



Markers for Controlling the Snap of Leader Note’s Elements. Creation of Arrow with Bends

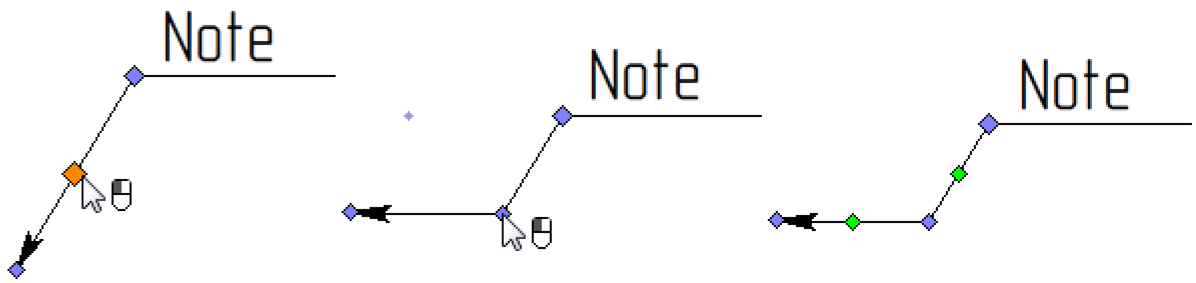
After preliminary snapping of a leader note, location of its snapping points can be modified. It is also possible to change the shape of an arrow by creating on it any number of bends. To control snapping of leader note’s points, and also for specifying bends on its arrow, special markers are used.

At the beginning of leader note creation, there are only two markers – they designate the main leader note’s snapping points and are highlighted with blue color. After specifying location of both points, one more marker will appear on the leader note’s arrow image, marked with a green color. This is a point in which an arrow bend can be created.

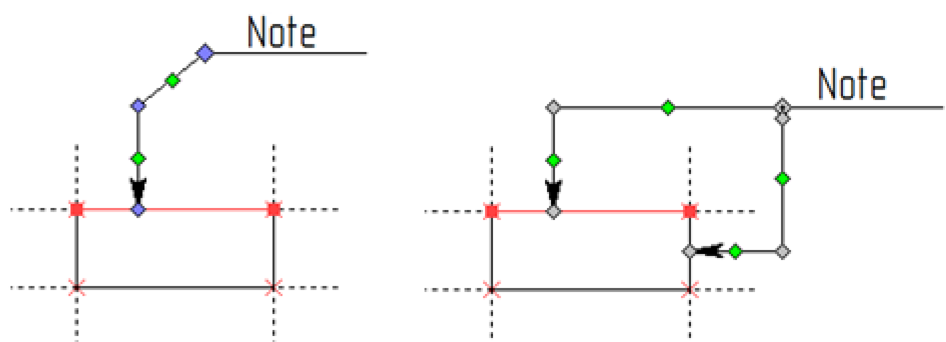


To correct location of any snapping point of the created leader note, it is sufficient to move the cursor closer to the marker of the desired point (it will be highlighted with orange color) and select it with the help of . The point will start moving after the cursor. In this way, for example, it is possible to specify the shift when snapping the leader note’s point to the node.

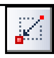
To create a bend, it is required to move the cursor closer to the green marker (it will also be highlighted with orange color) and select it with the help of . The arrow will be split into two segments, bend point – arrow bend point – will start moving after the cursor. Pressing for the second time will fix the location of the arrow bend point.



After creation of the first bend point, new green markers will appear on the two segments of the arrow. If necessary, it is possible to create the next bend at each of these markers. When arrow creation is completed, you can start creating the next arrow, or complete leader note creation with the help of the option.

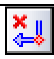


Snapping of each bend point with respect to the previous point of the arrow is controlled by the option:

	<F>	Tie to Arrow

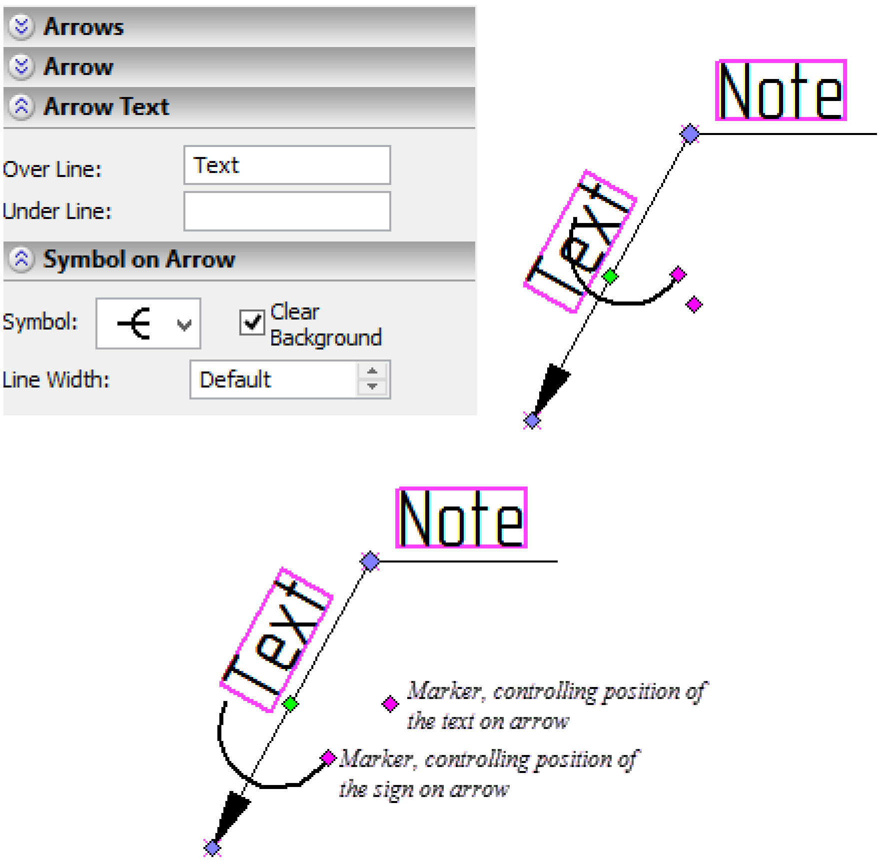
When this option is enabled, the bend point location is specified with respect to the previous characteristic point of the arrow; when this option is disabled – in absolute coordinates.

To remove a bend point, it is required to select it and use the option:

	<Del>	Delete Node

Markers (Manipulators) for Controlling Location of Symbol and Text on Arrow

In the leader note's parameters it is possible to specify the text and symbol on the arrow. Location of these leader note's elements can be controlled with the help of the special markers/manipulators.



After selection of the desired marker with the help of 🖰, the symbol or text will start dynamically move after the cursor. You can fix its location by pressing 🖰 for the second time.

Creating Additional Arrows

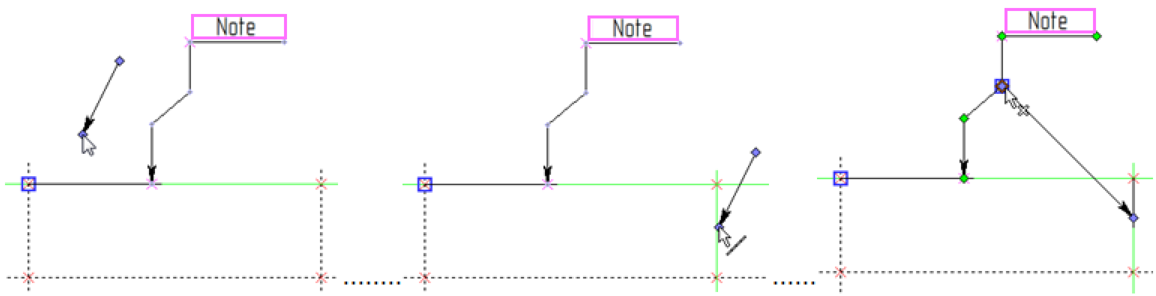
After specifying location of the main arrow and the leader of the leader note, additional option appears in the automenu:

	<Space>	Add Arrow

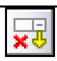
As a result of invoking this option, the image of the new arrow will appear next to the cursor.

When creating additional arrows, the same rules will hold as for the main arrow of the leader note. It is possible to tie them to lines and nodes, set relation with dimensions, form bends. In the properties window it is possible to specify individual parameters of the arrow which are different from the ones specified for the main arrow of the leader note.

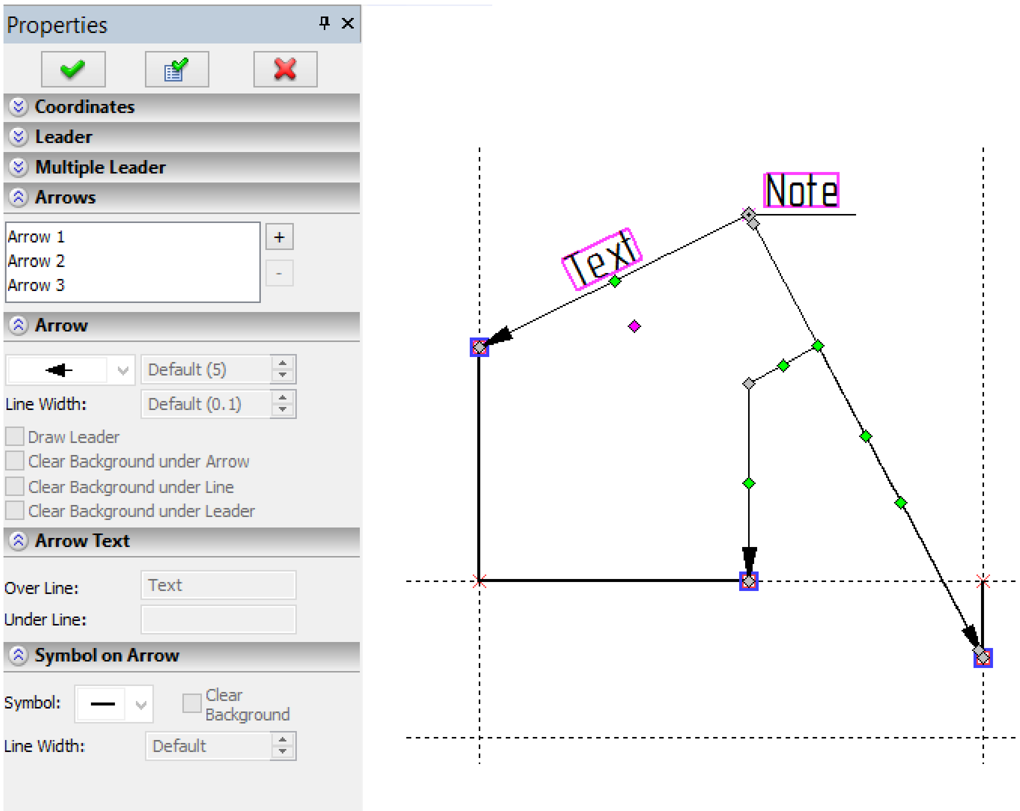
The end point of an additional arrow can be tied to any characteristic point of the main leader note. To do so, it is sufficient to indicate one of the markers on the leader note image as a snapping point for the arrow's end.



You can refuse creation of the new arrow by using the option:

	<Del>	Delete Arrow

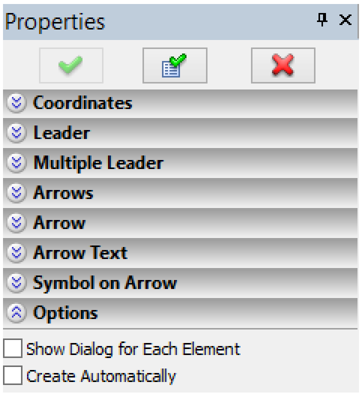
The list of arrows created for the leader note is displayed in the properties window.



The [+] and [-] buttons to the left and right of the arrows list can be also be used for creation and removal of additional arrows of the leader note.

Automatic Completion of Leader Note's Creation

By default, after specifying location of two main leader note's points (locations of arrow and leader), the command remains in the mode of editing the created leader note. This allows us to add to the leader note the desired number of additional arrows, create bends on the arrows, specify and edit leader note's parameters at any instant of its creation. After finalizing the leader note formation, it is required to explicitly complete its creation with the help of the  option or with the similar button in the command's properties window.



It is also possible to use another work algorithm. In the command's properties window, in the "Options" section you can find the **Create automatically** flag. When this flag is enabled, creation of the leader note will automatically be completed right after specification of the second snapping point of the leader note (the leader's snapping point).

Note that in this case it is required either to specify parameters of the leader note being created before snapping of the leader or additionally enable the **Show parameters dialog for each element** flag. In the latter case after specifying location of the leader, the leader note's parameters dialog will appear on the screen. This allows us to work as with the previous versions of T-FLEX CAD – first specify location of an element on the drawing and then specify its parameters.

Leader Note Parameters

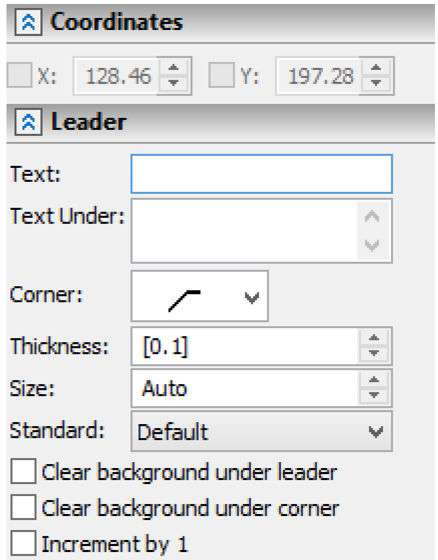
The dialog in the command's properties window contains all main parameters of a leader note. For working convenience, the dialog is divided into several sections.

«Coordinates» Section

The first section, "**Coordinates**" contains the fields to define the exact coordinates of leader note snap points. The current coordinates are dynamically tracked as the cursor moves in the drawing window.

Depending on the method of leader note snapping, different types of coordinates may be displayed in this section. For example, in the case of the free snap used for both points, the absolute coordinates of both snap points are displayed in the properties window. When using free snapping with the jog being snapped to the arrow, the offsets dx and dy will be used for the second point, relative to the first point, etc.

«Leader» Section



The "Leader" section serves to define all parameters of the leader note jog. Those include:

Text. This is the text on the leader note jog.

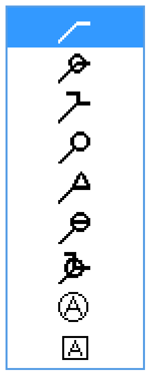
When defining this and other text parameters of a leader note, you can use variables by entering them in curly brackets (braces). Details on that are given for dimensions and text.

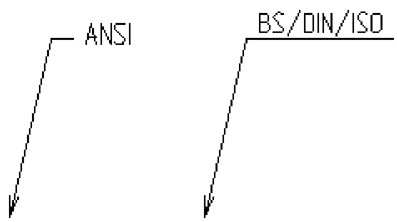
Text Under. This is the text under the leader note jog.

Size. It defines the length of the leader note jog in certain units. If the Auto value is used, it will automatically assume the length of the text line.

Thickness. Defines the thickness of leader lines. If the parameter value is set to Default, it is calculated based on the Line thickness > Other lines parameter defined in the command **ST: Set document parameters** the tab **Lines**.

Corner. Defines the type of the leader jog (see the figure on the right).





Standard. Serves to define the leader style standard. There are two standards for leaders - ANSI or BS/DIN/ISO. If this parameter is set to "Default", then leader notes will follow the standard specified in the command **ST: Set Document Parameters** (**Detailing elements** > **Leader Notes** > **Standard**).

Clear Background. When this parameter is enabled, the drawing image behind the leader jog (jogs) is erased.

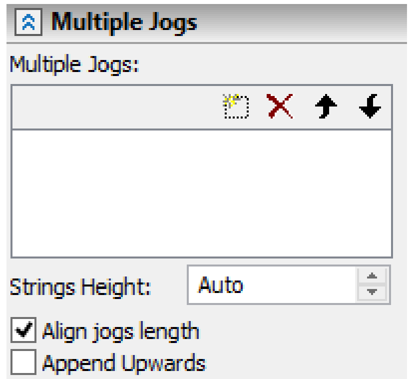
Clear Background under corners. When this parameter is enabled, the drawing's image under the leader note's corner is deleted.

Increment by 1. This parameter is available only at the time of creating a leader note. It serves to quickly define BOM items. A number should appear on the jog instead of text – a BOM item reference number. When the next leader is created, the former current number is automatically incremented by one. It should be noted that when using this approach, the number that appears on the leader will not be related to the BOM position number.

> For details about creating BOM item references, refer to the chapter "Bill of Materials".

«Multiple Leader» Section

Next section of the properties window is – "Multiple Leader" – which contains parameters of additional leader jogs (if any):



Multiple Leaders. This is the field to enter the text on the additional leader note jogs. The jogs in the leader note will be displayed in the order they follow in the window of this parameter.

A new jog can be created by double-clicking on an empty space of this parameter window(at position of the the next string) or with the button. You can modify the order in which leader note jogs follow using the buttons and. To delete a jog, use the button.

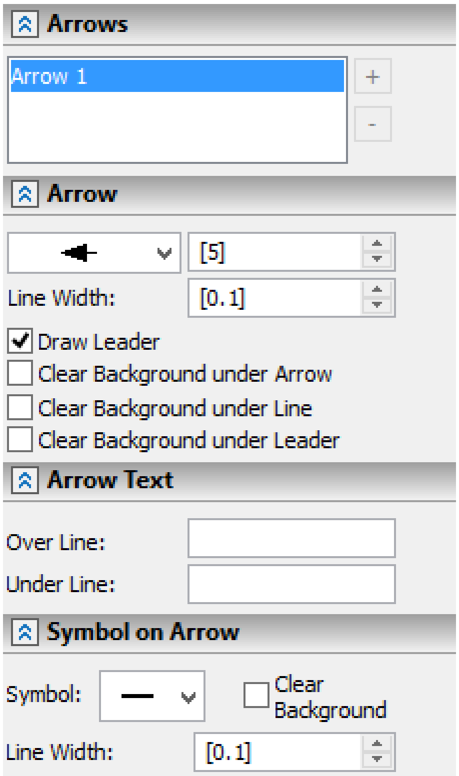
String Height. Defines the distance between jogs in a multiline leader note. When the **Auto** setting is used, the distance between jogs is set according to the font size.

Align jogs length. When this parameter is set (default), the jogs in a multiline leader note are drawn with the same length, otherwise – by the length of text lines of the respective jogs.

Append Upwards. This parameter defines the way of placing additional leader jogs. If the flag is enabled, the jogs will be added on top; if the option is disabled, the jogs are added at the bottom (see figures below).

Sections «Arrows», «Arrow», «Text on arrow», «Symbol on arrow»

The following several sections of the properties window – **Arrows**, **Arrow**, **Text on Arrow**, **Symbol on Arrow** – should be described together. They all together allow us to specify parameters for the leader note's arrows (main and additional).



The **Arrows** section contains the list of all created leader note's arrows. Buttons to the right of the list allow us to create/remove arrows. By selecting one of the arrows in the list with the help of , it is possible to specify parameters of this arrow in the following sections of the properties window.

The **Arrow** section contains the following leader note arrow parameters:

Drop down list for selection of the arrow type at the beginning of extension line. This list mostly coincides with the list of arrows used in the dimension and graphic line creation commands.

However, for leader notes the list is appended with two special arrow types: and . For these arrow types instead of the arrow size specification field (see below) two input fields for specifying the arrow's length and height are displayed. By varying those parameter values, you can get leader notes with a rectangle- or oval-shape tip of arbitrary size and aspect ratio.

Field for specifying arrow size at the beginning of the extension line. If the value is displayed in square brackets, the value can be calculated based on the specified parameter «Arrow (end) size» inside the command **ST: Set Document's Parameters** (the **Drawing** tab).

Line thickness. It defines the thickness of arrow lines. In case when the parameter's value is displayed in square brackets, the value will be calculated based on the specified parameter **Thickness of other lines** inside the command **ST: Set Document's Parameters** (the **Drawing** tab).

Clear Background under Arrow. When this parameter is enabled, the drawing's image under the leader note's arrow is deleted.

Clear Background under Line. This parameter allows us to delete the drawing's image under the leader note's arrow line.

Draw Leader. This parameter controls drawing of the extension line for the leader note, from the snapping point to the leader note's arrow.

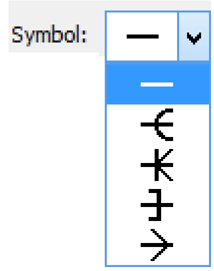
Clear Background under Leader. When this parameter is enabled, the drawing's image under the leader note's extension line is removed.

The **Arrow Text** section allows us to specify parameters of the text above and under the selected arrow:

Over Line. Text on the leader arrow.

Under Line. Text under the leader arrow.

The "**Symbol on arrow**" section contains parameters of the symbol on the selected arrow:

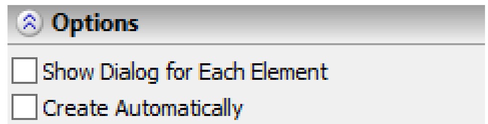


Symbol. Defines the type of the symbol (see the figure on the right), which will be put in the middle of the leader line. Normally it is used to indicate various drawnig notes.

Clear background. With this parameter enabled, the drawing image is erased behind the leader

Line Width. Defines the thickness of the arrow strokes. In the case when the parameter value is shown in square brackets, it is calculated based on the **Other lines** parameter defined in the command **ST: Set Document Parameters** (the tab **Lines**).

«Options» Section



The section contains following parameters

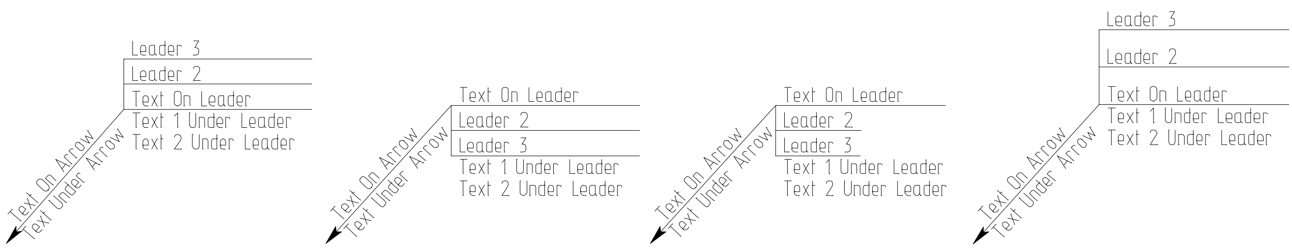
Show Dialog for Each Element. If this parameter is enabled, then the leader notes parameters dialog will automatically appear after defining the leader notes position in the leader notes creation command (the option ).

This mode allows working in the same way as in previous versions of T-FLEX CAD – by specifying the leader notes position in the drawing first, and then defining its parameters.

Create automatically. When this parameter is set, creation of the leader note will automatically be completed right after specifying the snapping point for the leader (without pressing ).

Leader Note Examples

The figures below show leader note appearances with different parameter settings: the first figure represents the state of the properties window as shown above; the second figure is obtained by disabling the “Append Upwards” parameter; the third figure illustrates the case with the disabled option “Align jogs length”; in the case of the fourth figure, the “Leader Height” parameter is set to the value 5.



Leader Note Parameters Dialog

Leader note parameters can also be defined in the parameters dialog called by the automenu option:

	<P>	Set Leader Note Parameters

The parameters on the tabs of this dialog duplicate the parameters in the properties window. Besides that, the parameters dialog contains several additional parameters. First of all, those are the system-wide parameters: level, layer, priority and color. Also, there is an additional tab in the parameters dialog that contains font settings. There, you can define the necessary font parameters that will be used to display the leader text.

> The leader note parameters dialog can also be called in the command-waiting mode from a leader's context menu (accessible by). This facilitates quick modification of a leader note's parameters without calling the editing command.

Defining Default Parameters

The default parameters that will be applied to all newly created leader notes can be defined in several ways.

First of all, those can be defined using the parameters dialog (the option). To do that, call the dialog before starting a leader note creation. The parameters defined in it will be copied into the parameters of each newly created leader note.

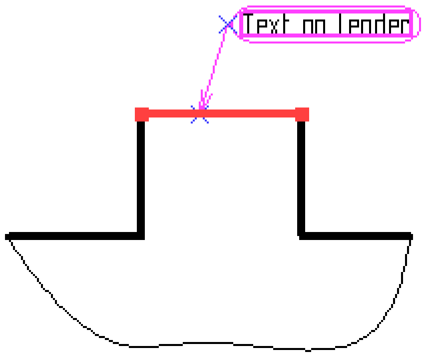
Besides that, you can save parameters of any leader note being created (or edited) as the default parameters, if you click the button in the command's properties window.

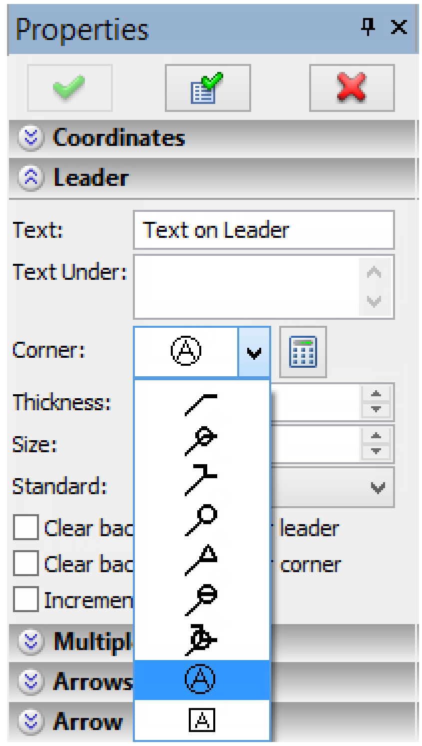
User's Defined Leader Notes

T-FLEX CAD allows a user to make his own types of notes, more precisely – types for the leaders of notes. User's defined leader notes are created as standard parametric fragments in the library in the folder *"System\Leader Symbols"* (library *"Leader Symbols"*).

In the dialog of the leader notes properties, the files located in the given folder are added as icons to the list of the accessible types of the leader. Upon creating the leader note, location of the fragment-leader image is determined by the fixing vector which must exist in the fragment model.

For controlling parameters of the user's leader note, the dialog of the fragment variables or the user's dialog (if it was created in the fragment) will be used. To call the dialog of the fragment variables (or the user's dialog), use the special button which appears in the dialog of the leader note properties or call the command **Annotation Properties…** in the context menu of the leader note that utilizes the user's leader type.





Editing Leader Notes

Editing leader notes is done by the command **El: Edit Leader Note:**

Keyboard	Textual Menu	Icon
<El>	Edit>Draw>Leader Note	

Upon calling the command, the following icons are provided in the automenu:

	<*>	Select All Elements
	<Esc>	Exit command

You can select one of the existing leader notes by pointing it with the mouse and clicking, or you can do multiple selections. As in the case of other drawing elements, multiple selections are done by the option. Using together with the depressed key <Shift> adds the element to the list of selected, while with the key <Ctrl> - removes from the list of selected.

Upon multiple selections, you can use the options:

	<P>	Set Leader Note parameters
	<Alt+P>	Copy Properties from Existing Element
	<Del>	Delete selected Element(s)

	<Esc>	Cancel selection

When selecting a single leader note, the selected element parameters are displayed in the properties window. The following icons become available in the automenu:

	<P>	Set Leader Note parameters
	<Alt+P>	Copy Properties from Existing Element
	<O>	Create Name for Selected Element
	<W>	Link To Product Structure
/ /	<U>	Fixing to Point/Angle Fixing/Orthogonal fixing
	<Space>	Add Arrow
	<Z>	Change leader line jog orientation
	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

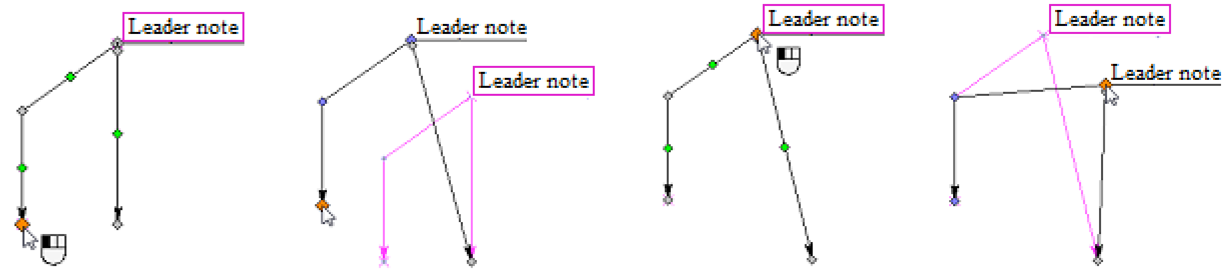
The general principles of the command **EI: Edit Leader Note** are similar to other editing commands. Keep in mind that a leader note has two or more (when there are bends or several arrows) attachment points. The positioning and attachment of each point is edited separately.

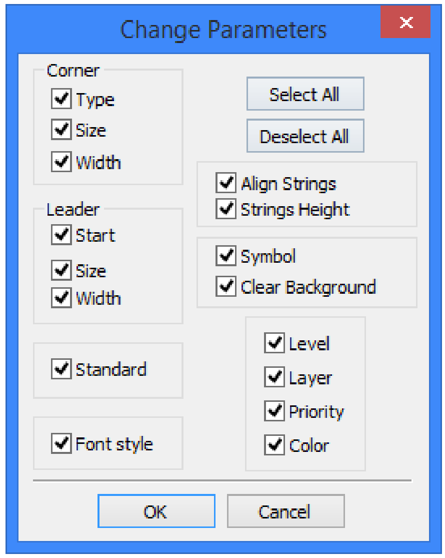
Select a leader note using . Now you can modify the position of either point of the leader note. To do this, move the pointer to the desired point and again click . At this moment, the following additional options will appear in the automenu:

	<T>	Link To Node (only when selecting a point snapped to the node with a shift)
	<K>	Break (kill) relations(only when selecting a point snapped to the node, line)
	<F>	Attach to arrow (available only when selecting the second attachment point of the leader note)
	<Del>	Delete Node
/ /	<U>	Fixing to Point/Angle Fixing/Orthogonal fixing
	<L>	Set relation with Line*
	<N>	Set relation with Node*

	<Z>	Change leader line jog orientation

Upon selecting a point, you start rubberbanding the leader note by the selected point.



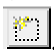


Now, you can fix the new position of the leader note attachment point. You can also break the attachment link using the option <K>, or attach the leader note to a node or construction entity.

Editing leader note parameters in the case when a single element is selected is similar to defining leader note parameters.

The option  allows modifying the parameters of several selected leader notes. The option <P> brings up the dialog box **Change parameters**. In this dialog box, check the parameters that you want to edit. By default, all the parameters of the selected elements are subject to editing. If a parameter is not supposed to be changed, clear the respective check box. Upon selecting the parameters for editing and pressing **[OK]**, you get the access to the standard leader note parameters dialog box. The parameters that were check marked are available for editing.

Option  takes parameters from another leader note.

The option  allows linking the leader note to a record (entry) in a BOM product structure window. Calling this option brings up the window **Product Structure**. To add the new jogs open **Multiple Jogs** tab of the properties dialog, select the desired item from the **Product Structure** and press <Ins> or button . After that, new position number will appear in **Multiple Jogs** and on the drawing.

Option *<Del>* allows deleting a leader note jog, when there are several, while options *<Alt+Up>*, *<Alt+Down>* - changing the order of the leader note jogs.

Callouts for Product Structure and Bend Notes

The **Leader Notes** command is used to create callouts for records of a product structure.

> More details about creation callouts in the product structure can be found in the chapter "Bill of Materials".

The **Leader Notes** command is used to create bend notes.

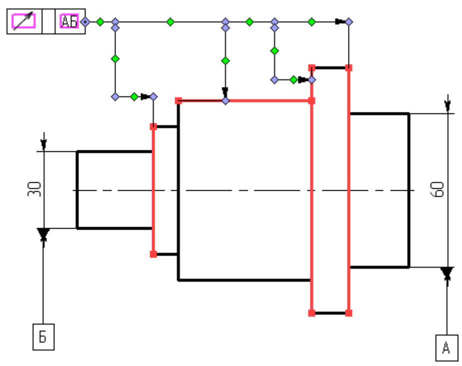
> More details about bend notes creation can be found in the chapter “Sheet metal operations”.

> The “By default” parameters for leader notes, callouts of the product structure and bend notes are stored in the system separately.

Geometric Dimensioning and Tolerancing Symbols. Datum symbols

To apply surface geometric dimensioning and tolerancing symbols (from now on – GD&T symbols) or datum symbols, use the command **FO: GD&T Symbols**.

A GD&T or datum symbol can be displayed with or without the leader line and contain any number of extension lines.



The size of the GD&T symbols is dependent on the font size specified among the parameters of a particular element or system-wide in the command **ST: Set Document Parameters**, the tab **Font**.

Creating Tolerancing and Datum Symbols

Enter the command **FO: GD&T Symbols**:

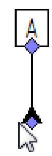
Icon	Ribbon
	Draw → GD&T Formlimits
Keyboard	Textual Menu
<FO>	Draw > GD&T Formlimits

You will be provided the following options:

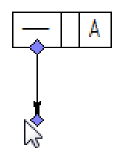
	<P>	Set GD&T formlimit parameters
	<T>	Copy properties from existing element
/	<O>	Create leader or datum with leader/Create leader or GD&T Formlimit with leader
	<M>	Select Linked dimension
	<N>	Select node

	<L>	Set relation with line
	<D>	Set relation with Dimension
/ /	<U>	Orthogonal Fixing/Fixing to point/Angle Fixing
	<Del>	Delete Arrow
	<F4>	Edit GD&T Formlimits
	<Esc>	Exit command

The / option allows us to select which type of elements will be created – tolerance or datum. The state of the option changes cyclically when we press on it in the automenu or by means of <O> on the keyboard. By doing so, the cursor view and the dialog of command's properties window will change.



When we start the **FO: Create tolerance of surface** command for the first time, the datum creation mode is automatically set up. The image of the datum with extension appears on the screen next to the cursor. The dialog in the properties window contains the parameters of the datum. In the automenu of the <O> option the icon is displayed that allows us to switch to the tolerance creation mode.



When switching to the tolerance creation mode, the image of the tolerance with extension line appears next to the cursor. In the command's properties window, the set of tolerance's parameters is shown instead of datum's parameters. The icon of the <O> option changes to .

It is possible to switch between work modes of the command (creation of tolerance/creation of datum) at any stage of command's work.

When we subsequently invoke the **FO: Create tolerance of surface** command (inside the same work session with T-FLEX CAD), the system will by default prompt us to create that type of the element (datum/tolerance) which was created the last in the preceding call of the command.

Creating tolerance

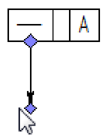
Creating tolerance with extension line

Specifying location of tolerance

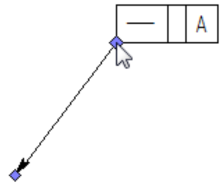
To create tolerance with extension, the user has to specify location of the tolerance on the drawing, shape of the extension, and also tolerance's parameters.

In general case the tolerance location is determined by sequential indication of two snapping points– snapping point of the extension's arrow and snapping point of the tolerance itself. Dynamic image of the tolerance on the screen shows which snapping point the user must specify at the current moment.

Location of both points can be specified in absolute coordinates and also via snapping to the elements of the drawing.



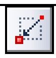
Dynamic view when the first snapping point is specified (snapping point of extension's arrow)



Dynamic view when the second snapping point is specified (snapping point of tolerance symbol)

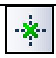
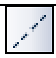
The first snapping point defines location of the extension's arrow. Snapping in absolute coordinates is carried out by pressing . Snapping point is created at the current location of the cursor.

Location of the second point can be defined relative to the first snapping point of the tolerance (with respect to the arrow) or in absolute coordinates. To select the mode, use the option:

	<F>	Attach to arrow

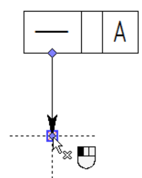
When this option is enabled, location of the leader is specified with respect to the arrow of the leader note, when this option is disabled – in absolute coordinates.

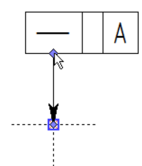
To snap the tolerance to drawing's elements it is possible to use object snapping and the following options of the automenu:

	<N>	Select Node
	<L>	Set relation with line
	<D>	Set relation with dimension

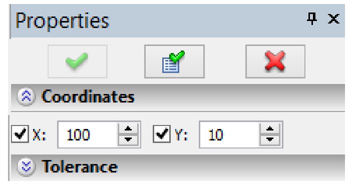
Object snapping can be used for snapping the tolerance symbol to such elements as construction line (line), graphic line (segment), 2D node, graphic lines that belong to 2D projections or 2D fragments, to points of confluence of graphic lines that belong to 2D projections or 2D fragments (a 2D node is created when selecting a point). It is also possible to read the value of the dimension for automatic calculation of the tolerance. With the approach of the cursor, the elements accessible for snapping will be highlighted. Both the extension line and the tolerance itself can be snapped to drawing's elements.

To snap to a 2D node or to a point of object snapping, it is sufficient to indicate the desired node/point and press . Location of the specified snapping point will be fixed.

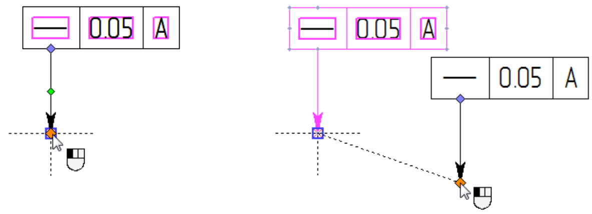




By default, the snapping of a tolerance is carried out to the node itself, without a shift.



To specify a shift, it is necessary after specifying location of all snapping points to return to editing of the desired point. After the point is selected it will start to follow the cursor. While doing so, the node will be highlighted and the rubber-band line will extend from it to the cursor. Pressing defines the tolerance shift with respect to the selected node. In the properties window it is possible to specify the exact values of the shifts along the X-axis and Y-axis.

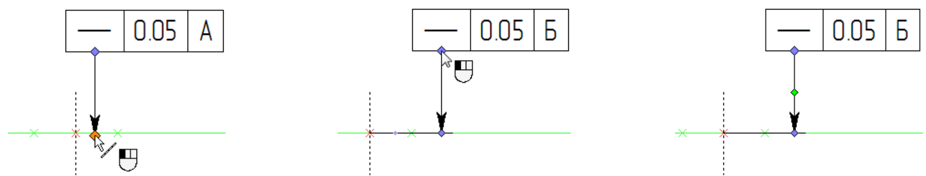


To return the snapping point to the selected node, i.e., to cancel the shift, use the option:

	<F>	Fix to a node

When snapping to the construction line or graphic line, it is necessary first to indicate the snapping line itself with the help of . The tolerance's image will start moving after the cursor along the selected line. Second click of will fix location of the extension's arrow or of the tolerance itself (in case of snapping the tolerance itself to the graphic line).

When snapping to the graphic line, the tolerance symbol can be situated beyond the graphic line bounds – on its continuation



In case of snapping the tolerance's arrow to the line, it is possible to control location of the arrow with respect to the snapping line. To do so, the following option of the command's automenu can be used:

/ /	<U>	Orthogonal Fixing/Fixing to point/Angle Fixing

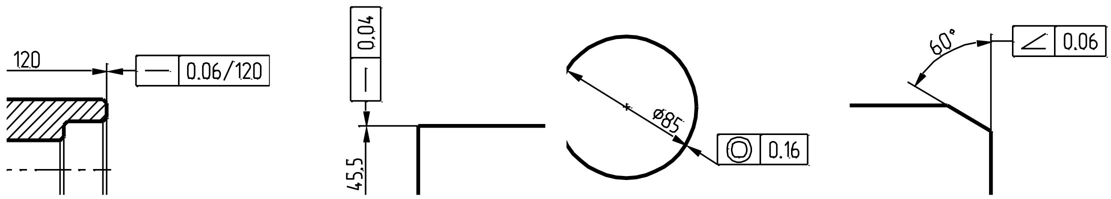
The option allows us to keep the extension line of the tolerance perpendicular to the selected snapping line. If the tolerance was created with the use of the option, then while being translated along the line it will be translated "as a whole" (without change in angles). The option allows us to retain the angle of the extension line with respect to the selected snapping line.

By default the "Tie perpendicularly" option is activated automatically. Note that in this case after selection of the snapping line it is sufficient to click only once to specify location of the arrow and of the tolerance itself. That is, the second click of immediately determines location of two snapping points of the tolerance.

To cancel snap to the line or node, use the option (available upon selection of the corresponding snapping point of the tolerance):

	<K>	Break (kill) relations

To snap tolerance to a dimension, it is necessary to specify the desired dimension, with the help of object snapping or with the option in the automenu. After selection of the dimension with the help of , the tolerance is tied to the dimension line. The tolerance's image will move after the cursor along the dimension line. Additional click of will fix the image of the tolerance in the desired location. The tolerance's parameters in this case are calculated automatically on the basis of parameters of the dimension. Figures below show examples of snapping tolerance with extension to the dimension:



There is also a possibility to link the tolerance's parameters with the dimension's value, without fixing location of the tolerance to the dimension itself. In the command's automenu the following option is available:

	<M>	Select linked dimension

The tolerance tied with the dimension with the help of the option can be positioned on the drawing in an arbitrary way, however, its parameters retain connection with parameters of the indicated dimension. When changing the dimension's value, the tolerance's parameters will change automatically.

To break connection between tolerance's parameters and the dimension's value, use the option:

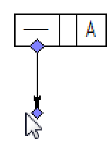
	<Ctrl+M>	Break link with dimension

To cancel the snap of the tolerance location to dimension, the option is used (available when selecting the corresponding snapping point of the tolerance).

Markers for controlling the snap of tolerance's elements. Creation of extension line with bends.

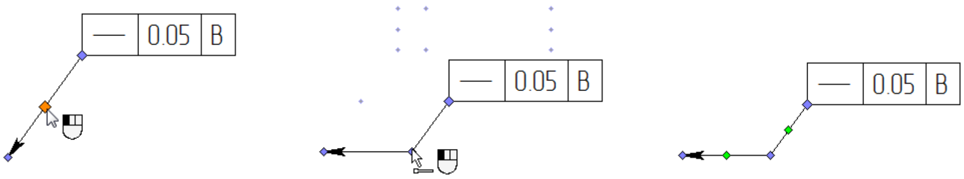
After initial snapping of the tolerance, location of its snapping points can be modified. Also it is possible to change the shape of the extension line by creating any number of bends on it. To control the snap of tolerance's points, and also for specifying bends on its extension line, the special markers are used.

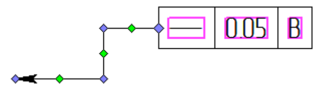
At the beginning of tolerance creation, there are only two markers – they designate its major snapping points and are highlighted with blue color. After location of both points has been specified, on the image of tolerance's arrow one more marker appears, highlighted with green color. This is a point in which it is possible to create a bend.



To correct location of any snapping point, it is sufficient to move the cursor closer to the marker of the desired point (it will be highlighted with orange color) and select it with . The point will start moving after the cursor. In this way, for example, it is possible to specify the shift when snapping the tolerance's point to the node.

To create a bend, it is required to move the cursor closer to the green marker (it will also be highlighted with orange color) and select it with the help of . The arrow will split into two sectors, flexure point – arrow bend point – will start moving after the cursor. The second click of will fix location of the arrow bend point.





After creation of the first bend point, new green markers will appear on the created arrow segments. If necessary, it is possible to create next bend on any of these segments. When arrow creation is finished, it is possible either to start creation of the next arrow or complete tolerance creation with the help of the option.

The following option controls the snap of each bend point with respect to the preceding point of the arrow:

	<F>	Attach to arrow

When this option is enabled, location of the bend is specified with respect to the previous characteristic point of the arrow, when this option is disabled – in absolute coordinates.

To remove the bend point, it is required to select it and use the following option:

	<Del>	Delete Node

Creation of additional extension lines (arrows) for tolerance

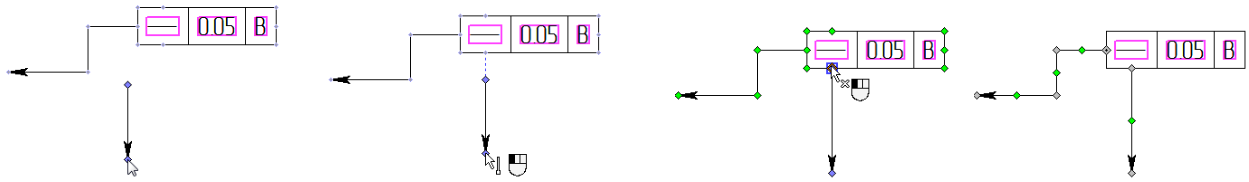
After location of the main arrow and the tolerance's image itself has been specified, additional option appears in the automenu:

	<Space>	Add arrow

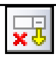
When this option is invoked, the image of the new arrow-extension appears next to the cursor.

Location of arrow of the additional extension line can also be specified in absolute coordinates or by using the snap to the lines and nodes of the drawing. With the help of markers it is possible to generate bends on the arrow. In the properties window the user can specify individual parameters of the arrow that are different from the specified parameters of the main arrow of the tolerance.

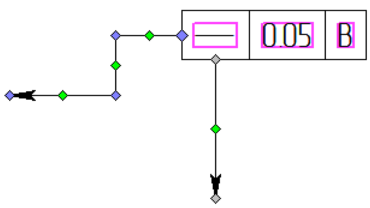
The end of the additional arrow can be snapped to any characteristic point of the tolerance's rectangle . (2D nodes can be created in those points if the option **Create Nodes on GD&T Symbol** is active, as defined in the command **Customize** > **Options...**, the tab **Snap**). The characteristic point nearest to the pointer is automatically selected when picking the GD&T symbol.

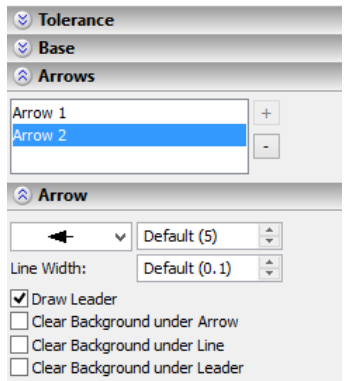


You can reject creation of the new arrow by using the following option:

	<Del>	Remove arrow

The list of created arrows for the tolerance is displayed in the properties window



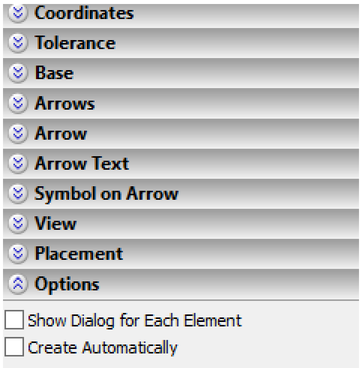


The [+] and [-] buttons to the right of the arrow list can also be used for creation and deletion of the tolerance's arrows.

Automatic completion of tolerance creation

By default, after specifying location of two main tolerance's points (location of the arrow and of the tolerance's rectangle) the command remains in the mode of editing the created element. This allows us to add to the tolerance the required number of additional arrows, create bends on the arrows, specify and edit parameters at any moment of tolerance creation. After creation of the tolerance is finalized, its creation needs to be completed explicitly with the help of the  option or analogous button in the command's properties window.

It is also possible to use another work algorithm. In the command's properties window, in the "Options" section, there is a flag **Create automatically**. When this flag is enabled, the tolerance creation will automatically be completed right after specifying the snapping point of the tolerance's rectangle.



Note that in this case it is required either to specify parameters of the element being created before the snap of the tolerance's rectangle, or to additionally enable the **Show parameters dialog for each element** flag. Then after specification of the tolerance's location the tolerance's parameters dialog will appear on the screen. This approach allows us to work the same way as in previous versions of T-FLEX CAD – first indicate location of the element on the drawing, then specify its parameters.

Creating tolerance without extension line

To create a tolerance without extension line, it is sufficient, at any moment of tolerance creation, to delete the extension line with the help of the option:

	<Del>	Delete arrow

The arrow will be removed, and the dynamic cursor will appear as a rectangle of the tolerance without extension line. It is more convenient to remove the arrow at the beginning of tolerance right away. If it is done at a later stage, for example, after snapping a tolerance, then it will be required to specify again location of the tolerance on the drawing.

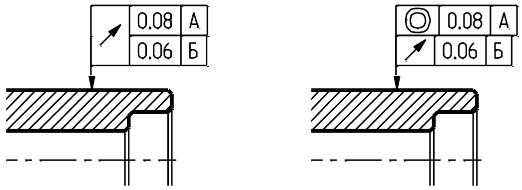
To remove arrows-extensions, it is also possible to use the command's properties window. To remove an extension line, it is sufficient to select the arrow from the list in the "Arrows" section (by default, it is assumed that there is only one arrow) and press the - button.

Location of the tolerance without extension line can be specified by the same rules as in general case. The only difference is that for positioning of the tolerance without extension line it is required to indicate location of only one snapping point – snapping point of the tolerance's rectangle itself.

Symbol for the tolerance without extension line can be snapped to another tolerance already existing on the drawing with the help of the option:

	<T>	Select tolerance's symbol

After this option is invoked it is required to indicate the tolerance's symbol. The newly created GD&T symbol will be positioned below the selected one. If the GD&T symbol type is the same as the other one, then the type notation fields of both GD&T symbols are merged. As the pointer approaches characteristic points on the GD&T symbol box, object snapping activates, allowing creating a 2D node at such a point and making an attachment to it. In this way, one can attach the new GD&T symbol to a characteristic point of another GD&T symbol . In this case, a 2D node will be created on the selected GD&T symbol. Respectively, you can select an attachment point on the GD&T symbol being created (see “GD&T Symbol parameters”).

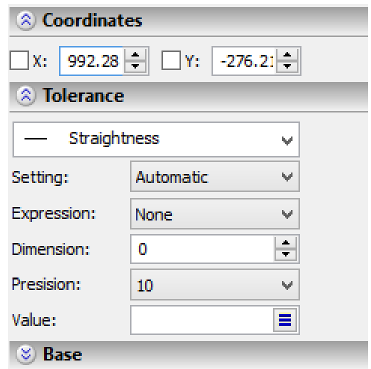


Note that completion of the creation of tolerance without extension line will occur automatically right after specifying the snapping point. Pressing  is not required in this case.

Tolerance’s parameters

The dialog in the command’s properties window contains all main parameters of the tolerance. To make the work with this dialog more convenient the dialog is divided into several sections..

«Coordinates» Section



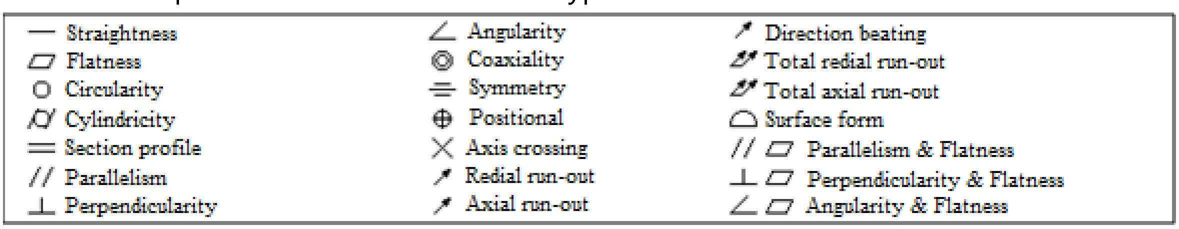
The first section **Coordinates** contains the fields for accurate specification of the coordinates of snapping points of the tolerance. The current coordinates are dynamically determined when the cursor moves inside the drawing’s window.

Depending on the tolerance snapping method various types of coordinates can be displayed in this section. For example, if snapping of both points is free, the absolute coordinates of both snapping points are shown in the properties window. When snapping the rectangle of tolerance/datum to the extension’s arrow, for the second point the shifts dx and dy with respect to the first point will be specified, etc.

«Tolerance» Section»

The **Tolerance** section contains the main parameters of the tolerance:

From the drop down list select the desired type of the tolerance:



Specification. This parameter determines how the value of the tolerance will be obtained. You can enter it manually or it can automatically be calculated depending on the dimension’s value and precision.

Expression. Can be one of the following choices:

	- not defined;
R	- if a circular or cylindrical GD&T symbol is defined by the radius;

D	- if defined by the diameter;
Sphere R	- if a spherical GD&T symbol is defined by the radius;
Sphere D	- if a spherical GD&T symbol is defined by the diameter;
T	- if the diametrical expression is used for GD&T symbol of Symmetry, Axis crossing, Profile form and Surface form, as well as the positional GD&T symbol (in the case when the positional GD&T symbol is bounded by two parallel lines or planes);
T/2	- if the radial expression is used for the same GD&T symbols as above.

Dimension. Value of the dimension upon which the value of the tolerance depends. If, when setting the tolerance, you tied it with the dimension, then the nominal value of the dimension will automatically be inserted into the parameter's field. The value of the dimension affects the tolerance's value in automatic calculation.

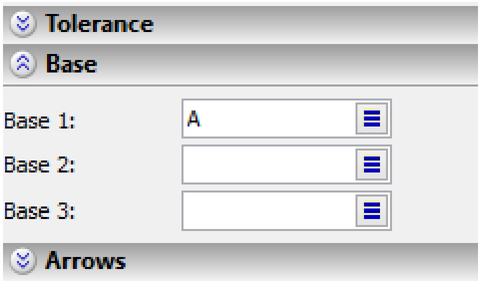
Precision. Takes integer values in the range from 3 to 16, inclusive. This parameter makes sense only when using the automatic tolerance calculation.

Value. This is the value of the tolerance, either input manually or calculated automatically. A predefined list of values is supplied for the manual parameter definition.

«Datum» Section

Datum 1, Datum 2 and Datum 3. These are the names of datums that can be used in the GD&T symbol, defined as textual strings.

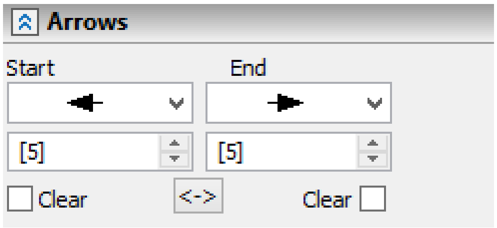
Text variables can be used as datum names just like in the case of any other text string parameters.



«Arrows», «Arrow», «Text on arrow», «Symbol on Arrow» Sections

Next two sections of the properties window – "Arrows" and "Arrow" – should be considered together. Together they allow us to specify parameters of the tolerance's extension lines.

The **Arrows** section contains the list of all created arrows-extensions of the tolerance. The buttons to the right of the list allow us to create/remove the arrows. By selecting one of the arrows in the list with the help of , in the next sections of the properties window it is possible to specify parameters of the given arrow.



The **Arrow** section contains the following parameters of the selected arrow:

Drop down list for selection of the arrow type at the beginning of the extension line. This list coincides with the list of arrow ends used in the commands for construction of dimensions, graphic lines and leader notes.

Field for specifying the arrow tip size at the beginning of the extension line. If the value is displayed in square brackets, it will be calculated based on the given parameter **Arrow (end) size** in the command **ST: Set document parameters** (the **Linesв** tab).

Line thickness. Specifies thickness of arrow lines. If the value is displayed in square brackets, it will be evaluated based on the given parameter **Other lines** in the command **ST: Set Document Parameters** (the **Lines** tab).

Clear under arrow. When this parameter is enabled, the drawing's image under the tolerance's arrow is removed.

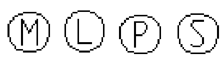
Clear under line. This parameter allows us to remove the drawing's image under the tolerance's arrow line.

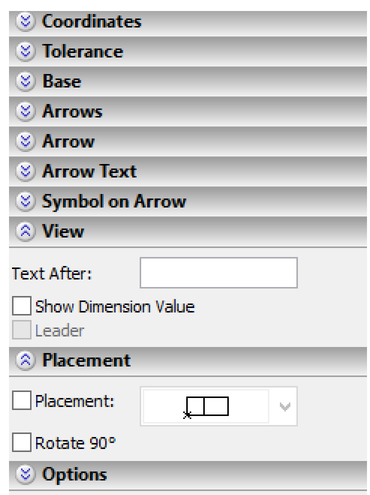
Draw extension line. This parameter controls drawing of the extension line for extension of the tolerance, from the snapping point up to the arrow.

Clear under extension line. When this parameter is enabled, the drawing's image under the extension line is removed.

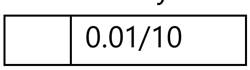
«View» section

Text after. This parameter allows entering an arbitrary text string that you want to be displayed after the tolerance value. You can enter, for instance, conditions for dependencies, unevenness, etc. Use the key combination <Alt+F9> for this purpose.





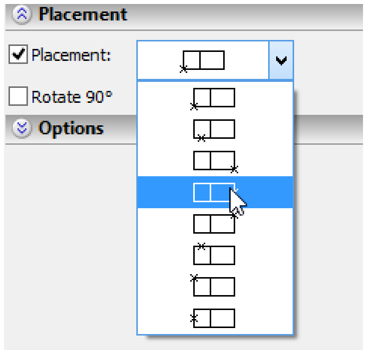
Show dimension's value. It can take the values "No" and "Yes". If this parameter has the value "Yes", then the dimension's value will be placed after the tolerance's value. If necessary, in this line use the variables by embracing them into curly brackets.



Extension line. This parameter is active only upon creation of the tolerance without extension line. It controls the drawing of the extension line from the snapping node/line up to the tolerance.

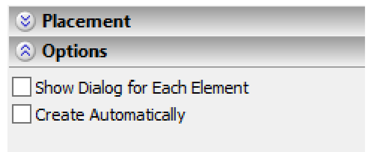
«Placement» section

- **Placement.** Defines the box positioning with respect to the attachment node. Eight different positioning options are provided in the drop-down menu.
- **Rotate 90°.** Can be set or unset. If set, the GD&T symbol will be rotated by 90 degrees.



«Options» section

This section contains the following tolerance's parameters:



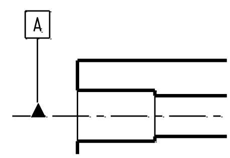
Show parameters dialog for each element. If this parameter is enabled, then in the tolerance creation command after specifying location of the tolerance on the drawing, the parameters dialog window will automatically appear on the screen (the  option).

This mode allows us to work in the same way as in the previous versions of T-FLEX CAD – first specify the tolerance location on the drawing, and then specify the tolerance's parameter.

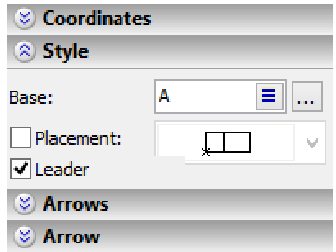
Create automatically. When this parameter is enabled, the tolerance creation will automatically be completed right after specification of the snapping point of the tolerance's rectangle (without pressing ). This parameter is active only if the tolerance with extension line is created.

Creating Datum

To create a leader or a datum with a leader, use the automenu icon .The creation and editing techniques of these elements are similar to creating and editing of GD&T symbol with leaders. The datum leader has a different ending tip (see the diagram on the right hand side).



Similar to the tolerance, at any moment of datum creation it is possible to specify the datum's parameters in the command's properties window. The dialog in the command's properties window contains all main parameters of the datum. To make the work more convenient, the dialog is divided into several sections. Some of the sections – **Coordinates**, **Arrows**, **Arrow**, **Options** entirely repeat analogous sections of the tolerance's parameters dialog.



The **Datum** section, used for datum creation, contains different, shorter set of parameters compared to the identically-named section in the tolerance's parameters:

Datum. Datum name parameter (text string). For the datum's name it is possible to use the variable.

Location. Defines location of the frame with respect to the snapping node. There are eight different options which are selected from the drop down list.

Extension line. This parameter is active only upon creation of the datum without extension line. It controls the drawing of the extension line from the snapping node/line up to the datum.

Editing GD&T/Datum's Symbols

To edit GD&T/ datum symbol parameters, position, attachment or to set or break the relation between a GD&T/ datum symbol and a drawing dimension, use the command "**EFO: Edit GD&T Symbols**":

Keyboard	Textual Menu	Icon
<EFO>	"Edit > Draw > GD&T Formlimits"	

Upon calling the command, the following icons are available in the automenu:

	<*>	Select All Elements
	<Esc>	Exit command

You can select a GD&T/ datum symbol by pointing it with the mouse and clicking, or by using multiple selections. As in the case of other drawing elements, the multiple selections are done by the option . Using together with the pressed key <Shift> adds an element to the list of selected ones, while with the key <Ctrl> - removes from the list of selected elements.

For multiple selections, you can use the options:

	<P>	Set GD&T Formlimit Parameters
	<Alt+P>	Copy Properties from Existing Element
	<Del>	Remove selected Element(s)
	<Esc>	Cancel selection

When selecting one element, the following icons become available in the automenu:

	<P>	Set GD&T Formlimit Parameters
	<Alt+P>	Copy properties from the Existing Element
	<M>	Select Linked dimension

	<Ctrl+M>	Break tie with dimension (only upon selection of the element tied with the dimension)
	<Space>	Add arrow
	<Del>	Delete Arrow
	<I>	Select Other Element
	<Del>	Remove selected Element(s)
	<Esc>	Cancel selection

To change location and snapping of the tolerance/datum, it is necessary, after element selection, indicate the marker of the desired snapping point with the help of . The selected point will start dynamically follow the cursor (according to the snapping method selected for the point). At the same time the additional options will appear in the automenu:

	<T>	Link to node
	<K>	Break (kill) relations
	<F>	Snap to arrow (available only upon selection of the second snapping point of the note)
/ /	<U>	Orthogonal Fixing/Fixing to point/Angle Fixing
	<L>	Set relation with Line
	<N>	Select Node
	<D>	Set relation with Dimension

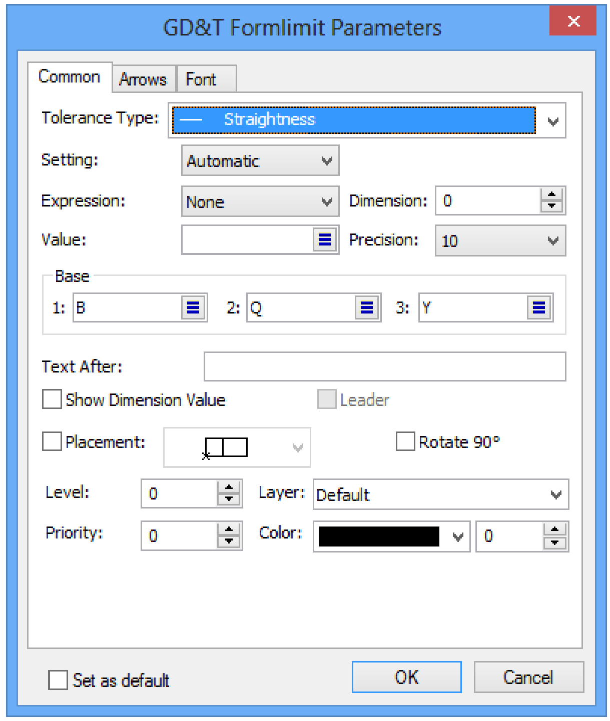
Pressing will fix the new location of the snapping point of the tolerance. It is possible to change location of any bend point exactly in the same way. It is also possible to cancel snapping of the point with the help of the option, snap tolerance to a node or line, specify/cancel relation with dimension.

To remove one of the arrow-extensions of the tolerance, it is possible to use the option. If the tolerance/datum being edited has only one extension, then this option is available in the automenu right after selection of the element being edited. In case if the tolerance/datum with several extension lines, the option will become available after selection of the desired extension line. The extension line can be selected in the command's properties window (the "Arrows" section) or, for example, by specifying its starting point in the drawing's window. To remove the extension line, it is also possible to use the button next to the list of arrow-extensions of the tolerance/datum in the command's properties window.

The option  allows modifying the parameters of the selected GD&T symbol. Modifying the parameters in the case of a single selected element is similar to original defining of the GD&T symbol parameters.

In the case of multiple selections, calling the option <P> brings up the dialog box "GD&T Formlimit Parameters"., All parameters of the selected elements are subject to editing.

The option <D> allows setting the relation between the selected GD&T symbol and the desired drawing dimension. To delete the selected GD&T symbol, press the key <Del>. To select and modify parameters of a group of GD&T symbols, use the same techniques as for other elements.



Roughness Symbols

Roughness notation symbols creation is similar in its nature to creating leader notes and tolerances. First, you define the position and attachment of the roughness, and then specify its parameters. The size of a roughness element is related to the font size defined either in the parameters of the specific element, or in the command **ST: Set Document Parameters**, the tab **Font.**

Creating Roughness Notation

To apply a roughness notation, you need to enter the command **RO: Create Roughness Symbol**:

Icon	Ribbon
	Draw → Title Block → Roughness Symbol
Keyboard	Textual Menu
<RO>	Draw > Roughness Symbol

The following options will become available to you in the command:

	<Enter>	Place a roughness symbol at the pointer position.
	<Alt+P>	Copy Properties from Existing Element
	<P>	Set Roughness Symbol Parameters
	<N>	Set relation with Node
	<L>	Set relation with Line
	<C>	Set relation with Circle
	<D>	Set relation with Dimension
	<R>	Set Relation with Leader Note
	<E>	Set relation with Ellipse
	<S>	Set relation with Spline
	<T>	Link to Node
	<Space>	Change Roughness Attach type
	<Z>	Change leader line jog orientation (available only with selection of the previous option)
	<K>	Break (kill) relations (available upon selecting attachment element)

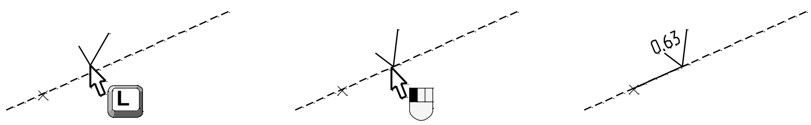
	<F4>	Execute Edit Roughness Symbol command
	<Esc>	Exit command

The roughness can be instantly placed in the absolute coordinates at the pointer position by clicking . An exact value of coordinates can be defined in the command's properties window (the section "Coordinates").

The way of attaching the created roughness is determined by the status of the option . This option contains a drop down list with the following choices:

	<Alt+N>	Roughness without leader jog
	<Alt+L>	Roughness with leader jog
	<Alt+T>	Attach to the sign point

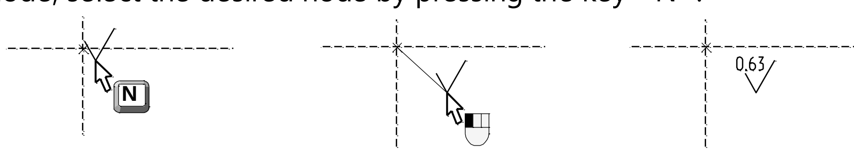
To attach a roughness symbol to a construction or graphic line (ellipse, spline, path or function), use the respective option <L> (<E>, <S>). The graphic pointer must be over the desired line when using the option. The intended construction entity must have at least one node on it. In this way, the roughness being created is attached to the entity and the nearest node on this construction entity. A leader line will be created by default from the node to the roughness symbol (you can cancel the leader line creation in the command's properties window).



With the object snapping engaged, to select an attachment element you just need to move the pointer over the desired element. As the element pre-highlights and the pointer changes its shape, indicating snapping to the element, click . In complicated configurations, you can use the element selection options for precise element selection.

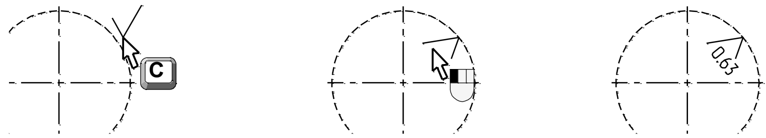
Upon selecting the construction or graphic entity (ellipse, spline, path or function), a roughness symbol starts rubberbanding with the pointer. To complete the creation, point the mouse to the desired position of the roughness symbol (the distance from the node to which the roughness being created is attached) and click . A precise position of the roughness on a construction or graphic line can be defined in the command's properties window.

For attaching to a node, select the desired node by pressing the key <N>.

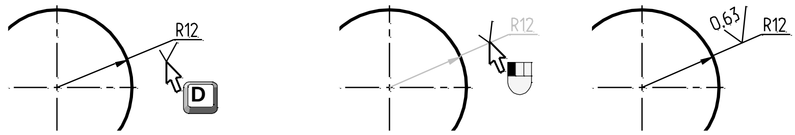


When selecting a node, the roughness can be created in two modes: with an offset from the node and without an offset. By default, the mode of snapping to the node without an offset is used. This is indicated by the enabled option in the command automenu. In this case, you just need to point at a node and click. To set the mode of snapping to a node with an offset, disable the option. In this case, after selecting the snap node you will need to specify an offset relative to the node for the roughness. This can be done by in the drawing window or by entering the exact offset value in the command's properties window.

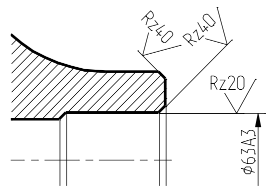
A roughness symbol can be attached to a circle (the option <C>).



The dimension is selected by the key <D>.



A roughness can also be attached to graphic lines. As the pointer approaches a graphic line, the entity is pre-highlighted due to the object snapping. If the roughness symbol is placed beyond the graphic line limits, then by default a leader is created along the graphic line extension up to the roughness symbol (the leader creation can be disabled in the roughness parameters).



To undo element selection (line, node, circle or dimension), and, thus, to cancel the attachment relation, use the option.

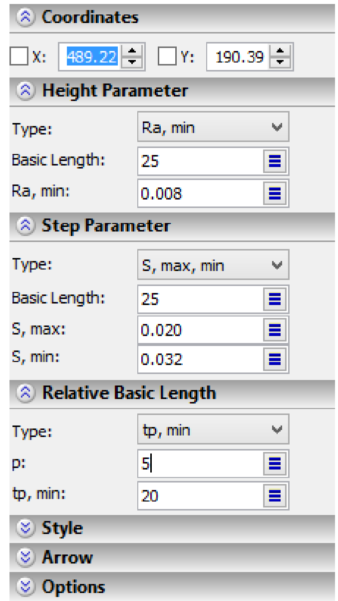
To cancel the last action (for example, to cancel the leader start attachment), use the <Esc> key or right click.

Roughness parameters are defined in the command's properties window prior to specifying the roughness position on the drawing. Besides that, you can enter the exact position of the roughness snap point in the properties window. Option will take parameters of roughness symbol from already existing roughness element. See chapter “Dimensions” for more details on this option.

Roughness Parameters

The first section – “**Coordinates**” – provides the fields to enter the exact coordinates of the roughness snap point. The current coordinates are dynamically tracked as the cursor moves in the drawing window.

The other sections of the properties window – **Height Parameter**, **Step Parameter**, **Relative Basic Length** – may contain various sets of parameters, depending on the **Type** field value selections in each section:



Height Parameter. The possible parameter combinations are – **Basic Length** and:

Ra,	Ra, max, min	Ra, min	Ra, nom
Rz,	Rz, max, min	Rz, min	Rz, nom
Rmax,	Rmax, max, min	Rmax, min	Rmax, nom

Step Parameter. The possible parameter combinations are – **Basic Length** and:

S,	S, max, min	S, min	S, nom
Sm,	Sm, max, min	Sm, min	Sm, nom

Relative Basic Length. The possible parameter combinations are:

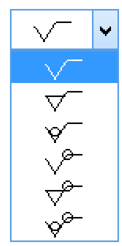
tp,	tp, max, min	tp, min	tp, nom

Note that the set of values that you can select can be either in the metric or in the inch notation. This is controlled by the parameter **Units** in the command **ST: Set document parameters**.

Remember that you can use variables (surrounded in braces) in any parameter field.

Besides, each parameter combo box provides the pre-defined list of values. The user can customize this list, modifying and appending it as desired. To edit the list, right click over the dialog combo box, and select the command **Edit Value List** in the context menu. For detailed information, refer to the chapter “Main Concepts of System Operation”, the topic “Context menu for dialog input boxes”.

In the “**Style**” section there are the following roughness parameters:

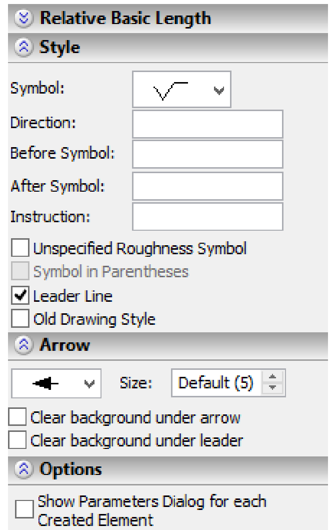


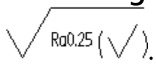
Symbol. The type of the roughness symbol notation can be selected from the set (see the diagram on the right).

Direction. Is defined by a symbol of icon or special font, invoked by the key combination <Alt><F9>.

Before Symbol, **After Symbol**. These parameters allow defining additional strings of text to be displayed before and after the roughness symbol, respectively.

Instruction. This parameter defines the string that will be put above the jog.



Unset Roughness Symbol. Creates the notation of unset roughness . To define the symbol inside the parentheses, set the flag **Symbol in parentheses**.

Leader Line. This parameter sets the mode of creating the leader line when attaching the roughness to a line or a graphic entity.

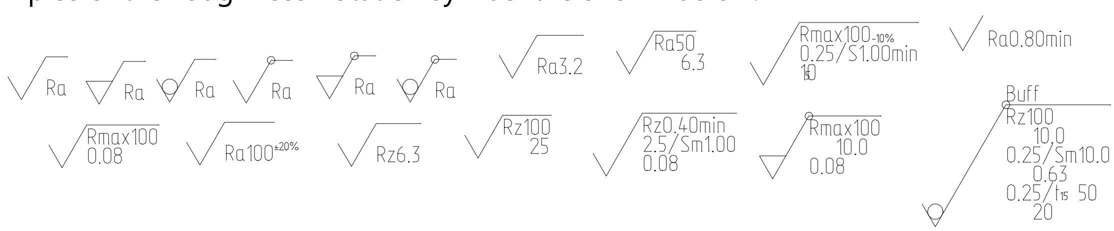
Old Drawing Style. This flag is reserved for switching between old and new roughness notation standards when applicable.

In the **Arrow** section you can define the type and size of the leader arrow. The parameters represented by textual strings allow use of variables (the variables must be surrounded in braces).

The section "Option" contains only one auxiliary parameter– **Show Dialog for each Created Element**. If this parameter is enabled, then the roughness parameters dialog will automatically appear after defining the roughness position in the roughness creation command (the option).

This mode allows working in the same way as in previous versions of T-FLEX CAD – by specifying the roughness position in the drawing first, and then defining its parameters.

Various samples of the roughness notation symbol are shown below:



Roughness Parameters Dialog

You can also define roughness parameters using the parameters dialog accessible by the automenu option:

	<P>	Set Roughness Symbol Parameters

The parameters available on the tabs of that dialog duplicate the parameters in the properties window. Besides that, the parameters dialog contains several additional parameters. First of all, those are the system-wide parameters: level, layer, priority, color. There is also an additional tab in the dialog that contains font parameters. There you can define the required font parameters to display the roughness text.

Defining Default Parameters

The default parameters that will be applied to all newly created roughness symbols can be defined in various ways.

First of all, those can be defined using the parameters dialog (the option). To do that, call this dialog before creating a roughness. The parameters defined in it will be copied over to the parameters of each newly created roughness symbol.

Besides that, you can save parameters defined for any roughness being created (or edited) as the default, by clicking the button in the command's properties window.

Editing Roughness Symbol

The command **ERO: Edit Roughness Symbol** allows changing the attachment, position and the parameter values of a roughness symbol (alternatively, use the option <F4> in the command **RO: Create Roughness Symbol**):

Keyboard	Textual Menu	Icon
<ER>	"Edit > Draw > Roughness Symbol"	

Upon calling the command, the following icons are available in the automenu:

	<*>	Select All Elements
	<Esc>	Exit command

A roughness symbol notation can be selected by pointing at with the mouse and clicking , or by multiple selections. As in the case of other drawing elements, multiple selections are done by the option . Using together with the depressed key <Shift> adds an element to the list of selected, while with the key <Ctrl> - excludes from the selected list.

For multiple selections, you can use the options:

	<P>	Set Roughness Symbol Parameters
	<Alt+P>	Copy Properties from Existing Element
	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

When selecting a single element, the properties window displays parameters of the selected element. The following icons become available in the automenu:

	<P>	Set Roughness Symbol Parameters
	<Alt+P>	Copy Properties from Existing Element
	<K>	Break (kill) relations (available when the selected roughness is attached to a node, construction or graphic entity)

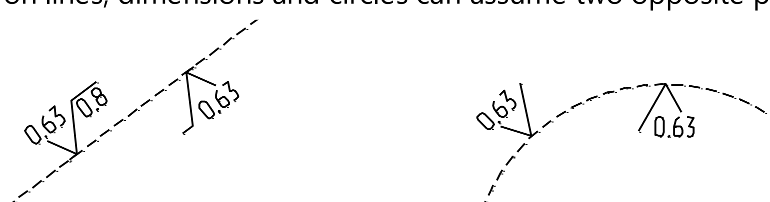
	<H>	Change leader/roughness position (available when selecting a roughness symbol with a leader)
	<Z>	Change leader line jog orientation (available only when selecting a roughness symbol with a leader)
	<T>	Link to Node
	<Space>	Change Roughness Attach type
	<N>	Set relation with Node*
	<L>	Set relation with Line*
	<C>	Set relation with Circle*
	<D>	Set relation with Dimension
	<R>	Set Relation with Leader Note
	<E>	Set relation with Ellipse*
	<S>	Set relation with Spline*
	<I>	Select Other Element
	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

* The respective attachment element selection option is available if the selected roughness symbol was defined in the absolute coordinates, or if the attachment of this element was canceled by the option .

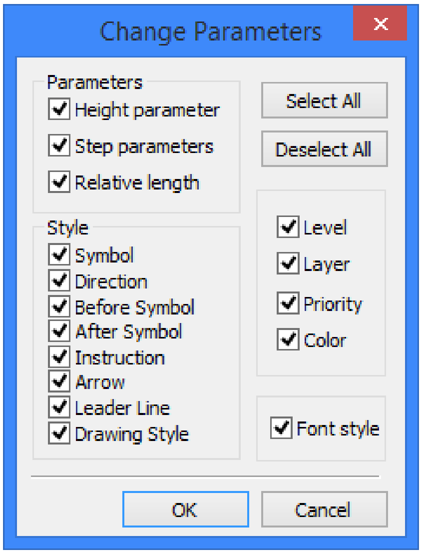
Once selected, the roughness symbol starts rubberbanding on the screen, following the pointer. The option allows selecting which point of the roughness notation to rubberband - the arrow tip or the leader jog. Clicking fixes the roughness symbol in the new position.

To change the attachment type, first you need to cancel the original attachment by using the option (<K>). After that, the options will be provided in the automenu for selecting new attachment elements: <N>, <L>, <C>, <S>, <D>. If the position of the selected roughness symbol was defined in the absolute coordinates, then you do not need to use the first option <K>.

Node, that roughnesses on lines, dimensions and circles can assume two opposite positions.



The option <P> allows modifying parameters of the several selected roughness symbols. Modifying parameters in the case of selecting a single element is similar to defining the roughness parameters. However, if you selected multiple roughnesses for editing and run **Edit** command, then you need to determine first, which parameters to modify, in the “Change parameters” dialog box. By default, all parameters of the selected elements are subject to editing. Upon selecting parameters for editing and pressing **[OK]**, you will access the standard dialog box for defining roughness parameters.



Option  takes parameters of roughness symbol from existing roughness symbol.

To delete a roughness symbol, select it, and then press the <Del> key.

Section View

The section view, arrow view and local area view are necessary detailing elements of a drawing. T-FLEX CAD system provides full range of functionalities for satisfying this requirement.

Creating Section View

The command for section view creation, **SE: Create Section** can be called as follows:

Icon	Ribbon
	Draw → Title Block → Section
Keyboard	Textual Menu
<SE>	Draw > Section

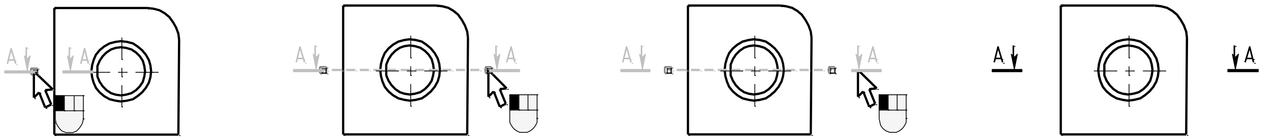
Upon calling the command, you will get access to the following set of options:

	<P>	Set Section Lines Parameters
	<Alt+P>	Copy Properties from Existing Element
	<S>	Create two point Section
	<D>	Create multiple point section
	<R>	Create Arrow View
	<V>	Create View
	<N>	Select Attachment Node
	<Z>	Change View Direction
	<F4>	Execute Edit Command
	<Esc>	Exit command

Next, select the type of the section view to create a simple (two-point) section, a multiple-point section, an arrow view or a view notation.

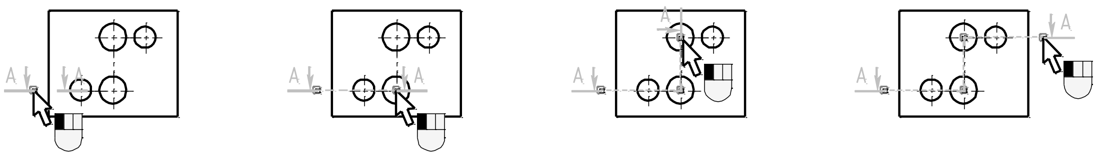
Two-Point Section

Two-point section creation begins with selecting two attachment points. Those can be defined either in the absolute coordinates or snapped to 2D nodes. Move the pointer to the desired position and click . As the pointer approaches 2D nodes, the object snapping activates, highlighting the nodes. Assign the second attachment point similarly. The preview of the element being created updates dynamically with your manipulations. Next (refer to the third diagram from the left), you need to define the offset of the view notation arrows from the attachment points. Do this by positioning the pointer appropriately. To fix the position, click . The result is shown on the right-most diagram.

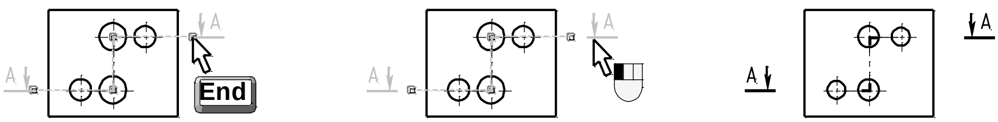


Multiple-Point Section

The multiple-point section creation is similar to creating a two-point section. The difference is in the number of the attachment points to select, which is unlimited in the case of a multiple-point section. To call the command for creating a multiple-point section, press the automenu icon or type the key <D>. The sequence of actions for creating a multiple-point section is shown on the following diagrams.



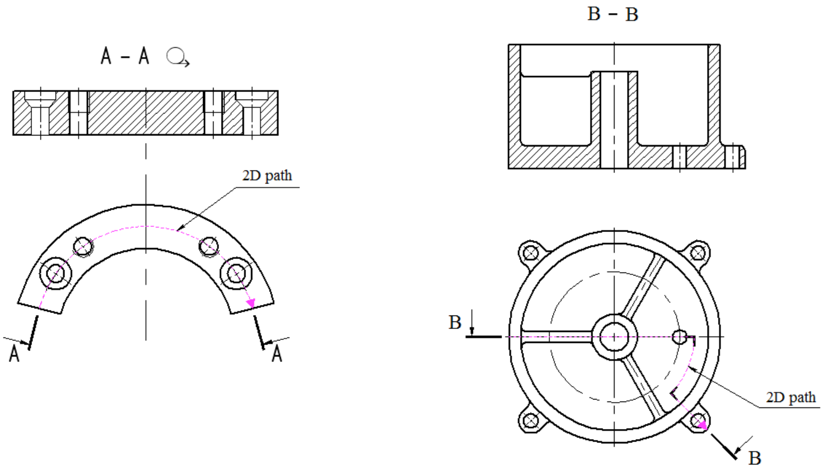
Upon entering the desired number of points, you need to press the automenu icon or the key <End>.



Instead of specifying the individual points of the section, it is possible to select an existing 2D path with the help of the option:

	<T>	Select 2D path

The created section will repeat the form of the selected path.



The view direction can be flipped to the opposite at any time by pressing the automenu icon  or the key <Z>.

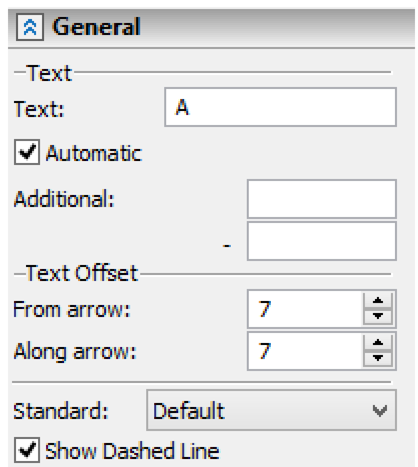
After defining all fixing points of section, you must specify the offset of arrow regarding fixing points with . The offset definition may be omitted by pressing  in automenu or in the properties window.

Section Properties

Section parameters can be set at any time prior to the completion of element creation in the properties window of the command.

The first sub-dialog **Coordinates** contains fields for entering precise coordinates of section fixing points. Current coordinates are dynamically traced on moving the cursor in the drawing window.

"General" sub-dialog allows you to set all the basic parameters of section. These settings include:



The group **Text**:

Text. The text to be displayed next to the arrows. This entry is filled automatically with a subsequent letter of the alphabet, beginning with the letter “A”. If the number of notations exceeds the number of letters in the alphabet, the multiple-letter combinations are used: AA, AB, AC, ..., AAAAAAA, etc.

If necessary, you can manually type a text string of an arbitrary length in this input box.

Additional. The input boxes of this parameter allow entering different text for each arrow. The specified text strings will be displayed next to the text defined by the previous parameter.

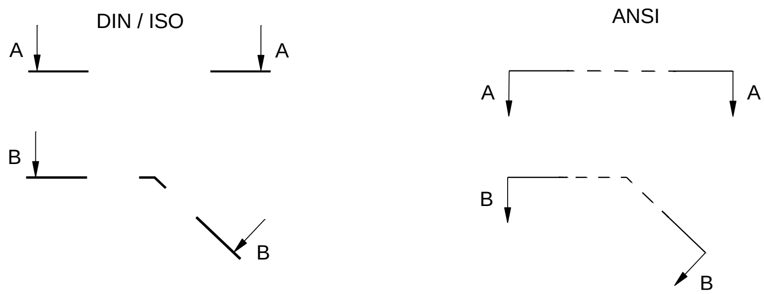
The group **Text Offset**:

From arrow. Defines the text offset from the arrow in the outward direction along the leader, in the model measurement units.

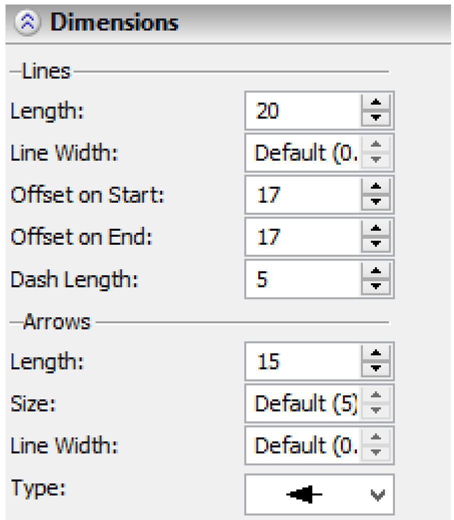
Along arrow. The text offset from the leader.

Standard. Defines the view notation standard. You can choose from the three options: the **ISO** standard, the **ANSI** standard and **Default**.

In the case of using the last option, the standard is defined by the **Standard/Dimension** parameter setting in the command **ST: Set Document Parameters**, the tab **Dimensions**. The ANSI standard permits displaying the dashed line (the parameter **Show Dashed Line**).



Dimensions



The group **Lines**:

Length. Defines the length of the leader.

Line Width. Defines the thickness of the leader. In the case of using the **Default** setting, this parameter value is calculated based on the **Line thickness/Thick lines** parameter setting in the command **ST: Set Document Parameters** (the tab **Lines**).

Offset on Start. The offset of the leader from the first node of the view notation.

Offset on End. The offset of the leader from the last node of the view notation.

Dash Length. Sets the length of the medium dashed strokes (displayable only in ANSI standard).

The group **Arrows**:

Length. Sets the arrow length of the view direction notation.

Size. Sets the size of the view direction arrow. In the case of using the **Default** setting, this parameter value is calculated based on the **Arrow (end) size** parameter setting in the command **ST: Set Document Parameters** (the tab "Lines").

Line Width. Sets the line thickness of the view direction arrow. In the case of using the **Default** setting, this parameter value is calculated based on the **Thick lines** parameter setting in the command **ST: Set Document Parameters** (the tab **Lines**).

Type. Sets the type of the view direction arrow.

The parameters on this tab are defined in the measurement units set in the drawing parameters (the command **ST: Set Document Parameters**).

Section parameters can be also set in the section Parameters dialog box, opened by option . In this dialog, in addition to the above section properties, you can specify system-wide settings: Color, Level, Priority, Layer and [Font]. They are entered the same way as in other elements.

> You should note that the size of the view designation text is associated with the font size that is specified in the parameters of a particular element, or in command **ST: Set Document Parameters** on the **Font** tab.

Copying Parameters from Existing Notations of Section

The values of parameters of the notation of the section being created can be quickly copied from an already existing notation of the section. To do so, it is necessary to use the option:

	<Alt+P>	Copy Properties from Existing Element

> This option is available in the automenu of the command before creation of the view notation or during the process of creation.

After this option is invoked, it is sufficient to indicate the notation of the section whose parameters' values must be transferred to the element being created.

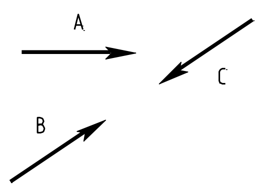
To assign the copied values of the parameters to all new notations of the section, it is required to activate the additional option before selection of the original element:

	<S>	Set Properties as Default

When the option is activated, the copied parameters will be saved as default parameters.

> This option simplifies creation of the notations of a section with identical parameters. However, it does not allow us to copy individual parameters or parameters from an object of another type. In such cases it is more convenient to use the general mechanism of editing parameters of elements in the properties window.

Arrow View



An arrow view can be attached to a node, defined by assigning the view direction vector by two nodes, or positioned in the absolute coordinates without attachment to any drawing objects. To create an arrow view, press the automenu icon or the key <R>. Upon calling the command, the arrow view notation starts rubberbanding on the screen, and additional options appear in the automenu:

	<B>	Set First Attachment Point
	<E>	Set Second Attachment Point
	<Z>	Change View Direction

	<H>	Change Text Placement

For attachment in the absolute coordinates, you can simply click . The view notation will then fix at the current pointer position.

For attachment to a 2D node, you need to use the option <N> (the automenu icon) or rely on the object snapping. As the pointer approaches a 2D node or a line intersection, the respective entities are highlighted. You can then click .

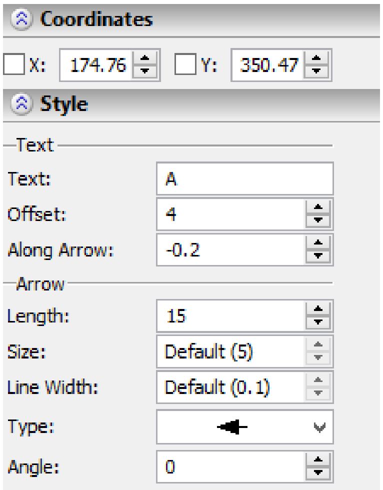
By default, the arrow is positioned horizontally and directed from left to right. The arrow direction can be quickly rotated by the angle multiple of 90° by pressing the automenu icon or the key <Z>.

For attachment to two nodes, subsequently use the automenu options:

	<B>	Set First Attachment Point
	<E>	Set Second Attachment Point

and consequently select two 2D nodes.

You can change the text position with respect to the arrow line at any time by using the option or the key <H>.



Coordinates. The input boxes for specifying the exact X and Y coordinates when positioning the view notation in the absolute coordinates.

The group **Text**:

Text. The text entered in this box will be displayed next to the arrow. By default, this entry is filled with letters in the alphabetical order, starting with the letter “A”. You can manually input a text string of arbitrary length.

Offset. Sets the text offset from the arrow.

Along Arrow. Sets text offset along arrow.

The group **Arrow**”:

Length. Sets the length of the view direction arrow.

Size. Sets the size of the view direction arrow. In the case of using the **Default** setting, this parameter value is calculated based on the **Arrow (end) size** parameter setting in the command **ST: Set document parameters** (the tab **Lines**).

Line Width. Sets the line thickness of the view direction arrow. In the case of using the **Default** setting, this parameter value is calculated based on the **Thick lines** parameter setting in the command **ST: Set document parameters** (the tab **Lines**).

Type. Defines the type of the view direction arrow.

Angle. Angle of arrow rotation that will define the view direction.

The group **Placement**:

Provides the respective fields for entering X and Y coordinate values defining the element position. You can also specify the arrow rotation angle that will determine the view direction.

Color, Level, Priority, Layer and **[Font]** are defined in the same way as in other T-FLEX elements.

The option allows us to copy parameters from another already existing arrow view.

View Notation

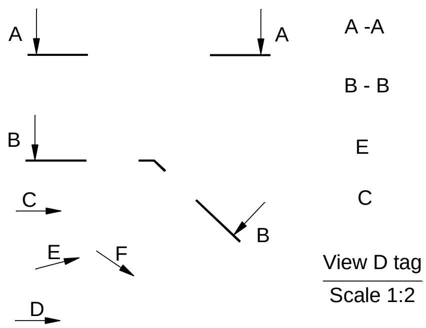
To create a view notation, press the automenu icon or the key <V>.

A view notation starts rubberbanding on the screen. Move the pointer to the desired position on the drawing and click to fix the element. In this way, the view notation will be positioned in the absolute coordinates that can be entered exactly in the “View Properties” dialog box (see below).

For attachment to a 2D node, you need to use the option <N> (the automenu icon) or rely on the object snapping. As the pointer approaches a 2D node or a line intersection, the respective entities are highlighted. You can then click .

The default text is automatically created next to the leader of the view notation. The first such created element is labeled “A-A”, the subsequent ones – the respective letters in the alphabetical order, as “B-B”, etc.

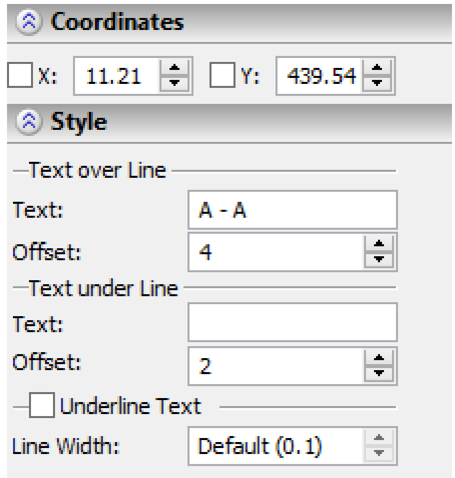
You can copy the text from an existing section or arrow view. To do this, press the automenu icon or the key <C>. Next, select by the mouse the desired arrow view notation or section.



Coordinates. The input boxes for specifying the exact X and Y coordinates when positioning the view notation in the absolute coordinates.

The group **Style**:

Text over line. The text string is positioned above the leader. You can manually input a string of an arbitrary length.



Offset. The distance from the leader to the text above it.

Text under line. The text string is positioned below the leader. You can manually input a string of an arbitrary length. Not displayed by default.

Offset. The distance from the leader to the text below it.

Underline Text. This flag will underline text of the view notation.

Line Width. Sets the width of the dividing leader line.

The option allows us to copy parameters with another already existing view notation.

Editing View Notation

The attachment, notation position and parameter values of a view can be modified by the command **ESE: Edit Section** (the option <F4> in the command **SE: Create Section View**):

Keyboard	Textual Menu	Icon
<ESE>	Edit > Draw > Section	

Upon calling the command, the following icons become available in the automenu:

	<*>	Select All Elements
	<Esc>	Exit command

A view can be selected by pointing the mouse to it and clicking, or by multiple selections. As in the case of other drawing elements, multiple selections are done by the option. Using while holding down the key <Shift> adds an element to the list of selected, while with the key <Ctrl> - excludes from the list of selected.

In the case of multiple selections, you can use the options:

	<P>	Properties (accessible only when same-type elements are selected)
	<Del>	Delete selected element(s)

When a single element is selected, the set of the available options depends on the type of this element.

Axes Creation

This command is provided for automatic creation of axes (centerlines) for graphic entities. The axes created in this way maintain associative relationship with their reference elements and adjust to modifications of those elements.

Axes Creation

Axes creation is done with the **AX: Create Axis** command. **The command can be invoked in one of the following ways:**

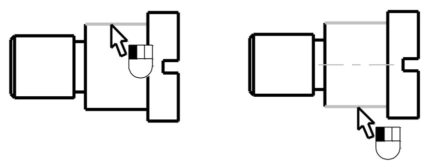
Icon	Ribbon
	Draw → Draw → Axis
Keyboard	Textual Menu
<AX>	Draw > Axis

Upon calling the command, the dash-dotted line type is set automatically, as it is used for axes. You can modify the line type either in the system toolbar CENTER2 or among the graphic line parameters. The line parameters dialog box is invoked with the option. The dialog settings affect the type of the line used by this command in the current drawing.

To create an axis, you need to select a graphic line using one of the following options:

	<1>	Create Axis of two Graphics lines

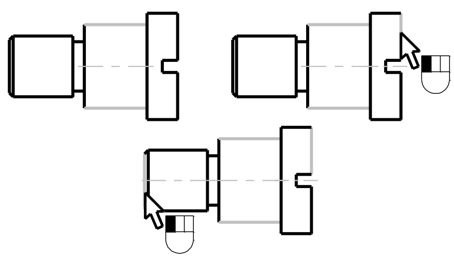
This option allows us to create an axis line between two graphic lines. You can select two straight segments or two arcs of equal radii as the source graphic lines. The selection is done with . The selected elements are highlighted.



If the object snapping is engaged in the current session, the axis being created appears as soon as you select the second segment or arc. It the object snapping is turned off, the axis is displayed only upon confirming the selection. Confirm the selection of elements for creating the axis by the option.

The axis is bounded by the projections of the end points of the selected graphic lines on the axis line.

It is possible to extend the axis on one or both sides. To do this, you need to select additionally one or two graphic lines. The axis will then be extended up to the projection point on the axis of the selected graphic line end point. If two graphic lines are

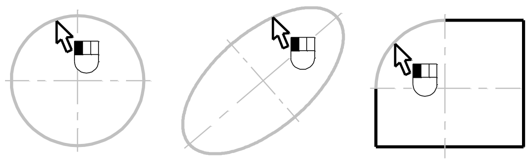


selected as the additional lines, the axis creation is completed automatically, without the confirmation by the option .

	<2>	Create two Axes of Circle or Ellipse

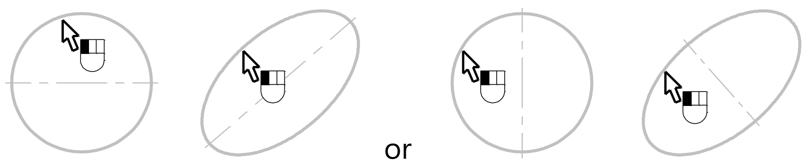
This option allows us to create simultaneously two axes for a circle, ellipse or an arc. To do this, just select the graphic entity - the circle, ellipse or arc.

The selected elements are highlighted, and the pair of axes is instantly displayed (if the object snapping is turned on). The limits of the axes are defined by the radii of the selected elements. The axes are created automatically upon the selection, without the confirmation.



	<3>	Create Horizontal Axis
	<4>	Create Vertical Axis

These options us to create, respectively, the horizontal and vertical axes for circles and circular arcs. In case of ellipses or elliptical arcs, the major axis is created instead of the horizontal one, and the minor – instead of the vertical.

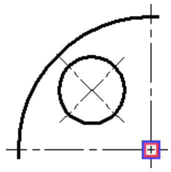
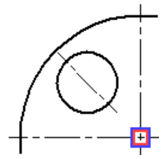


When using the following options, it is first required to specify a 2D node and then the graphic line of the circle, ellipse or arc:

	<5>	Axes make an angle with the node
	<6>	Radial axis through the node
	<7>	Tangent axis along the node
	<8>	Axis along a circle and radial axis

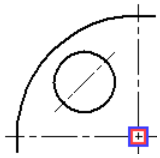
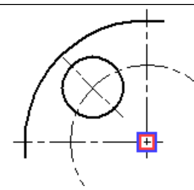
The option allows us to draw two perpendicular lines of the axis of the circle (the arc of the circle, ellipse), one of which goes through the given center.

The option allows us to create the radial (straight) axis of the circle (arc of the circle, ellipse) passing through the given center.

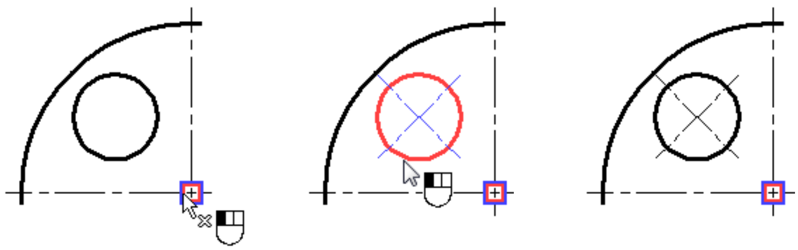
	
Axes make an angle with the node	Radial axis through the node

The option creates the tangent axis of the circle (arc of the circle, ellipse), i.e., the straight axis perpendicular to the radius that goes through the given center.

The option allows us to draw the axis on a whole circle and the radial (straight) axis, going through the given center, for the circle or ellipse.

	
Tangent axis along the node	Axis along a circle and radial axis

To create axes with the use of the , , and options, it is first required to specify the center node and then the graphic line of the circle, ellipse or arc. After the selection, the axial lines are created automatically without the need to confirm.



It is important to note that after creation of the axes, the command will remain in the mode of axes creation with the same center node. To create one more axis (or a couple of axes), it is sufficient to select the next graphic line of the circle, ellipse or arc. To return to the stage of selection of the center node, it is sufficient to press or use the option in the automenu of the command.

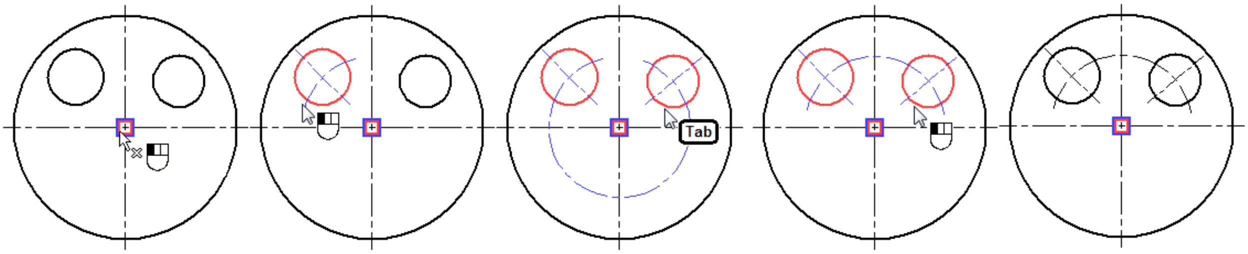
One more option of the automenu of the command allows us to create inclined axes through the center for two circles (ellipses) with an arc of the size of these circles (ellipses):

	<9>	Axis along arc and radial axis

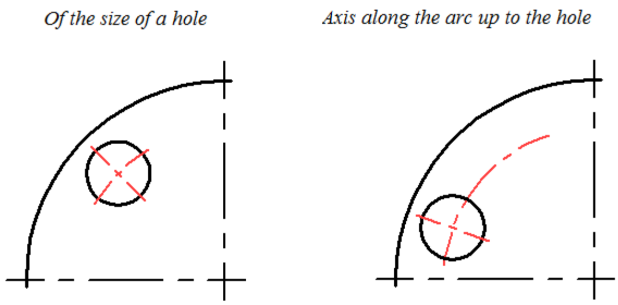
To create the axes with the help of this option, it is first required to select the center node. Then it is required to specify two graphic lines – of the circle or ellipse. When the cursor of the mouse is over the second graphic line, the view of the future axis along the arc will appear on the screen in a dynamic mode. At this moment the direction of the arc can be modified with the help of the option:

	<Tab>	Change direction of arc

To finish creation of axes, press in the window of properties or in the automenu of the command.



The second graphic line does not have to be specified; instead a 2D node can be selected. In this case is created an inclined axis with an arc of the size of one of the circles or an inclined axis with an arc up to the node.



Parameters of Axes

Axes parameters are the standard graphic line parameters that can be defined at any time by calling the parameters dialog box with the option:

	<P>	Set Graphic line parameters

The detailed description of graphic line parameters is provided in the “Graphic Lines” chapter.

Editing Axes

Since the created axes maintain associative relationship with their reference elements, their modifications occur together with the reference elements. Axes can be deleted, or their parameters can be changed, or their name can be defined, by using the context menu that appears upon right clicking over an axis, or help of the **EG: Edit Graphic Line** command (see the “Graphic Lines” chapter).

CHAMFERS

T-FLEX CAD allows creating chamfers and various fillets without any preparatory constructions. The existing elements of the drawing are modified, and new ones created in the process.

Chamfer Creation

To create a chamfer, use the command **FE: Create Chamfer**. The command is called in one of the following ways:

Icon	Ribbon
	Draw → Draw → Chamfer
Keyboard	Textual Menu
<FE>	Construct > Chamfer

To create a chamfer, you need to do several subsequent steps:

1. Select the chamfer type and set its parameters.
2. Select the defining nodes or graphic lines.

Upon calling the command, a dialog box is displayed on the screen for defining the necessary parameters and selecting the type of the chamfer being created. (By default, the dialog box is instantly launched by the system).

Upon confirming the selections with the graphic button **[OK]**, the following actions can be performed:

	<P>	Set command options
	<N>	Select Node
	<Space>	Select Graphic line
	<Esc>	Exit command

T-FLEX CAD supports three main techniques of creating chamfers:

Chamfer creation by selecting the node through which the hatch contour is passing. The chamfer is displayed in this case as a construction entity.

Chamfer creation by selecting the node through which two graphic lines are passing. The chamfer is displayed in this case as a graphic entity.

Chamfer creation by selecting graphic lines. The chamfer is displayed in this case as a graphic entity.

> Note that the corner chamfer can be created based on straight graphic lines only.

If the selected elements are not appropriate for the selected chamfer type or were selected incorrectly, then a dialog box is displayed with an appropriate error message.

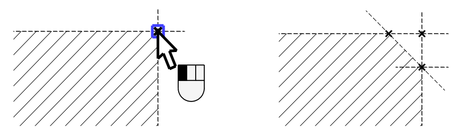
Chamfer Creation by a Node through which the Hatch Contour is Passing

This technique is intended mainly for creating chamfers at the drafting stage when the graphic lines are not created yet. A chamfer created in this way is drawn as a construction entity. Later, you will have to apply graphic lines over this chamfer manually. On the other hand, if the graphic lines were created before introducing the chamfer, then the chamfer will be displayed as a graphic entity. This technique is intended for constructing only isolated chamfers. Therefore, when constructing chamfers on the surfaces of revolution or two-sided chamfers on the edges of faceted parts, you will have to add the missing construction lines manually.

> Remember that this chamfer creation technique works only in the case when the parameter is set among the operation parameters, "Auto change Hatches".

After selecting the type of the chamfer being created and defining the necessary parameters, you need to select the node at which the chamfer will be constructed. This is done by the option .

The selected node must belong to a hatch contour. Upon selecting the node, the chamfer and the construction entities used for its creation are constructed automatically according to the specified parameters.



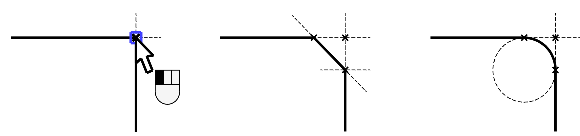
The direction of creating the angular chamfer is determined by the system automatically (depending on where the hatch contour lies).

Chamfer creation by the node through which two graphic lines are passing

This technique is used when the graphic lines are already applied on the drawing. A chamfer created in this way is drawn as a graphic entity. All construction entities necessary for the chamfer creation are produced automatically.

> When creating a chamfer in this way, make sure that no more than two graphic lines are passing through the selected node. Otherwise, we recommend using another way of chamfer creation, described below.

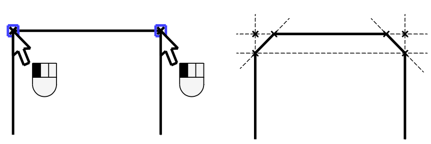
To create an isolated chamfer, in the operation parameters select the desired chamfer type and set the necessary parameters. After that, specify the node through which two graphic lines are passing.

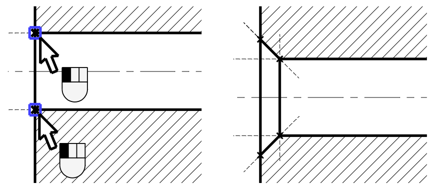


The direction of creating the angular chamfer is determined by the system automatically (depending on where the graphic lines lie).

For inner chamfer creation, we recommend using the special chamfer types intended for this purpose. The pointer should be positioned in this case on the desired side of the chamfer creation.

When creating dual chamfers, upon defining the parameters just select two nodes where two respective pairs of graphic lines intersect. The pointer position is of no importance in this case.





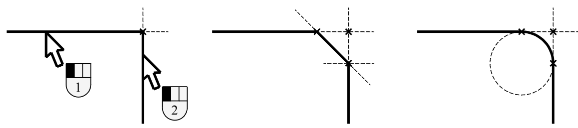
Chamfer Creation by Graphic Lines

This technique, as well as the previous one, is used when the graphic lines are already applied on the drawing. The chamfer created in this way is displayed as a graphic entity. The existing graphic entities are modified, and all necessary construction entities necessary for the chamfer creation are produced automatically in the process.

When creating an isolated chamfer in this way, then, upon selecting the chamfer type and defining the necessary parameters, you need to specify two graphic lines intersecting in one point. The graphic line selection is done by or by the option:

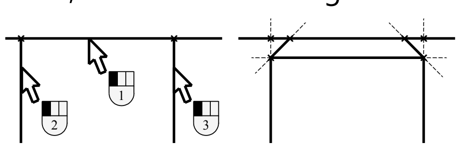
	<Space>	Select Graphic line

In this case, the chamfer distance will be counted from the first selected graphic line.

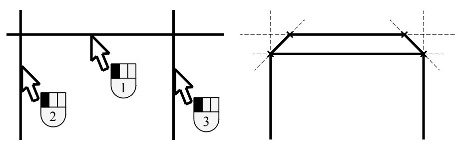


When creating dual chamfers, then, upon defining the necessary parameters, you just need to select their common graphic line lying in between. In this case, each end point of this line should be connected with exactly one graphic line.

In the ambiguous cases, when more than two lines pass through the end nodes, or the lines intersect without passing through the end nodes, then three graphic lines need to be selected for the dual chamfer creation, in the order starting with the common line of the to chamfers.



or



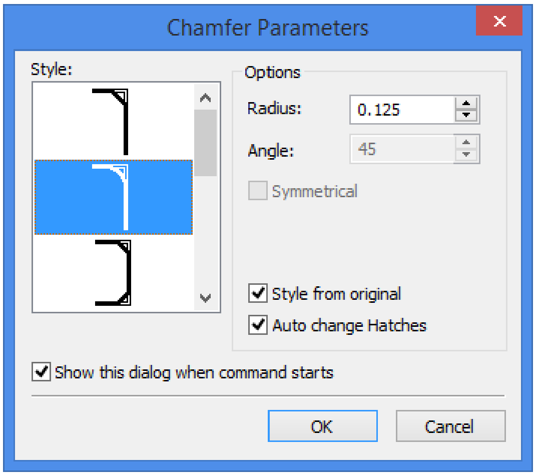
When creating any type of a chamfer, error messages are displayed on the screen in the cases of incorrect element definition.

Chamfer Parameters

Since the chamfer parameter definition is the first step of its creation, the default system behavior implies launching the chamfer parameters dialog box immediately upon calling the command. Otherwise, the dialog box is called by the option:

	<P>	Set command options

You can select a predefined type of a chamfer in the **Type** pane of the dialog from the menu of icons.



Radius/Distance. Defines the radius of the inscribed circle in the case of the fillet-type chamfer, or the distance in the case of the corner chamfer.

Angle. This parameter is accessible only when the corner chamfer is selected, and defines the angle of the chamfer being created.

Symmetrical. This parameter is accessible only when making the corner chamfer. Setting this parameter grays out the parameter **Angle,** since only the distance is required for defining the symmetrical chamfer.

Style from original With this parameter set, the chamfer-making line strokes will have the same parameters as the graphic lines on which the chamfer is based. Otherwise, those strokes will have the parameters currently used by the command **G: Create Graphic Line or SK: Create Sketch.**

Auto change Hatches. Setting this parameter causes automatic adjustment of the defining hatch to geometry of the chamfer being created. It also makes possible creating a chamfer as a construction element at the node through which the hatch contour is passing.

Show this dialog when command starts. If this parameter is set, this dialog box will be automatically launched upon entering the command. If the parameter is cleared, you will have to call the dialog by the option "Set command options" (the icon )

WELDING

In T-FLEX CAD there is a group of commands intended for design of welded parts. Welding commands are gathered in the **Tools > Welding** submenu. With the help of these commands it is possible to create on a 2D drawing or a 3D model various types of standard welded seams, and if necessary nonstandard ones. For created welded seams it is possible to automatically create captions and make tables of welded seams.

To invoke welding commands, the main toolbar can be used (the "Welding" set).

This chapter describes the commands used for creation of welded seams on a 2D drawing, i.e., 2D welded seams.

General Information

T-FLEX CAD allows us to create three types of elements intended for design of welded parts: *welded seam types*, *welded seams*, *welded seam captions*. All these elements are interconnected. The structure of connections between them is displayed in the tree of welded seams in the system's service window "Welded seams" (see below).

Welded seam type – a special element of T-FLEX CAD that itself stores characteristics (description, name, regulatory norms, geometric parameters) of the welded seam used in the current document of T-FLEX CAD. T-FLEX CAD document can contain several types of welded seams.

> Welded seam types created in the current document of T-FLEX CAD are common for 2D and 3D welded seams.

Welded seams types required in the current document are determined by the user. To specify characteristics of the welded seam types it is possible to use the standard database of welded joints that is available in the system. If necessary, the user can specify arbitrary characteristics.

> The types of welded seams created on the basis of welded joints of T-FLEX CAD are stored in the current document of T-FLEX CAD. But the connection with the database is not stored. This method of storage of welded seam types allows us not to worry about the presence of welding database when transferring the document.

Specifying only the types of welded seams is not sufficient for design of welded parts. The types determine which welded seams can be present in the current document of T-FLEX CAD, but they do not specify the welded seams themselves. Using the welded seam types specified in the document, the user has to create other elements of T-FLEX CAD – welded seams.

Welded seam – element of T-FLEX CAD that defines the welded joint existing in the given drawing (or 3D model). In T-FLEX CAD the welded seams are divided into 2D welded seams and 3D welded seams depending on method of specification.

> This chapter describes only 2D welded seams, i.e., the seams that are not directly connected with the elements of a 3D model.

When creating the welded seam, the user indicates to which of the types predetermined in the given document the given seam refers. In doing so the basic characteristics of the created welded seam are defined − they will be taken from the parameters of the specified type. If needed, the user can specify a number of individual parameters for a welded seam that are not defined in the corresponding type.

2D welded seam can be both free and associated with 2D drawing's elements (graphic lines or paths). Free seam is determined only by its type and, if necessary, by the individual parameters. It is not associated with a 2D drawing, but it is present in the current document, displayed in a welded seams tree and taken into account when making the welded seams table. The length of such a weld is defined by the user.

If, when creating a 2D welded seam, we specify the elements of a drawing with which the seam must be associated, then in the future these elements will be treated as the image of the given welded seam on the 2D drawing. The length of such a welded seam can be either specified by the user or automatically calculated by the system based on the length of 2D elements with which it is associated.

> When creating a welded seam, the user must determine whether the welded seam being created will be indicated on the drawing. If yes, then upon seam creation the user should indicate the drawing's elements which will be the image of the welded seam. Otherwise, it is sufficient to create free, i.e., not associated with any drawing's elements, welded seam.

Welded seam caption – a special element of the "Label" type, which serves for designation of the welded seam on the drawing. To create a welded seam caption, an additional command is used. It constitutes an adapted version of the command for creating notes. Caption is automatically generated on the basis of parameters of the indicated welded seam. If necessary, the user can edit parameters of the note-caption of the welded seam. In the future created note-caption retains connection with the parameters of the welded seam.

From welded seams created in the current document *a table of welded seams* can be created. Welded seam table constitutes a text report about welded seams generated in the current document. The contents of the welded seam table (available types of seams, number of seams of each type, the total length of the seams of each type) are created automatically by the system. The table can be created in the current document of T-FLEX CAD and also exported to the Excel document. If necessary, the user can specify his own ways of creation of the welded seam table by creating corresponding macros.

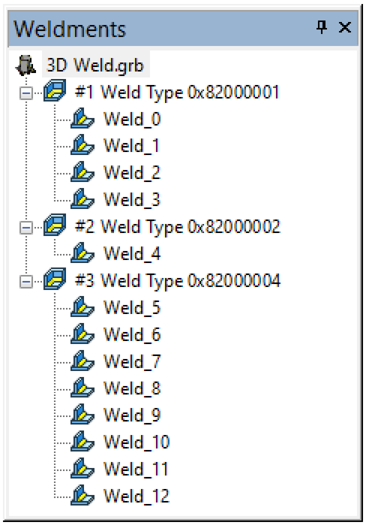
«Welded Seam» Window

All types of the welded seams specified in the current document of T-FLEX CAD, the welded seams themselves and their captions are shown in the system's special window "Welded seams". Control of this window visibility is carried out with the help of the icon found in the set of buttons "Welding" of the main toolbar by the command **WW: Show weld project window**:

Icon	Ribbon
	Assembly → Assembly → Weld → Weld Project Window
Keyboard	Textual Menu
<WW>	Tools > Weld > Weld Project Window

In the given window the elements of welding (welded seam types, welded seams themselves and their captions) are sorted by the types and are represented in the form of a tree. The welded seam tree allows us to visualize intuitively all information on the welding's elements created in the current document (in the drawing or 3D model of a product).

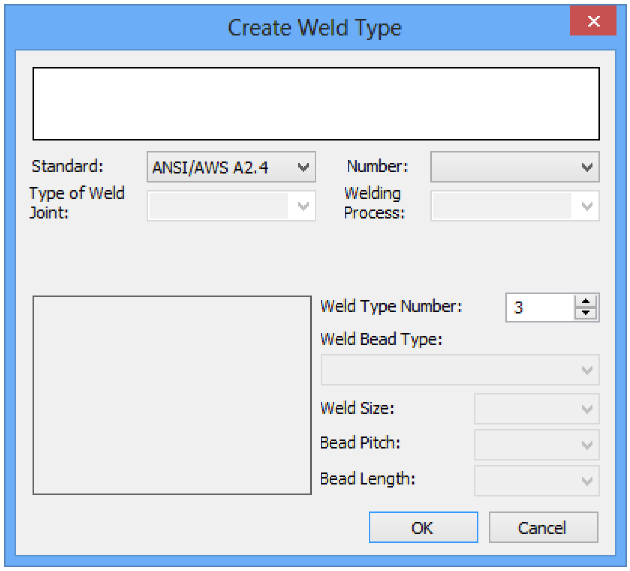
The context menu invoked with the help of when clicking on any point of the "Welded seams" window or upon selection of any element in the given window contains various commands for creation and editing of the welding's elements.



Specifying welded seam type

Specifying the new type of the welded seam is carried out with the help of the special command **Create welded seam type**. It is possible to invoke this command with the help of the icon found in the set of buttons "Welding" of the main toolbar or from the context menu in the "Welded seams" window. In addition, the command of creation of the new welded seam type can be invoked directly from the command of creation of the welded seam.

After the command is invoked, the window of the dialog appears in which parameters for created welded seam type are selected. When working in this dialog, firstly the user has to indicate the **regulatory norms** for the welded seam type being created (GOST, ANSI/AWS A2.4, ISO 2553,). Depending on the selected norms, the **number** is selected according to the norms. The selected **regulatory norms** and **number** determine the remaining contents of the given dialog.

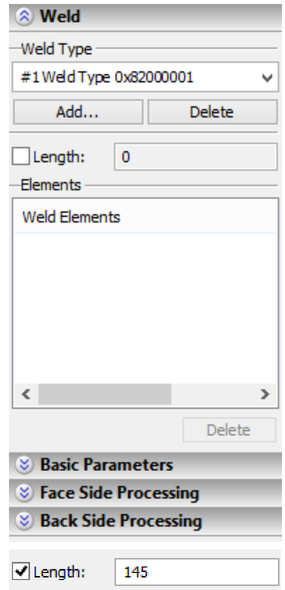


Creating Welded Seam

To create a welded seam, the **WE: Create welded seam** command is used. Invoking this command is carried out by one of the following methods:

Icon	Ribbon
	Assembly → Assembly → Weld → Weld
Keyboard	Textual Menu
<WE>	Tools > Weld > Weld

To create a welded seam, after invoking this command it is required to select the welded seam type in the "Welded seam" section of the properties window. In the drop down list those types of welded seams which were defined in the given document will be present.

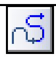


If the required welded seam type is not yet created, it is possible to use the[Add...] button. When pressing this button, the command of creation of the new welded seam type will be started. After completion of the type creation the system will return to the command of welded seam creation. Created seam type will automatically be selected.

If not a single welded seam type is defined in the current document at the moment of launching the “WE: Create welded seam” command, the command of welded seam type creation will automatically be invoked.

After selection of the type, it is possible to complete the seam creation. 2D welded seam not associated with the drawing’s elements will be created. Its length by default is equal to zero. If necessary, the length of the welded seam can be specified manually. To do so, it is required to enable the “Length” flag and specify the desired length of the welded seam in the input field that appears to the right.

To link the welded seam being created with the lines of a 2D drawing, it is required to indicate the graphic lines or 2D paths in 2D window with the help of the options of automenu:

	<I>	Select graphic lines
	<P>	Select paths

By default, both of these options are enabled.

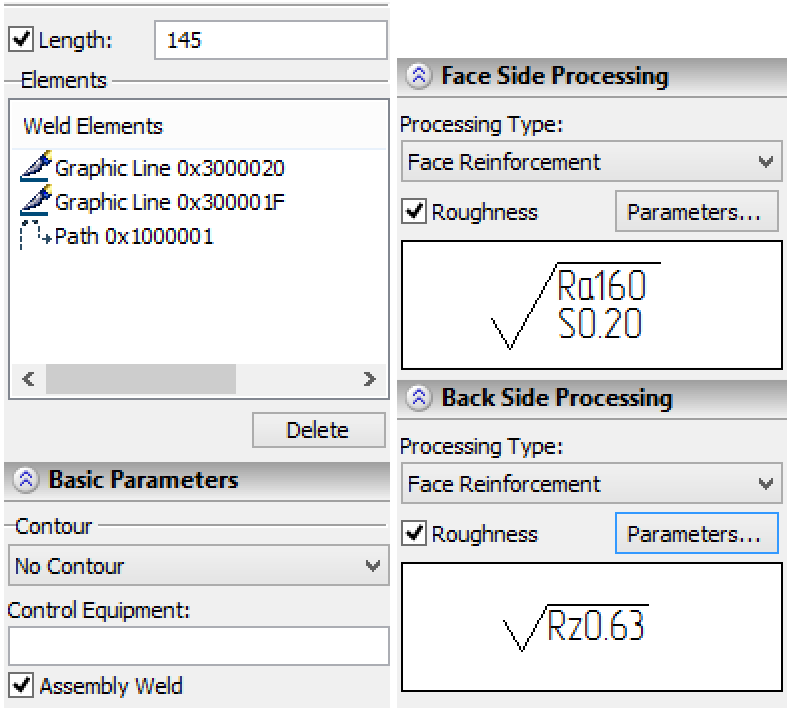
The elements selected in 2D window are added to the “Elements of welded seam” list. To refuse selection of any element, it is required to select it in the list with the help of and press the [Remove] button.

By default the length of welded seam associated with the drawing’s elements is determined automatically based on the length of selected lines or paths. If necessary, the length can also be specified manually by enabling the flag “Length”.

In the sections “Main parameters”, “Processing front side” and “Processing back side” it is possible to specify individual parameters of the welded seam being created.

In the “Main parameters” section the following parameters can be specified:

- Contour:
- no contour;
- by closed contour;
- by open contour;
- Field seam: yes/no;
- Caption of controlling complex or of the seam’s control category.



In the “Processing front side” and “Processing back side” sections, the ways of processing the front and back sides of the welded seam are specified. The “Processing type” parameter found in each of these sections allows us to select one of the following types of processing for each side of the welded seam: “No processing”, “Remove reinforcement”, “With smooth transitioning”.

To specify roughness of the front or back sides of the welded seam after machining, the **Roughness** flag in the corresponding section of the properties window should be enabled. As a result the **[Parameters]** button will become available to the right of the given flag. When pressing this button, the roughness’s parameters dialog will open. After specifying the desired parameters and closing the dialog, the image of the given roughness will appear in the preview field located below.

> All individual parameters of the welded seam described above can automatically be transferred to the module of technological design “T-FLEX Technology”.

Creating Welded Seam Caption

Caption of the welded seam is created on the drawing with the help of the **WN: Create Weld Symbol** command. Invoking this command is carried out by one of the following methods:

Icon	Ribbon
	Assembly → Assembly → Weld → Weld Symbol
Keyboard	Textual Menu
<WN>	Tools > Weld > Weld Symbol

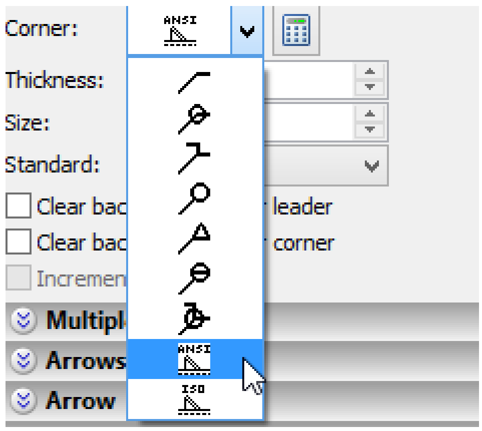
After invoking the command, the automenu and properties window will become available, which are similar to the caption creation command by the structure of elements. In the automenu there is additional option that allows us to select the welded seam on the drawing:

	<R>	Select weld

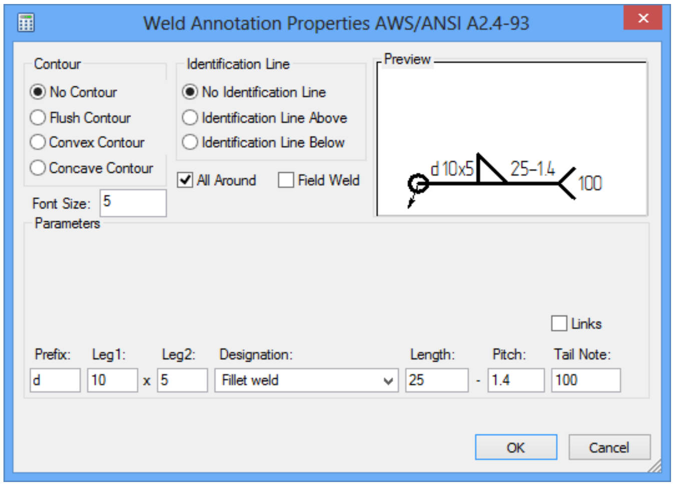
To select a seam, it is required to move the cursor closer to those lines or 2D paths of the drawing with which the welded seam is associated. When the seam is highlighted, it can be selected by pressing . On the dynamic image of the note snapped to the cursor the text caption of the selected welded seam will appear. Next this caption is tied to the drawing in the same way as the ordinary note.

> Note that only welded seams that are associated with lines of the drawing can be selected in this way. The welded seams that are not associated with lines of the drawing can be selected only in the "Welded seams" window. Moreover, such a welded seam can be selected with the help of the "Other" item of the context menu if the context menu is invoked in 2D window in the command waiting mode.

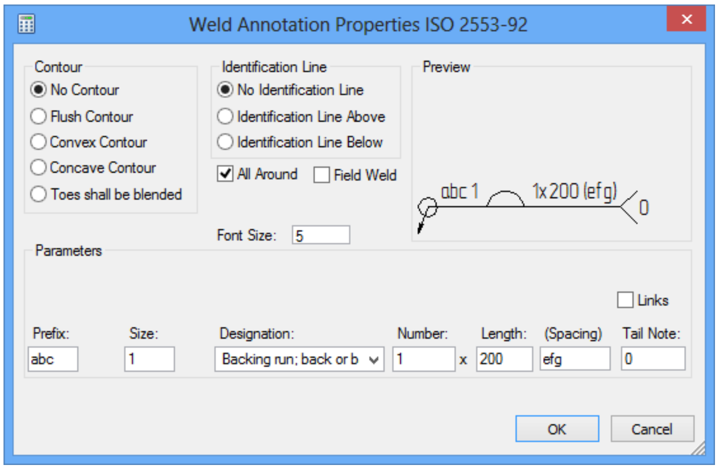
For ISO and ANSI standards the welded seam captions are specified manually. To do so, in the properties window in the «**Type**» section it is required to select the ISO or ANSI standard from the list.



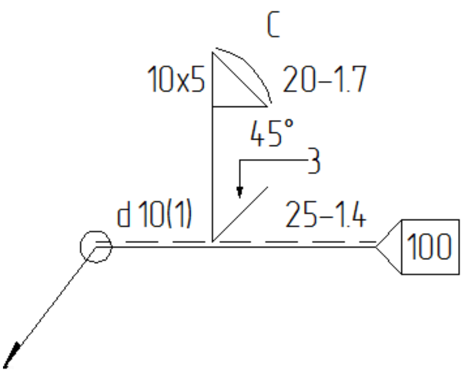
All required parameters of the note are specified in the dialog window «Note's properties». This dialog window is invoked by pressing the icon.

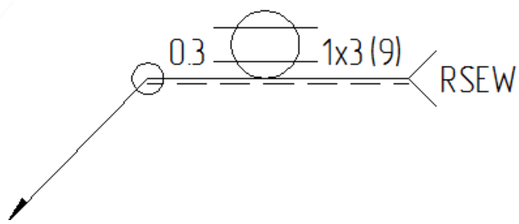


Different dialog windows will appear for the ANSI and ISO standards.



After all input values are confirmed and all necessary parameters for a welded seam are specified, the welded seam caption will be created on the drawing.

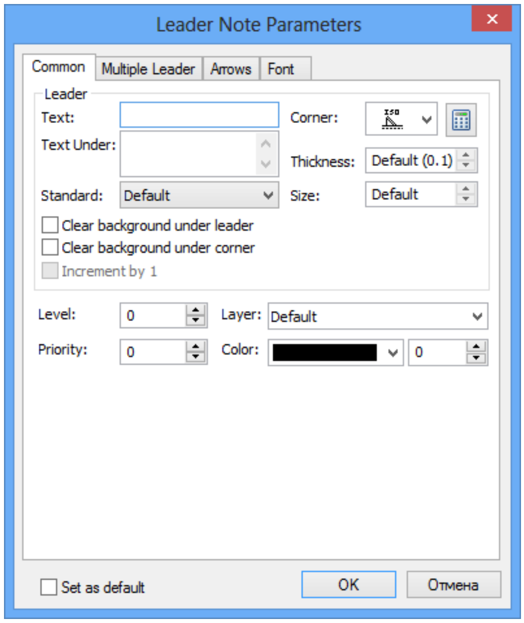




The note's parameters can also be specified in the parameters dialog that is invoked with the following option of the automenu:

	<P>	Note's parameters

Parameters on the tabs of this dialog duplicate parameters in the properties window.



> The note's parameters dialog can also be invoked in the command waiting mode from the context menu of the note (invoked with). Owing to this capability this dialog can be used for immediate change of the note's parameters without invoking the command of editing.

T-FLEX CAD allows the user to input his own welded seam captions. User's captions are created in the same way as regular parametric fragments in the library in the folder "System\Welding Symbols".

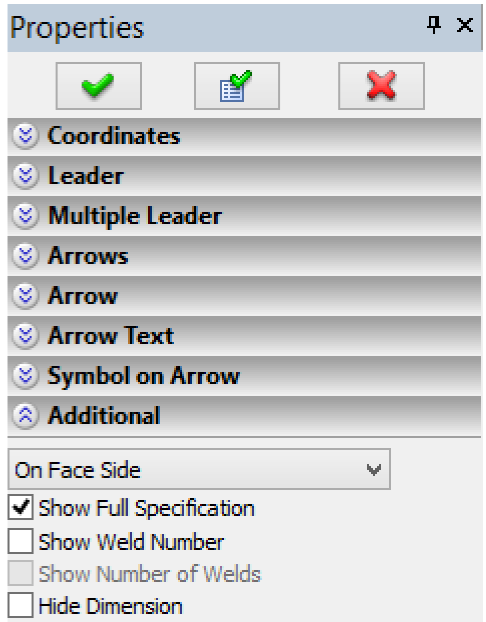
In the command's properties window there is the "Additional parameters" section that contains the following parameters of the welded seam caption:

Seam's location. This parameter shows from which side of the seam the caption is positioned: **"On Face side"** (the seam's caption will be positioned on the leader of the note) or **"On back side"** (the seam's caption will be positioned under the leader of the note).

Full caption of a seam. If this flag is enabled, then the full caption of the welded seam is indicated on the caption. When this flag is disabled, the shortened caption that contains the seam type number and individual parameters of the given seam are indicated.

By default, for each type of the welded seam the first extension line is created with the full caption, all subsequent ones for the given type of the welded seam – with the shortened caption.

Show sequence number. When this flag is enabled, the number of the given welded seam type is shown on the arrow of the the note-caption. This parameter is available only when the "Full caption of a seam" flag is enabled.



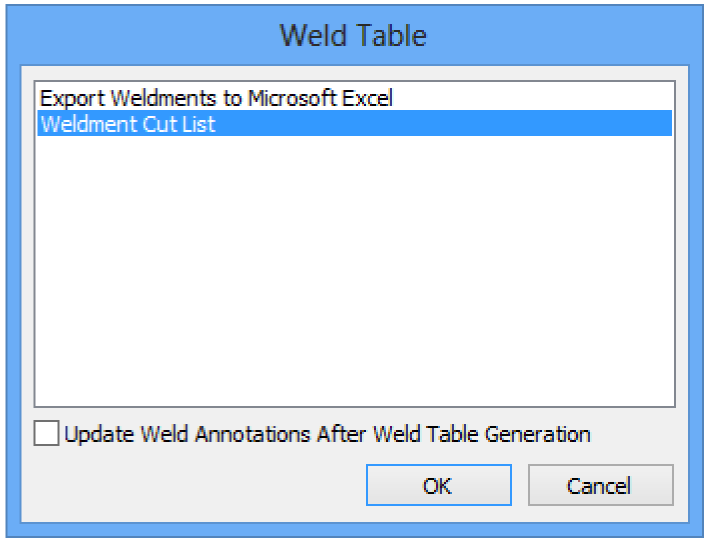
Show number of seams. This parameter is available only when the "Show sequence number" flag is enabled. If it is enabled, before the welded seam type number, which is shown on the note's arrow, the total number of welded seams of the given type in the current document is specified.

Hide dimension. If the given flag is enabled, then the seam's leg dimension is not shown in the full caption of the welded seam. When the flag is disabled, the full caption with indication of the leg's dimension is specified.

Creating Welded Seams Table

Creation of welded seams table is carried out with the help of the **WT: Create welded seam table** command. The command is invoked:

Icon	Ribbon
	Assembly → Assembly → Weld → Weld
Keyboard	Textual Menu
<WT>	Tools > Weld > Weld Table



After this command is invoked, the dialog window appears in which the user has to choose the method of creation of the table of welded seams. Initially two options are present:

Simple table of welded seams. Welded seam table is created on the first 2D page of the current document. Approximate appearance of this table is shown on the figure below.

ITEM NO.	QTY.	DESCRIPTION	LENGTH
1	1	Fillet weld 12	145,00
2	1	Plug weld 15	100
3	2	Seam weld 7	115

Export of welded seams into Microsoft Excel. Welded seam table is exported into the document of Excel.

Creation of welded seam table in both cases is carried out with the help of macros found in the folder *"…/T-FLEX Parametric CAD 15/Program/WeldReport"*. If necessary, these macros can be modified independently by defining inside them another type of a welded seams table. It is also possible to create user's own macros for creation of welded seams table and place them into the same folder. In this case in the aforementioned dialog the additional options will be present that correspond to the user's macros.

> The work with macros is described in more detail in the "Macros" chapter.

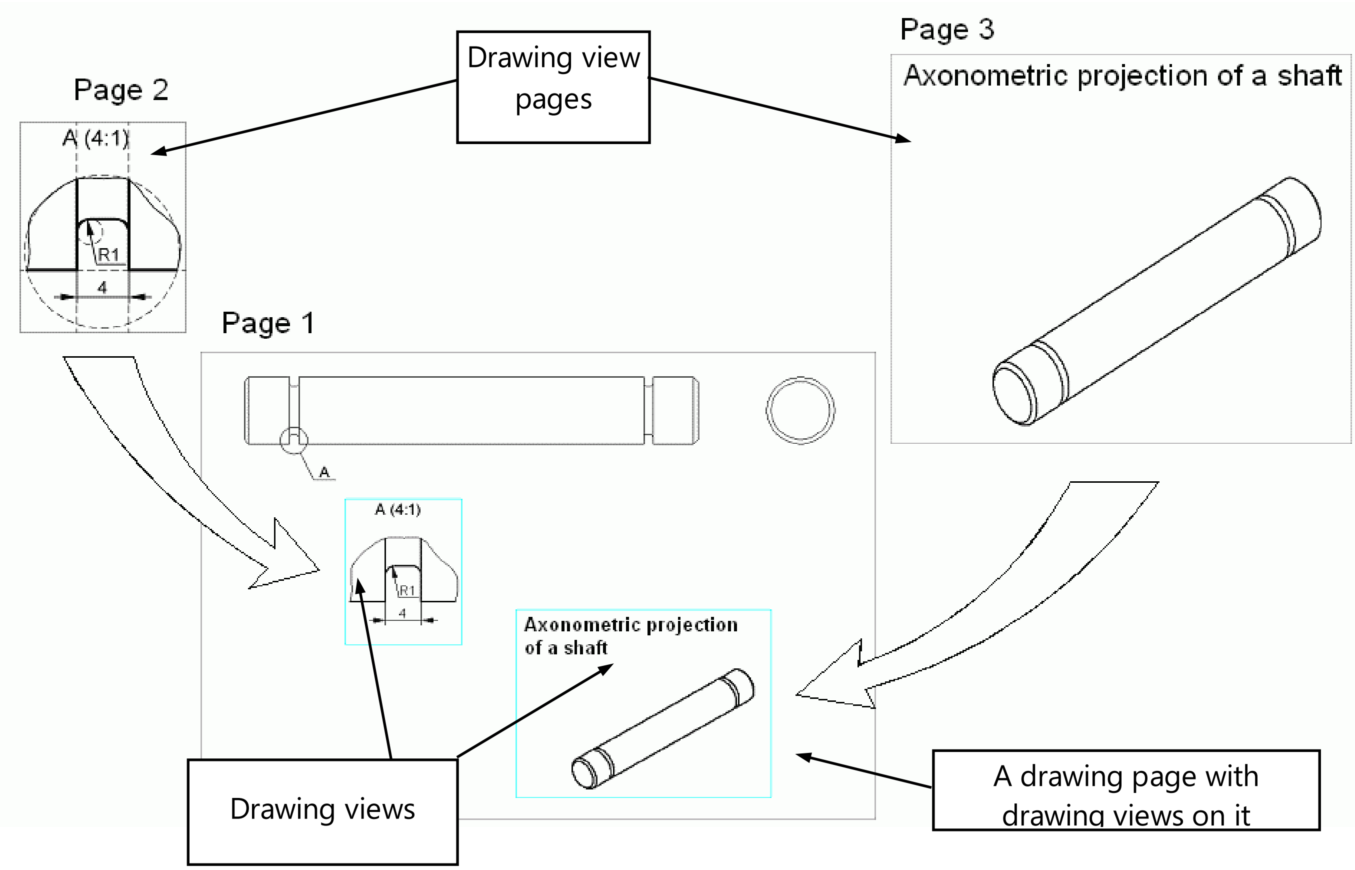
Supplementary Drawing Elements

DRAWING VIEWS. DETAIL VIEWS

This chapter describes an auxiliary element of T-FLEX CAD system – the drawing view. Additional views, detail views, local section views and sections – all these instances are supported by Drawing View. This functionality handles situations when elaboration to the images on the main views is required. This can be an additional view, and, in particular, a scaled view. The functionality allows collecting together on one page elements from different pages, inserting an image (or a portion thereof) from one page into another page in various scales, even create a simple assembly from parts contained in the same document.

Main Concepts

A drawing view is an element of T-FLEX CAD that allows displaying contents of one page (or a portion thereof) on another page, scaled to the desired factor. It is a rectangular area of specified size used for displaying the contents of another page.

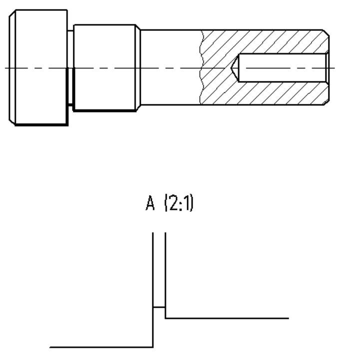


The page displayed within a drawing view can either be selected from the list of existing pages in the document or additionally created for the drawing view. The newly created page is assigned the type "Auxiliary". A drawing view can be nested, that is placed within another drawing view.

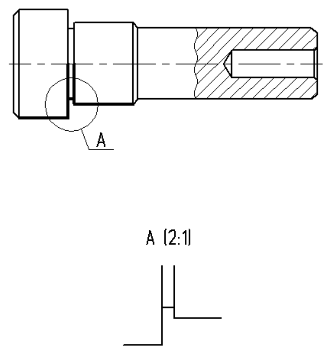
Location of the drawing view box on the main page can be specified by either direct selection of the fixing point and the view rotation angle or with the help of the fixing vector located on the view page.

The main purpose of a drawing view is to display on one page drawing elements of different scales. For example, one can create a drawing of some object on one page and then create drawing views of this object on other pages that will display portions of this drawing to various scales.

Besides, drawing views can be used simply for collecting the contents from different pages. This may be used for adding to the main drawing some additional, local views and sections created on other document pages in different scales. This capability is the most common way of using drawing views in T-FLEX CAD. Note that the images on the detail views can be copied from other pages of the current document when the drawing view is created (see section "Creating Detail View"), and enhanced with captions of the view notations (see section "Creating Detail View with Caption").



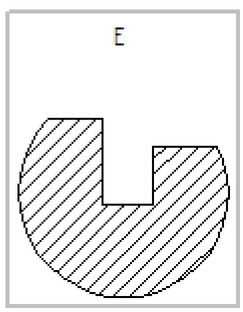
Detail View



Detail View with Caption

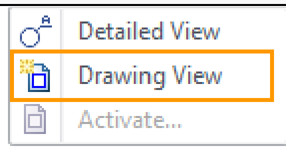
The image displayed on a drawing view can be created or edited either on the original page of the drawing view elements, or directly in the view area of the main page by activating the view.

Limits are shown on the drawing for drawing view and detail view. The limits are not printed. If you want to change color of the limits use the option **Limits of inactive drawing view** on the **Color** tab in the **SO: Set system options** command. You can specify colors of the other drawing view elements on the tab.



Creating Drawing Views

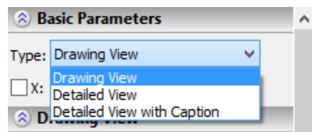
Drawing views are created using the command **SD: Create Drawing View**:

Icon	Ribbon
	Draw → Draw → 
Keyboard	**Textual Menu**
<SD>	Draw > Drawing View

After invoking this command, the following options will appear in the auto menu:

	<O>	New Drawing View
	<V>	New Detailed View
	<C>	New Detailed View with Caption
	<S>	Create drawing view by contour

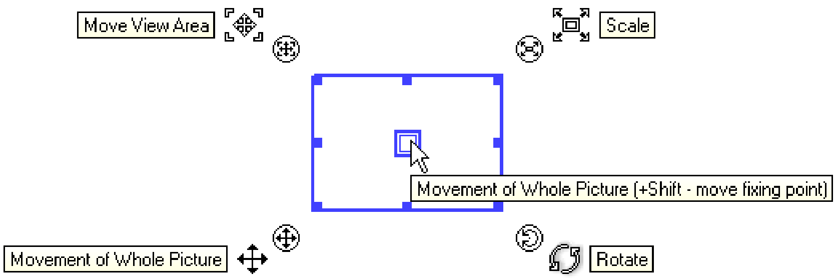
These options define different modes of creating drawing views.



These command operation modes can be also specified in the properties window (parameter **Type** in the group "Basic parameters"). The command's properties window and the auto menu work synchronously.

Working with Draggers of Drawing View

For defining location, rotation angle and dimensions of the drawing view, a special type of the cursor in the form of the dynamically movable rectangle with markers is used. Markers can be used for modifying location, rotation angle, the scale and the size of the view, location of the fixing vector or fixing point.



The markers are used as follows: a marker is activated simply by pointing the cursor and clicking. Then the marker can be moved to the desired position. The second click fixes the new position of the view. Alternatively, the marker can be "dragged" by moving the cursor with the mouse button held down. In this latter case, the placement of the view in the new position occurs on releasing the mouse button.

As the placement or parameters of the view are modified, the original state stays in display.

To change the view box size, the square-shape markers are used, located in the corners and at side midpoints of the view box. When pointed at by the cursor, the latter changes to a two-headed arrow ↕ .

The marker provides for modifying the view scale, that is the scale factor of the page display in the view box. The exact value of the scale can be entered in the view parameters dialog box.

The marker allows changing the view rotation angle about the fixing point. (In the case the view was based on an existing page and a fixing vector, this will be the rotation angle of the fixing vector.) The exact value of the rotation angle can be entered in the view parameters dialog box.

For moving the view image relative to the page, on which it is located, the marker or the marker of the fixing point of the view (in the form of the filled square) can be used.

Location of the fixing point of the view is shown with the marker in the form of the unfilled square. When creating the drawing view on the basis of the existing page with the help of the fixing vector, the location of the given marker coincides with the origin of the fixing vector.

For modifying the fixing point of the view, point at the marker for fixing the view while holding the button <Shift>. After that, to select the fixing point, you can pick (by pressing) one of the characteristic points of the view box (corners of the view box and the midpoints of its sides). You can also specify the fixing point of the view arbitrarily by using <Ctrl>+ .

> It is also possible to change the location of the fixing point of the view with the help of the option of the auto menu for the command of creating/editing the drawing view.

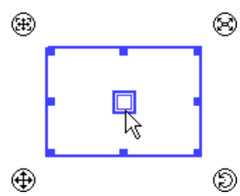
For moving the drawing view area (that is, the drawing view box relative to the page displayed in the view), the marker is used.

> Originally, the boundaries of the box coincide with the boundaries of the displayed page. Shifting the view with respect to its page makes sense only when a portion of the page is supposed to be displayed on the view. In this case, the "Clip Image" flag is usually checked in the parameters dialog box, and the view box manually reduced to the desired size.

Creating Drawing View

After entering into the command **"SD: Create Drawing View"** the option becomes active by default. This option allows the user to create a simple drawing view. The created view can display the contents of already existing page of the current document as well as of the new page, automatically created by the system for the given drawing view.

After activating the option the cursor starts moving a box with markers for moving the view area, rotating and scaling the view. Meanwhile, the status bar displays the help message "Select Drawing View type or set new Drawing View placement". The drawing view can be placed on the current page by specifying a point on the drawing (a 2D node or an arbitrary point).

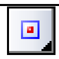


To fix the view at a node, one can use the following automenu option:

	<N>	Set relation with Node

The location of the created view (coordinates **X** and **Y** of the fixing point) can be also specified in the command's properties window (in the group **Basic parameters**).

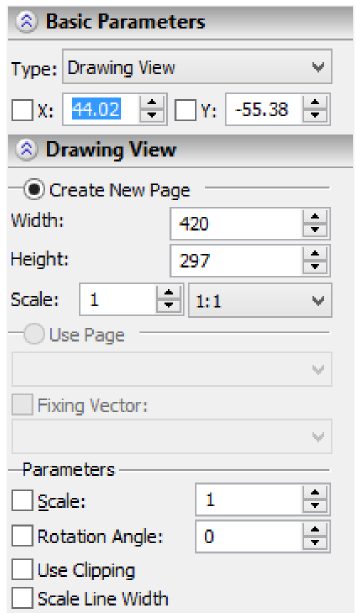
By default, the snap of the drawing view is carried out at the center of the view box. It is possible to select another fixing point of the view with the help of the option:

		Fixing Point of View

After specifying the view location, the user can define more accurately the location, the size and the scale of the view by using the markers around the view box (the use of markers is described in section "Editing drawing views" in more detail).

It is possible to specify parameters of the created drawing view in the command's properties window:

Create New Page. A new page ("Auxiliary" type) will be created along with a new drawing view. The parameters "Width" and "Height" define the page size. These also define the initial size of the drawing view box. The parameter "Scale" defines the scale factor of the view page. These settings are entered as the **Paper size** group of parameters and **Scale** parameter of the newly created drawing page on the tab **Paper** of the **ST: Set Document Parameters** command.



The created page will not contain any constructions. The user is supposed to create the necessary contents on this page himself. The editing techniques of drawing views and their contents are described in the section "Editing drawing views".

Use Existing Page. Allows selecting any existing page in the document (except the current one) from the list. This parameter is inaccessible for one-page documents. The size of the drawing view box is automatically set in accordance with the size of the selected page.

Fixing Vector. Setting this parameter allows the user to select and use the fixing vector existing on the selected page for snapping the created drawing view. The parameter is accessible only when the flag "Use Existing Page" is set and when a fixing vector exists on the current page. Note that, when using the fixing vector, it is necessary to specify two fixing points of the drawing view.

> Fixing Vector can be used for placing a drawing view, just like a 2D fragment. This also allows controlling the current page layers visibility as the page is displayed on a view. Use of fixing vectors is described in details in the chapter "Assembly Drawing".

The group of parameters **Parameters** combines auxiliary parameters of the drawing view:

Scale. Defines the scale factor of the drawing view, which is the scale of transformation of the view page as it appears on the main page.

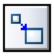
Rotation Angle. This parameter is inaccessible when using a fixing vector.

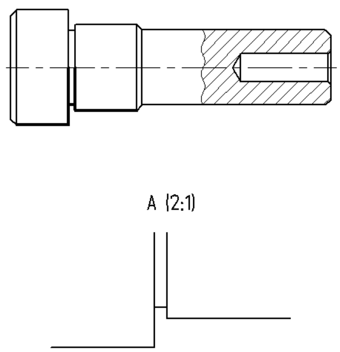
Clip Image. When set, the page in the view is clipped to the extents of the actual image contained on the page.

Scale Lines. When set, the thickness of the graphic lines on the drawing view will be affected by the "Scale" parameter.

To complete view creation, use the option . As a result, a rectangular area of the drawing view will be created, with the contents of the specified page or its portion displayed within.

Creating Detail View

The option  allows the user to automatically create the detail views on the basis of the drawing view. As a result of its use, the drawing view containing automatically created copy of the selected elements of the drawing will be created. In addition, the image on the drawing view page will be automatically augmented with the view caption.



When using this option, the following will be created:

1. A new drawing view (based on a new "Auxiliary"-type page). The drawing view parameters are set "by default";
2. A translated copy (as in the command **Drawing** > **Copy** > **With translation**) for associative copying of the selected elements onto the drawing view page with specified parameters (scale and rotation angle);
3. The caption for the detail view.

As a result, a rectangular area of the drawing view is created on the main page with copies of selected elements displayed within. The size of the drawing view, as well as the size of its respective page, is defined automatically by the system based on the size of the copied entities. If necessary, the size of the drawing view and its page can be modified manually.

For creating the detail view, it is necessary to do the following:

1. Select the elements to be copied from the original drawing;
2. Select the fixing node for the copied elements;
3. Define location of the detail view (that is, the location of the drawing view containing the detail view).

After invoking the option the options for selecting the copied elements will appear in the auto menu:

	<M>	Select Mode
	<M>	Deselect Mode
	<I>	Select Other Element

When the option is turned on all elements selected on the drawing are added to the list of the copied elements. For quick selection of several elements, you can use the selection with the window. When the option is turned off the selected on the drawing elements will be removed from the list of the copied elements. For quick selection, you can use the selection with the window here as well.

Complete the selection of the elements, which will be copied onto the detail view, by pressing .

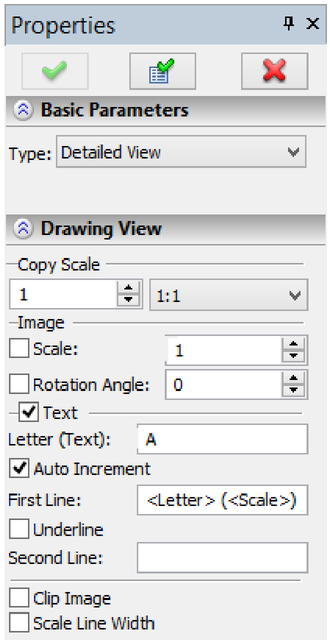
After selecting the copied elements, the system will prompt the user to specify the fixing point of these elements (it will serve as the initial point of the copy to be created). The option for selecting a 2D node will appear in the auto menu:

	<N>	Set relation with Node

After choosing the fixing node, the dynamic image of the created detail view attached to the cursor will appear on the screen. The location of the detail view on the drawing has to be selected next. After snapping the view, its location and size can be adjusted with the help of the markers for editing the drawing view.

> To make the caption of the detail view change its location according to the modifications of the drawing, it is recommended to fix the drawing view to the 2D node.

Before completing the detail view creation, the dialog for specifying parameters of the created view will remain available in the command's properties window. In this dialog you can specify the precise values for the fixing point of the view, and also the following parameters:



Copy Scale. This scale determines the scale with which the selected elements of the drawing will be copied.

The group of parameters **Image** defines parameters of the drawing view being created (these parameters can be also specified directly in the drawing's window with the help of the markers on the view image):

Scale. The drawing scale of the drawing view, that is the scale with which the drawing view page is shown in the view area;

Rotation Angle. Angle of rotation of the drawing view.

Group of parameters **Text** allows specifying parameters of the view caption:

Letter (Text):. In this field the lettered caption for the created view is shown (selected by the system automatically). If necessary, you can choose another caption.

Auto Increment. When this flag is set, for each new detail view the "available" letter is automatically selected. It is shown in the field of the parameter "Letter (Text)". When this flag is turned off, the automatic selection of the lettered caption does not occur.

First Line. The text in the first line of the view caption. This parameter supports the special format. To insert the lettered view caption (the value of the parameter "Letter (Text)") into the line, the group of symbols "*<Letter>*" is used. To insert the copy scale caption ("1:1", "1:2" ...) – the group of symbols "*<Scale>*". By default, the string "*<Letter>* (*<Scale>*)" is set for this parameter.

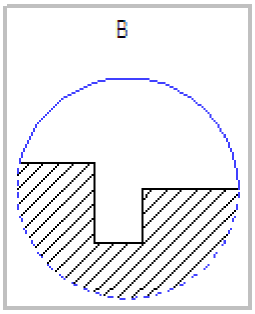
Underline. When this flag is set, the text in the first line of the view caption will be underlined.

Second Line. This parameter defines the text of the second line of the view caption.

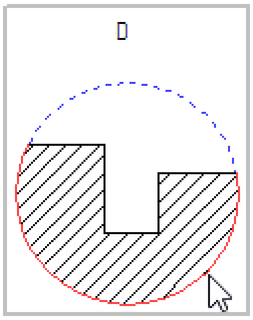
Clip Image. When this parameter is set, the view image will be cropped by the drawing view borders.

Scale Line Width. If this parameter is set, the width of the graphic lines of the detail view will be scaled according to the specified drawing scale of the view.

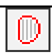
The last step of the detail view creation is the view outline creation. The system allows to define the outline after pressing .



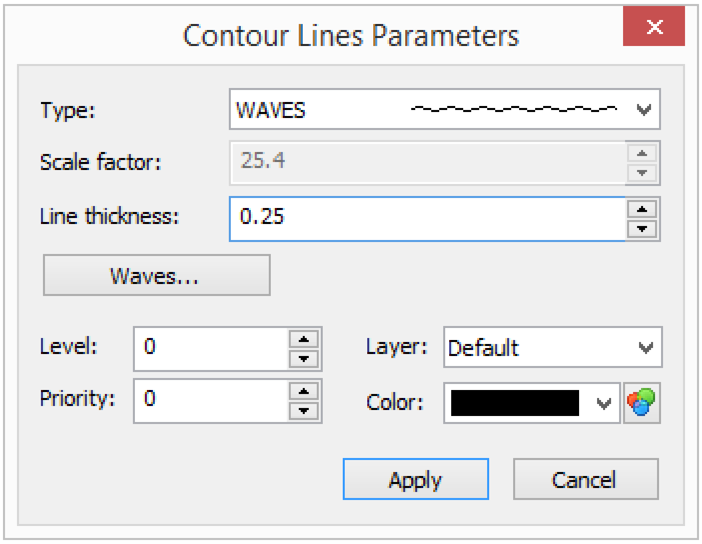
A dotted line appears around the selected area for the detail view. The line is divided into segments. A click into a segment enables/disables the display of the selected segment.



The following options are available:

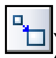
	<S>	Show all segments of detail view outline
	<H>	Hide all segments of detail view outline
	<I>	Invert visibility of detail view outline segments

You can specify parameters for the outlines using the option .

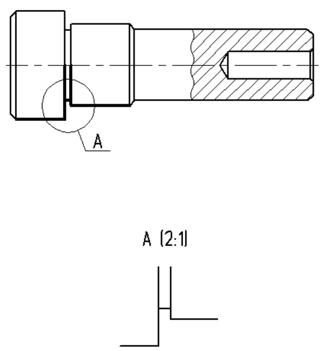


To complete view creation, use the option . For the view constructed, the caption will be automatically created. The obtained view can be edited and augmented, if required.

Creating Detail View with Caption

Creating the detail view with the caption with the help of the option  is an extension of the case of creating just a detail view (the option ). The option  creates the detail views clipped by the specified contour and with automatic creation of the caption for the detail view on the given drawing.

As a result of its use, the caption for the detail view is created on the drawing. The image of the detail view itself is automatically clipped by selected contour (circle, oval or rectangle) and augmented by the view caption. The obtained image can be edited later by using the standard tools of the T-FLEX CAD.



When creating the detail view with the caption, the following will occur:

1. Creating the new drawing view (on the basis of a new page of the type "Auxiliary"). The values of the parameters of the drawing view are set "by default";

2. Creating a copy with translation for associated copying of the selected elements onto the drawing view page with the specified scale and rotation angle. Clipping the image to be copied by the automatically created hatch of the specified form;
3. Creating detailing elements for the detail view (view caption, caption of the detail view).

After invoking the option it is necessary to do the following:

1. Select the fixing node for the elements which will be copied;
2. Specify the center of the selection area for the detail view;
3. Specify the form and size of the selection area for the detail view;
4. Specify the location of the leader line jog for the detail view caption;
5. Specify the location of the detail view (that is, the location of the drawing view with the detail view).

Note that in the described sequence of steps, there is no step for selecting the original elements to be copied. When creating the detail view with the caption, the collection of the copied elements is determined by the view selection area – all 2D elements at least partially contained in the specified area will be copied.

For selecting the fixing node, in the auto menu of the command the following option will be available:

	<N>	Set relation with Node

The fixing node will determine the initial location of the fixing point for the created view. If, upon specifying the fixing node, the user selects an arbitrary point on the drawing (with the help of), then at this point the free node, which will be selected as the fixing node, will be automatically created.

After specifying the fixing node, the system will ask to specify the center of the selection area. This can be done by picking an arbitrary point in the drawing's window (with the help of) or by specifying exact coordinates in the command's properties window.

The center of the selection area can be also defined by selecting a 2D node or offset of the center point from the fixing node (selected at the previous step). For selecting the 2D node the following option is used:

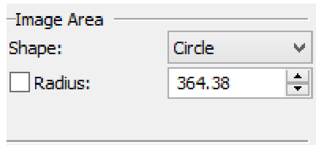
	<N>	Set relation with Node

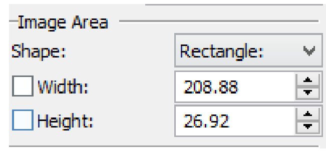
For specifying the offset of the center of the selection area from the fixing node, the following option is used:

	<A>	Set Relative Position

After selecting the center of the selection area, the form and dimensions of the area need to be specified.

The form of the area is specified in the command's properties window: *circle*, *rectangle*, *oval*.





The dimensions of the area can be specified either in the properties window or directly in the drawing's window (the dynamic image of the area's bounding frame will follow the cursor, pressing will fix the frame location).

After specifying the view selection area, the location of the leader line jog for the view caption has to be defined. Dynamic image of the leader line jog will follow the cursor. Pressing in the drawing's area will fix the jog location. The jog location can be also specified in the command's properties window (by specifying the coordinates of the starting point of the jog in the group “Basic parameters” of the properties window).

For changing orientation of the jog for the view caption, the following auto menu option can be used:

	<Z>	Change leader line jog orientation

After specifying the jog location, the dynamic image of the created detail view, attached to the cursor, will appear on the screen. The location of the view on the drawing has to be specified next.

> To make the caption of the detail view change its location according to the modifications of the drawing, it is recommended to snap the copied element to the 2D node. After snapping the view, its location and size can be adjusted with the help of the markers for editing the drawing view.

Complete the view creation with the help of the option .

For the constructed view, the caption will be automatically created. The obtained image can be edited and augmented, if required.

Creating Detail View with Breaks

On any detail view the user can create breaks, if necessary. The number of breaks on a single view is not limited. To create breaks, use the following option:

	<B>	Add or edit Broken View

After invoking this option, the system will transfer to the mode of creating/editing breaks on the current view. In the command properties window an additional section **Breaks** appears in which all breaks created on the given view and their parameters will be included.

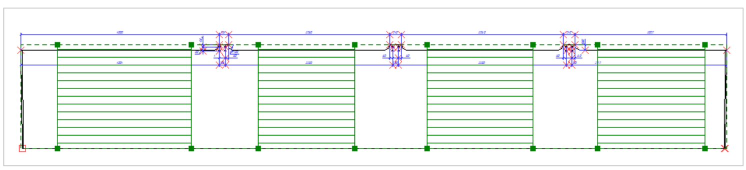
Note that if the breaks are not yet present on the current view, then after entering into this mode the new break will automatically be created. If the view already has the breaks, then the new break is not created and the system waits for selection of one of the existing breaks for editing its parameters. In this case it is possible to create the new break by pressing the button next to the list of breaks in the command's properties window. To remove the break, we use the button.

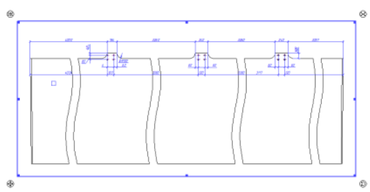
The following options will be available in the automenu:

	<Ctrl+Enter>	Finish break editing
	<C>	Select base point
	<A>	Select line defining break orientation
	<L>	Select lines defining label's boundaries
	<Esc>	Exit break creation/editing mode

By default, the system prompts us to create a horizontal break. This will be shown on the view as a hatched rectangular domain. The hatch lines show the direction of the space compression (perpendicular to the break lines).

The next figure shows an example of the view with the break.

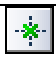




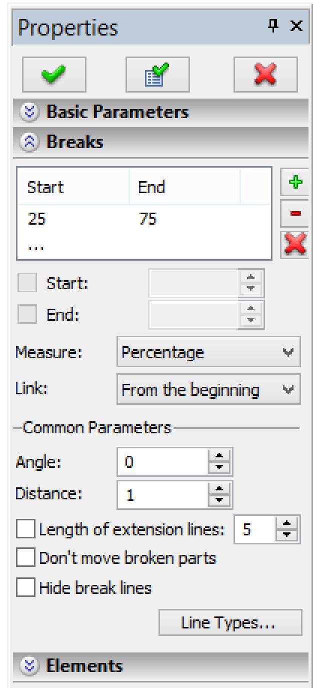
If several breaks are created on the view, then the current break (that is being created or edited at the present moment) is displayed with green color, and the remaining breaks are shown with grey color.

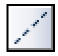
Location of the right and left boundaries of the break can be modified simply by selecting the boundary line with the help of and by moving it to the desired location. By pressing again we fix the new location of the boundary.

When moving the break's boundary, the additional option is available in the automenu:

	<N>	Select insertion node

With the help of this option, the break's boundary location can be snapped to a 2D node.



Break's orientation can be modified by specifying it with the construction line or with the angle of rotation from a horizontal line. To specify break's orientation with the help of construction line, it is required to invoke the  option and select the desired line on the view page. The domain of the break will be reconstructed parallel to the selected line. Orientation of the break with the help of an angle can be specified directly in the command's properties window (the **Angle** parameter).

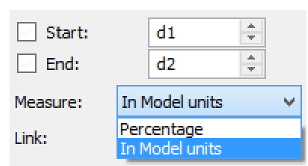
All breaks created on the view are added to the list of breaks in the properties window. For each break the location of boundary lines (**Start** and **End**) is indicated in the list. These coordinates are determined automatically when the user indicates the break zone on the drawing. They can also be specified with precise numeric values or variables (when the mode "In units of a model" is enabled, see below). To do so, it is required to select the desired break in the list and enter the required values for the following parameters:

Start. The value that determines the location of the left boundary – for horizontal break, or the lower boundary – for break at an arbitrary angle.

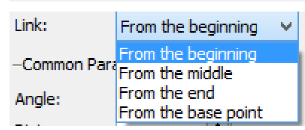
End. The value that determines the location of the right boundary – for horizontal break, or the upper boundary – for break at an arbitrary angle.

> If the values are specified as variables, then after the numeric value the corresponding name of the variable is indicated in square brackets.

Other parameters of a break specified in the properties window are common for all breaks of the same drawing view.

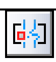


The **Measure** parameter determines the way of specifying locations of the break lines – either with numeric values in the units of the view page (**In units of a model**) or as a percentage along the direction of a break (**Percentage**).



Positions of breaks can be defined relative to the start, the middle, the end of the image, or from an arbitrary base point (by default it is selected by the system, the base point is marked with an arrow on the view's image).

We can select our own base point for specifying the break's boundary location with the use of the following option:

	<C>	Select base point

Boundaries of the lines of break's label are defined based on the marginal graphic lines, individually for each side of each break. When defining the boundaries, the formatting elements (dimensions, texts etc.), and also 2D fragments for which the special viewing mode without clipping was specified are not taken into account.

If necessary, the user can make automatically defined view's boundaries more precise. To do so, we can use the option:

	<L>	Select lines defining break's label

After invoking this option, it is necessary to indicate two lines of the drawing view. These lines can be both the graphic lines and construction lines (line, spline or function).

When creating/editing a break it is also possible to specify the following parameters in the properties window:

Distance between break lines determines the distance between the parts of the broken view.

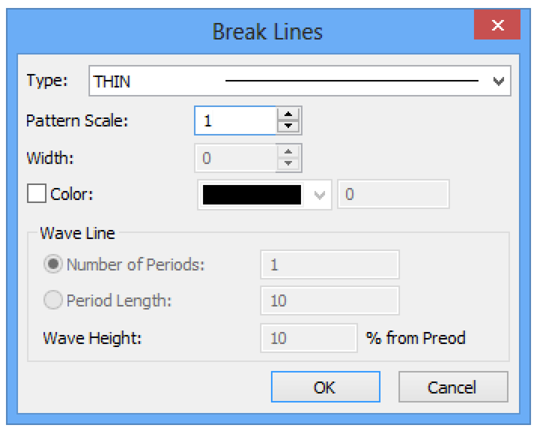
Angle. This parameter specifies the angle of rotation of break lines with respect to a horizontal line. Positive direction is in the counterclockwise direction.

Extension lines length. If this flag is absent, then only that part of the break line will appear on the view which directly intersects the image of the view. Otherwise, the break line will go beyond the view's margins by the magnitude indicated by this parameter.

Do not move breaks. When this flag is enabled, the view's lines that remain after applying the breaks do not change their location.

Hide break lines. With the help of this flag visibility of break lines can be controlled. When the flag is enabled the break lines are not displayed on the drawing view.

The break of a projection can be bounded by the lines that have various shapes. It can be a straight line, wavy line or zigzag line.

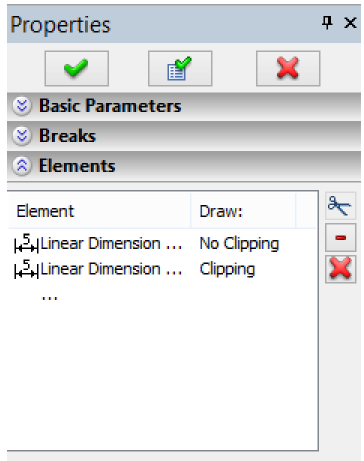


To select properties of the lines that bound the break, it is required to invoke additional dialog. To invoke it, press the **`[Line's Property...]`** button in the properties window.

Line's parameters such as the **type**, **stroke scale**, **thickness** and **color** define the style of break lines.

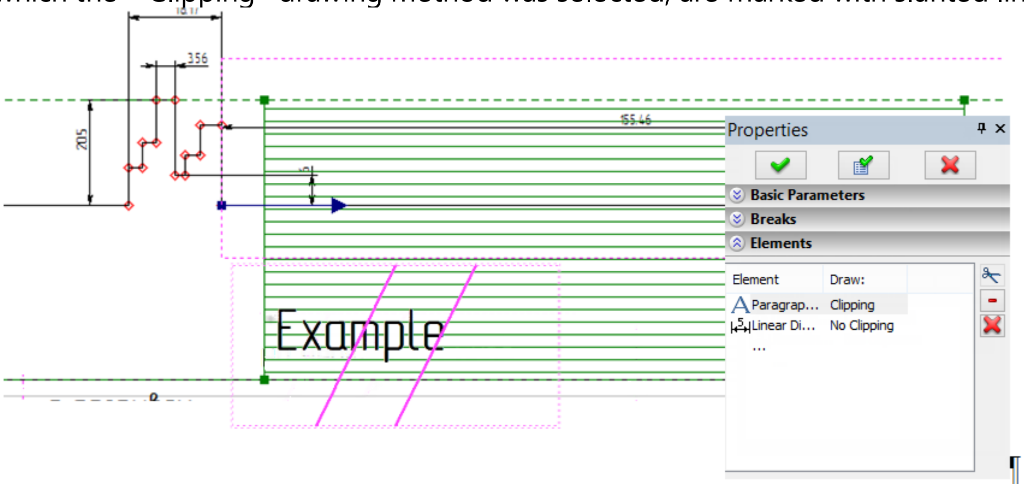
Properties of wavy or zigzag line. For a wavy or zigzag line it is possible to select parameters such as **the number of periods, period length** and **wave height**. Units correspond to the units of a model. For a zigzag line you need to select such value for the period that the break line has only one chip. The size of a chip is controlled by the amplitude.

For a view that contains breaks, a list of "special elements" can be created. It can include formatting elements and 2D fragments. By default the formatting elements and fragments are shown on the drawing view without clipping on top of the main view. This rule also holds for the views with breaks. If it is required that the formatting element or the fragment on the view be cut off at the boundaries of the break, it is required to add the element to the **Special elements** list in the command's properties window and select the **With clipping** mode for it.



To edit the list of special elements it is required to press the button to the right of the list. To add the element to the list, select it on the drawing with the help of . By default for all selected "special" elements, the "without clipping" mode is set. To change the clipping mode, it is sufficient to select the required element in the list – the mode will change (cyclically) whenever we click .

On the view the elements included into the list are marked with a frame, and furthermore, the elements for which the « Clipping» drawing method was selected, are marked with slanted lines.

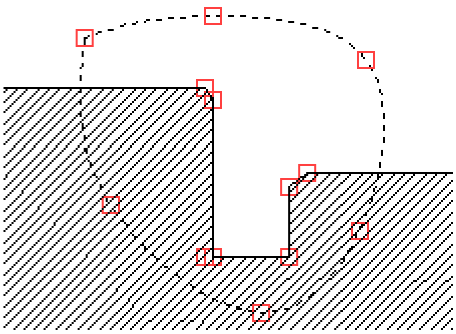


To remove an element from the "Special elements" list, it is required to select it in the list with the help of and press the button. It is also possible to indicate the required element in the drawing's window by pressing <Ctrl>+ . You can remove all "special elements" with the help of the button.

After all parameters of the break are specified, it is required to complete its creation by pressing

Creating Detail View by Contour

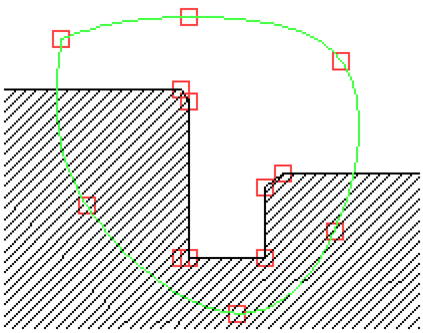
To create a drawing view by contour you should create a contour in advance. Both the construction lines and graphic lines can be used for the contour creation.



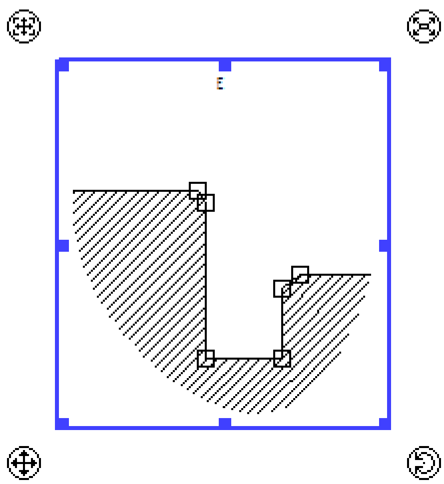
Use the following option to create a drawing view:

	<S>	Create detail view by contour

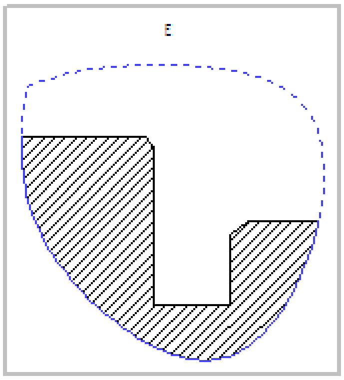
You should select a contour after the option activation.



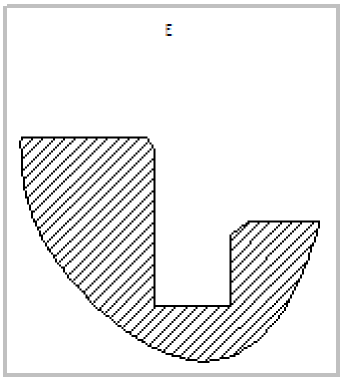
Then you should select a fixing point and specify the position of detail view on the drawing.



Then you should edit outline.



To finish a detail view creation select the option .

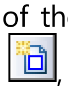
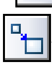


Particulars of Scaling Drawing Views

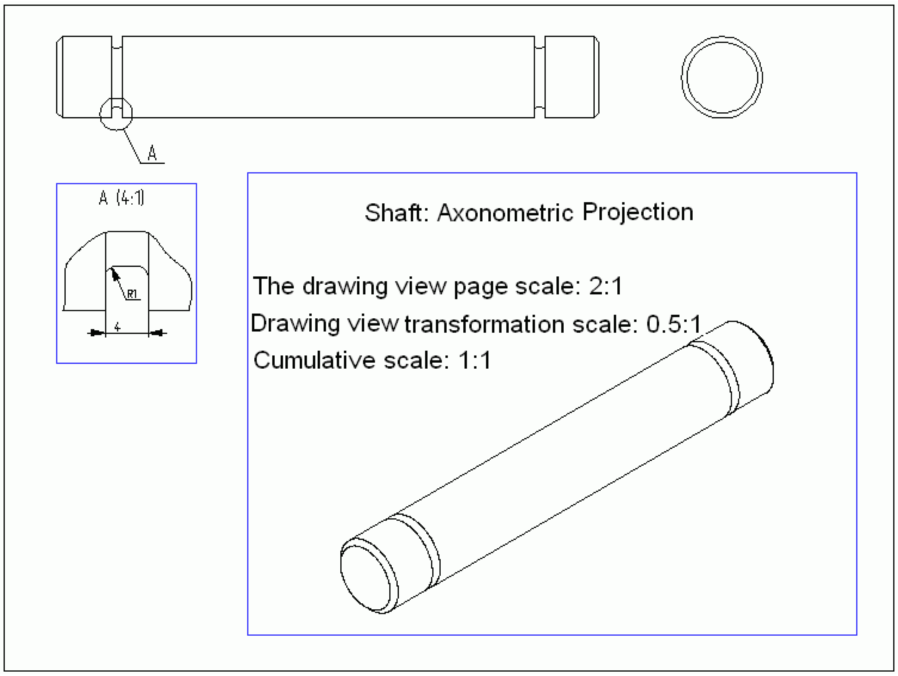
The "Drawing view" element and based on it detail views are created using various T-FLEX CAD instruments (the view page, the drawing view mechanism, the copy functionality). Each of these instruments has its own settings, including the scale:

1. Drawing view page scale (defined under the command **ST: Set Document Parameters**);
2. Drawing view scale which is the scaling factor of the drawing view page display on the detail view (defined in the drawing view properties dialog box – "Image|Scale");
3. The Scale of the copying the view elements (available in the properties window of the drawing view when using the options  and ).

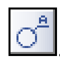
All these scales are superimposed in the image of the drawing view on the host drawing page.

When creating a drawing view using the options of the command **SD: Create Drawing View**, the user defines only two of these scales (for example, the scale of the drawing view and the page scale when using the option , or the scale of the view image and the copying scale when working with the options  and ). The missing scale is assigned to default value. The user can manually modify the values of all types of scale (see next section), however in this case the user needs to clearly understand the mechanism of the scale array.

For example, consider the drawing view of a shaft on a diagram in the beginning of this chapter. This drawing view of the shaft in axonometric projection was performed with the page scale equal to 1 and the drawing view scale equal to 0.5 (1:2). As a result, the image of the view on the main drawing page has the cumulative scale equal to 0.5. By changing the drawing view page scale to 2 (2:1), we will get the axonometric projection of the shaft on the main page to the 1:1 scale.



Should we change the view page scale to 4 (4:1), the cumulative scale becomes equal to 2 (2:1).

The detail view (View A) in this example was created using the option . The copying scale and, therefore, the drawing view page scale is equal to 4 (4:1). Meanwhile, the drawing view scale is equal to 1. As a result, the cumulative scale of the detail view image is equal to 4.

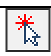
Editing Drawing Views

Editing a Drawing View

Editing of a drawing view is done by the command **SD: Edit Drawing View** The command is called as:

Keyboard	Textual Menu	Icon
<ESD>	Edit > Draw > Drawing View	

Upon calling the command, an option appears:

	<R>	Select element from list
	<*>	Select All Elements
	<Esc>	Exit command

Once all drawing views are selected by the option , they are highlighted, and additional options appear in the automenu:

	<Del>	Delete selected Element(s)
	<Esc>	Cancel selection

Upon selecting a particular drawing view by clicking it , another options appears in the automenu:

		Fixing point of view
	<P>	Set entity parameters
	<I>	Select another nearest element
	<B>	Add or edit breaks

The option allows the user to modify the fixing point of the drawing view being edited.

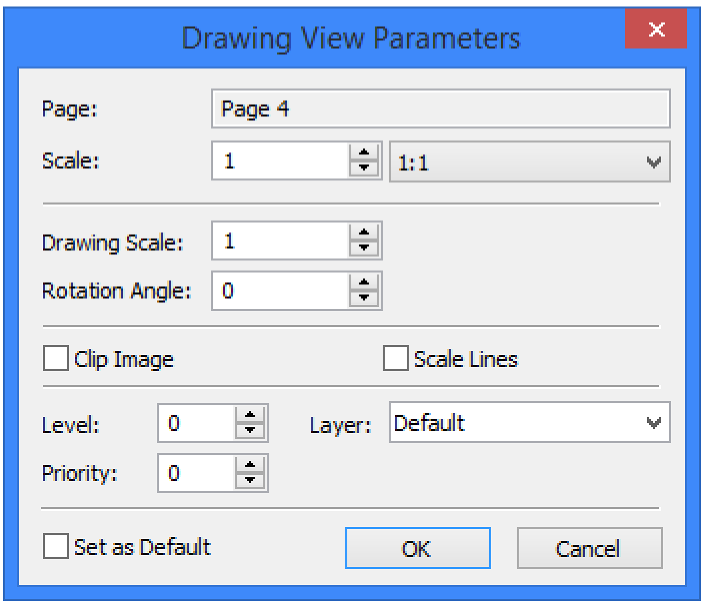
The option brings up the “Drawing View Parameters” dialog box:

Page. Displays the name of the auxiliary page corresponding to the selected drawing view. The parameter **Scale** defines the scale factor of this page.

Clip Image, **Scale Lines**, **Drawing Scale, Rotation Angle**. These parameters affect the display of the drawing view area on the original page.

Layer, level, Priority. Define the values of the respective system-wide parameters.

The selected view is highlighted as a box with markers along the perimeter. The markers control position, rotation angle, scale, view size and the position of the fixing vector or fixing node.



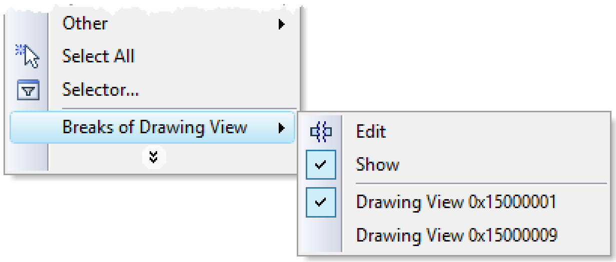
Working with the markers is carried out in the same way as when creating the drawing view (or the drawing view of the element).

The following option completes drawing view editing:

	<End>	Finish Page View editing

The option enables the mode of creating/editing breaks on the view.

The command of editing the breaks of the drawing view can also be invoked without the use of the view edit command – from the context menu of the page displayed by the given view. In this context menu it is possible to specify if the diagram of breaks must be shown on the page (**Show**), and also to select the drawing view whose breaks have to be shown and edited (if there are several views displaying the given page).



Editing the Image on a Drawing View

The image on a drawing view can be edited in two modes:

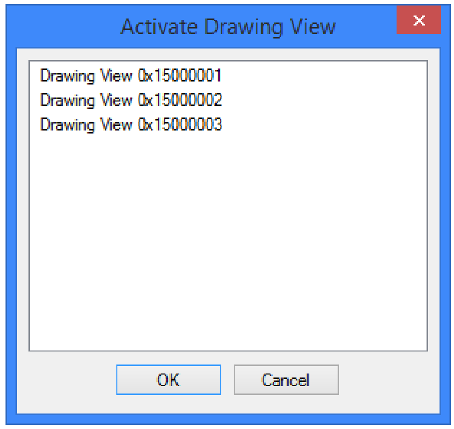
1. On the drawing view page – as a regular drawing;
2. On the main page, by activating the appropriate drawing view. The view can be activated “in place” or in a separate window. Drawing view activation is done via the context menu after selecting the drawing view, or via commands in the menu **View** > **Drawing view**.

Upon selecting one of the drawing views by clicking on the drawing page, the following commands become available in the context menu:

Activate Drawing View . Upon calling the command, all elements on the current page, except for the elements of the drawing view, are grayed out. The boundaries of the drawing view and the drawing view page become highlighted. The graphics of the drawing view can then be edited in a usual way.

Activate in new window . This mode differs from the previous in that a new window is opened for editing that contains the graphics of the selected drawing view.

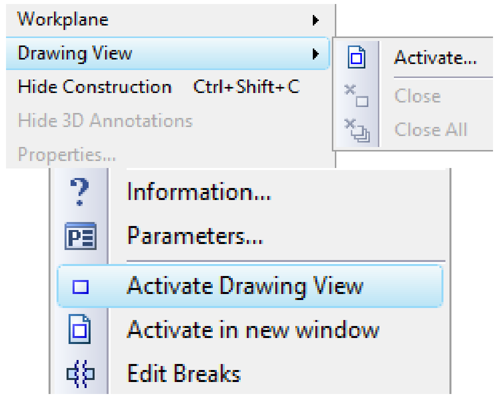
Drawing View Parameters . Brings on the screen the dialog box of the command **ST: Set Document Parameters** with the selected drawing view page settings.



When using the menu **View** > **Drawing view** first time, only the command **Activate...** is accessible. Upon calling this command, the drawing view selection window is displayed. The selected from the list view will be activated in the current page.

To return to the mode of editing the main page graphics, the active drawing view must be closed by using the command **Close Drawing View** in the context menu or the command **Close** in the **View** > **Drawing view** menu.

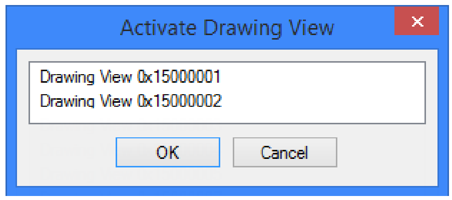
When using these commands for handling nested views, that is the views that appear within other drawing views, activate the views one by one, starting with the "outmost" one.



Meanwhile, the command name **Activate Drawing View** in the context menu is appended with the nesting level of the next drawing view.

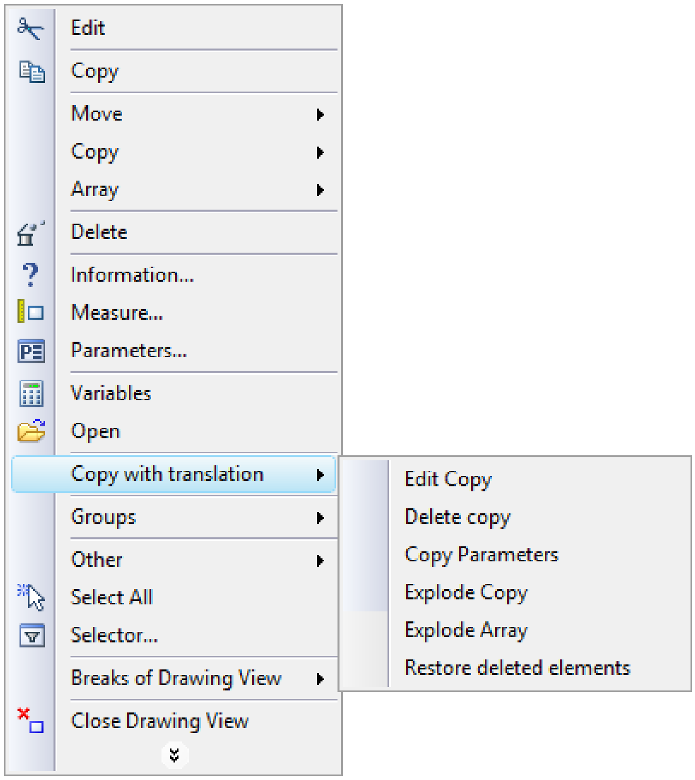
All active drawing views can also be closed by using the command **Close All Drawing Views**, available in the context menu. Besides, the activated nested views can be subsequently closed by using the command **Close Drawing View** for each active drawing view.

Calling the command **View** > **Drawing view** > **Activate...** from an active nested drawing view brings up a window with a list of accessible drawing views that includes <Main View>. Selecting the latter acts in the same way as the call of the command **View** > **Drawing view** > **Close All**, closing all levels of activated nested drawing views.



When editing graphics in the drawing views created via the options "Create New Drawing View" and "Create New Drawing View with Caption", all commands for editing copy with translation are available in the activated drawing view mode.

A context menu appears on right clicking over the lines on the view created by copying (in the activated drawing view area on the main page or on the view page), with the following commands under the **Move Copy** group:



Edit Copy - calls the editing command **EY: Edit Copy;**

Delete copy – deletes all elements of the copy;

Copy Parameters – allows defining the scale and rotation angle of the translation;

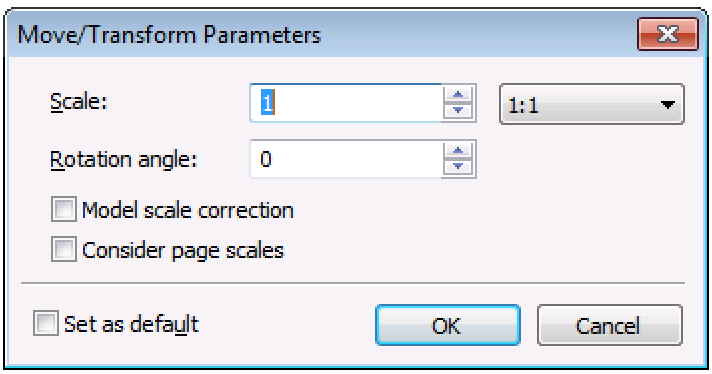
Explode Copy - breaks association of the moved elements with the original elements. The copied elements become independent objects, regardless of their parents creation history;

Explode Array - breaks association of the moved elements with the original elements; however, relations are preserved between the objects of the copy analogous to the relations between the original objects;

Restore deleted elements – allows restoring deleted elements of the copy.

- "Parameters" - opens dialog of transformation parameters where you can change scale and angle of rotation.

Two extra options appear at this stage:



Model scale correction (enabled by default). This option is available only for "move" transformations automatically created for the detail view. When this option is enabled, the assigned scale value is replaced by the calculated scale as follows:

$$\textit{The resulting scale} = \frac{\textit{Assigned scale}}{\textit{Scale of original page}}$$

Thus, the scale of original page is considered in the movement result. This allows you to obtain correct image of detail view when scaling is used on the original page.

Consider page scales. This flag can be used for the drawing views created with the option , when scale of the original page and view page differ from 1:1 and do not match. When this flag is enabled the resulting view scale will take into account the scale of both pages:

$$\textit{Resulting scale} = \frac{\textit{Assigned scale* Scale of drawing view page}}{\textit{Scale of original page}}$$

Setting this parameter allows you to achieve the correct correspondence of element sizes between the original drawing page and drawing view.

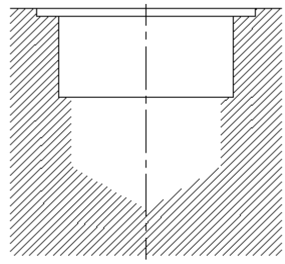
Adding to and modifying the image of drawing view

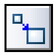
The image created by the and options may need modifications or additions, for example, add chamfers, rounds, clipping line, or place dimensions). One shall keep in mind that these options create graphics based on the **XM: Copy with translation** functionality, therefore, it is subject to all rules and limitations imposed on direct use of this command. Thus, only graphic lines, hatches, 2D fragments and projections are copied. The snapping nodes of the copied lines can't be modified. The copied elements can only be deleted, hidden by the command **ESO: Hide/Show Elements**, or their parameters modified (for example, lower the level). Besides, these can be used for snapping the construction lines (see the chapter "Lines") for new element creation. Therefore, if editing a view involves modifying existing graphic lines, two ways of handling are possible:

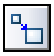
1. Break association of the copies with the original elements using the command **Explode Copy** of the context menu. As associations are broken, free nodes are created, thus turning the graphics on the view into a sketch suitable for modifications. One should keep in mind that further modifications of the original elements of the drawing will not propagate on the drawing view graphics, thus breaking its parametric behavior.
2. Create new graphic lines based on the copies. To do this, first create construction entities snapped to the copy entities. Then, hide the entities of the copy if necessary. Instead of the hidden lines, create new ones by snapping to the newly created construction entities.

Clipping image by an arbitrary hatch

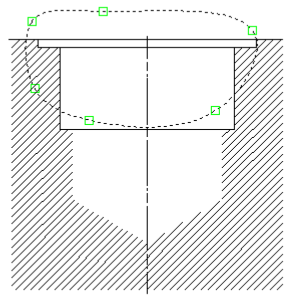
Original image of the detail view



The image created by the option  can later be clipped by an arbitrary hatch, using the context menu command **Edit Copy**.

In the following example, a new (detail) drawing view is created using the  option. Let's clip it by a hatch and an additional clipping line.

Spline defining clipping hatch and clipping line



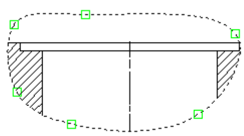
To edit the drawing view, let's activate its page. Both the hatch for clipping the view and the clipping line shall preferably be defined based on the same spline.

Let's create an invisible hatch based on the spline, using the command **H: Create Hatch**. Then, right click  over the copied lines and in the popping up context menu select the item **Move Copy** for calling the editing command **EY: Edit Copy**.

The coming up automenu will contain an option  - "Select Clipping Hatch". Pick this option, and then select the hatch. The image of the view will be clipped by the hatch as shown on the diagram.

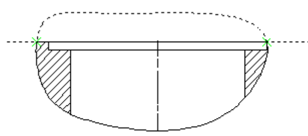
Note that the hatch can't be based on the detail view elements, as their use will create a recursive dependency.

The original image is now clipped by the hatch created in the previous step

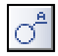


The final step is creating the necessary construction lines and creation of the clipping line based on the spline.

The original image is bounded by a thin line. The line ends are defined by the intersection points between the spline and an additionally created construction line



Updating detail view with label

In the context menu of detail views created with the help of the  option (detail view with label), additional command **Update drawing view** is available. It allows us to update the structure of the drawing view after the original image has been edited.

The selection of elements is carried out in the domain of the label's note, with the use of the current filter of elements. Only those elements which were created upon creation of the view or the previous update of the view are removed from the displayed page. The elements added to the displayed page after creation of the view are retained.

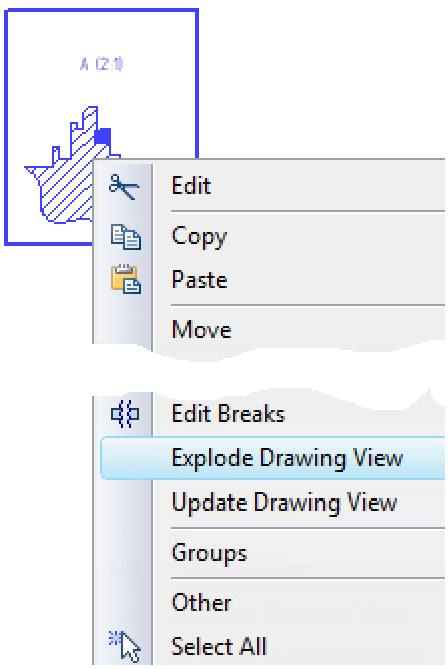
Exploding Drawing View

The command of exploding the drawing view can be found in the context menu of the view in the command wait mode.

As a result of exploding, the object "drawing view" is removed. In the same space, a group of standard elements of 2D drawing (nodes, graphic lines, etc.) resembling the image of the given view is created. If the drawing view image contains dimensions, then after exploding the view, the scale of the dimensions is modified in such a way that dimension's value remains unchanged.

When exploding the drawing view, whose contents is clipped by hatch (for example, when the drawing view was created as "Detailed Drawing View with Caption"), the following rules apply:

lines clipped by hatch (for example, lines of associate copy) are replaced by standard graphic lines shortened according to the clippings on the view;



- fragments clipped by hatch are copied into the resulting set of elements together with the clipping hatch;
- hatches, clipped on the original view, disappear after exploding the view.

Upon exploding, the page with the view can be removed or retained in the document. A query about the necessity of removal is sent.

PICTURES

Pictures

You can add to a T-FLEX drawing vector or bitmap images stored in files using different graphic formats – grb (T-FLEX CAD drawing), bmf (T-FLEX Metafile), bmp (Windows Bitmap), wmf (Windows Metafile), emf (Enhanced Windows Metafile), JPEG (Joint Photographic Experts Group), GIF (Graphic Interchange Format), DIB (Device-independent bitmap), and images in the TIFF, PCX and TGA formats.

> T-FLEX CAD Metafiles are graphic images of T-FLEX drawings, they do not require parametric regeneration and are quickly displayable on the screen. Those are stored on disk with the extension ".BMF". Metafiles can be generated in the command **EX: Export** or produced by an animation (the command **AN: Animate Model**).

Such images can be inserted using various scale, rotated various angles and fixed to nodes, which helps parametrically define their position on the drawing. Unlike fragments, pictures themselves are not parametric, do not use variables or special fixing elements.

Transparent font color can be set for bitmap pictures.

Connection with Source File. Links Mechanism

Pictures, just like fragments and databases, use the links mechanism that allows managing relations of objects with their source files. A picture is related with an object by a "link" that indicates the source of external data.

Recall that the object of a link can be external (external file) or internal (a picture copied from internal file and saved inside the main document, fragment document etc.). Links management is done by the command **AL: Links to other files…**.

> Links to files are used for 2D and 3D fragments and pictures. The command **File** > **Assembly** > **Update links** updates all links to files.

By default an inserted picture uses an "external link" to the source file. By changing the link to embedded (in the command **AL: Links to other files…**), you can save the picture directly in the drawing file (without relation to the source file).

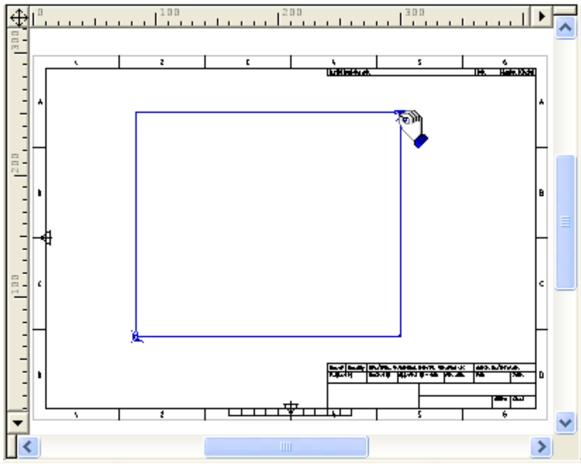
Internal document-picture can be generated automatically right after inserting the picture. In this case a created picture will be at once connected with two links: a link to an external document (source file) and to an internal document. Such picture will function as an internal one, keeping the image of the source document in the form that the image had at the moment of insertion, but updating an internal image from the source file in case of need will still be possible.

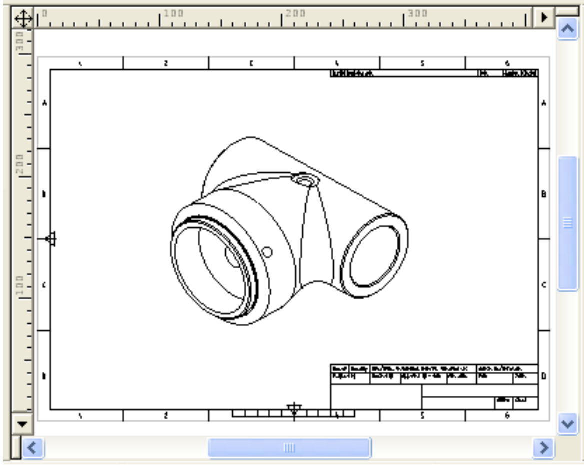
Pictures on the basis of files "*.grb" are always generated as internal ones with the preserved connection with the source file.

Methods of Fixing on Drawings

The position of a picture on the drawing can be defined by the following methods:

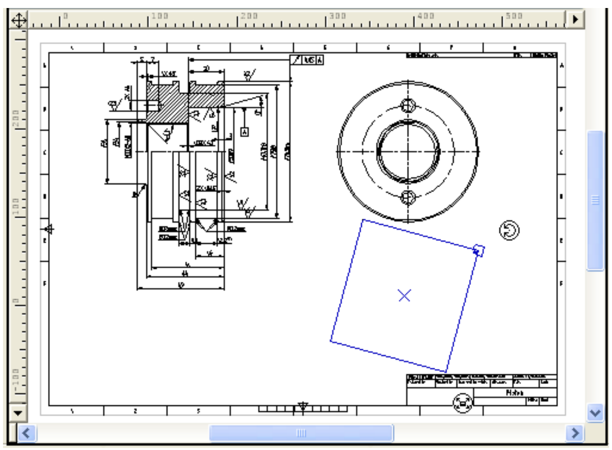
By two corner fixing points. The graphic image of a picture is bounded by a box. By defining two opposite corners of this box on the drawing, you can change the position and size of the picture's image on a drawing. The original aspect ratio of the picture is not preserved in this way. The picture is fit in the specified box.

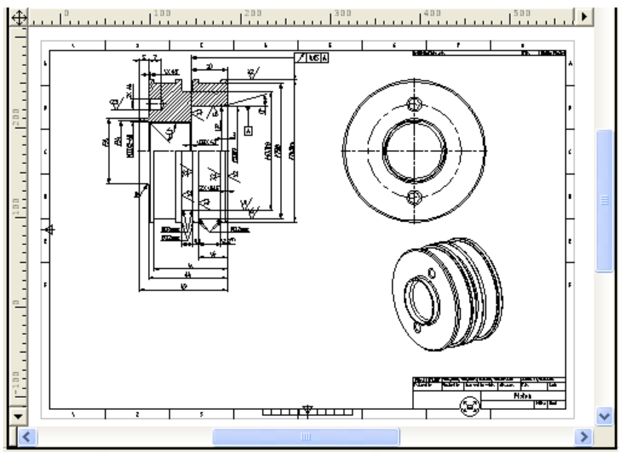




The corner points of the picture can be fixed to nodes on the drawing. This will allow parametric control over the position and size of the picture's image. Additionally, one can specify an angle of the picture's rotation about its center.

By one point, rotation angle and scale. In this case, just one reference point is used for fixing. As in the previous method, the fixing point can be related to a node on a drawing.





Creating Picture

To insert a picture in a drawing, use the command **IP: Insert Picture**:

Icon	Ribbon
	Draw → Title Block → Picture
Keyboard	**Textual Menu**
<IP>	Construct > Picture

After calling this command, the options for creating the picture will appear in the automenu, and the command's properties dialog box — in the properties window.

Choosing source file and fixing method

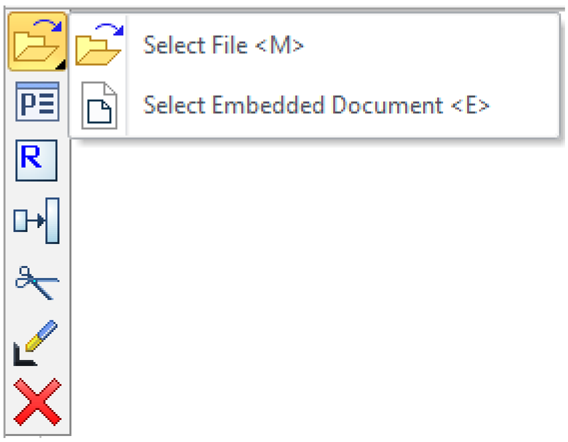
First of all, it is necessary to choose the source file of the picture being created. This can be done with the help of the button in the command's properties window or with the help of automenu option:

	<M>	Select File

Files can be selected either from the T-FLEX CAD library or just from any folder on the disk.

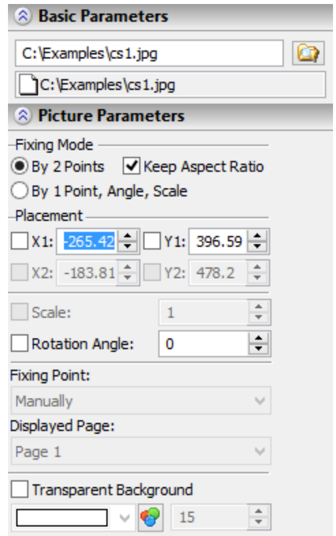
> A picture name can be variable. For this, you shall use a text variable as the picture name.

If the current document (in which the picture is generated) contains internal pictures, the symbol manifesting the existence of the drop down list will appear next to the icon of the option ***<M>***. Another method for selection of the picture source can be picked from the drop down list:



	<E>	Select Embedded Document

In this case the generated picture will not have the external source file.



The name of the selected/internal document will be shown in the command's properties window (to the left of the button).

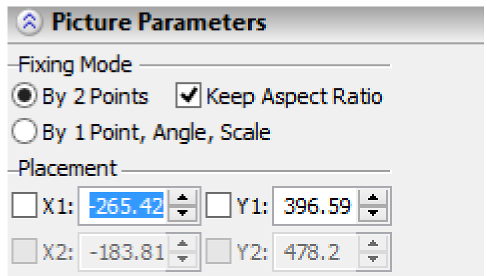
If the picture is inserted from a library, then the name of the library is entered in the corner brackets, for example, "<Schemes>Graph node". If the library name in the corner brackets is omitted, the picture is taken from the same library as that in which the drawing is located.

Grey (not accessible for editing) field, located a little below is informational and shows the absolute path to the file from which the picture is taken.

Upon picture insertion, as a file with bitmap image, a page being inserted from the source document can be selected in the command's properties window (parameter "Displayed Page").

The fixing method for the inserted picture (by two points or by one point, angle and a scale) is selected in the dialog box of the properties window (parameter "Fixing Mode").

Fixing picture by two points



After choosing the fixing method (parameter **Fixing Mode > 2 Points** in the command's dialog box) it is necessary to successively indicate location of two fixing points of the generated picture. This can be done with the help of in the drawing's window or by indicating the exact values of the coordinates in the command's properties window.

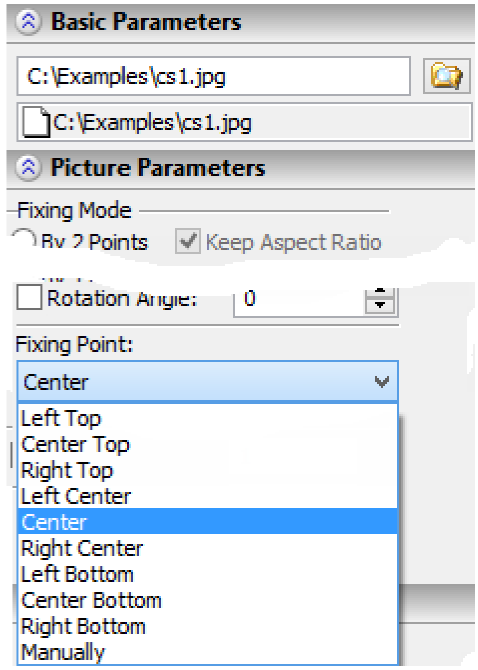
Upon defining fixing points, it is possible to select nodes of the drawing with the help of the option:

	<N>	Set relation with Node

When the second fix point has been defined, the creation of the picture is automatically completed.

After defining location of the first fixing point, it is possible to finish creating the picture by pressing (in the automenu or in the properties window). In this case, location of the second fixing point will be determined by the system automatically (on the basis of the initial size of the inserted picture).

Fixing picture by point, angle and scale



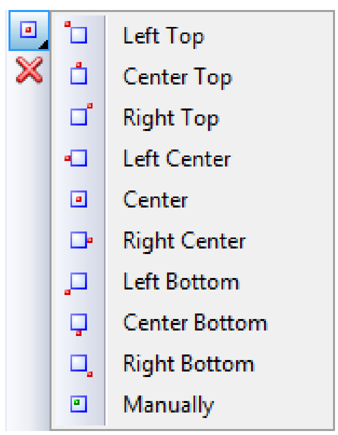
Upon inserting picture by one point, angle and scale, the fixing point is defined first. Note that in the command's properties window location of the fixing point can be selected with respect to the box of the picture (parameter **Fixing Point**).

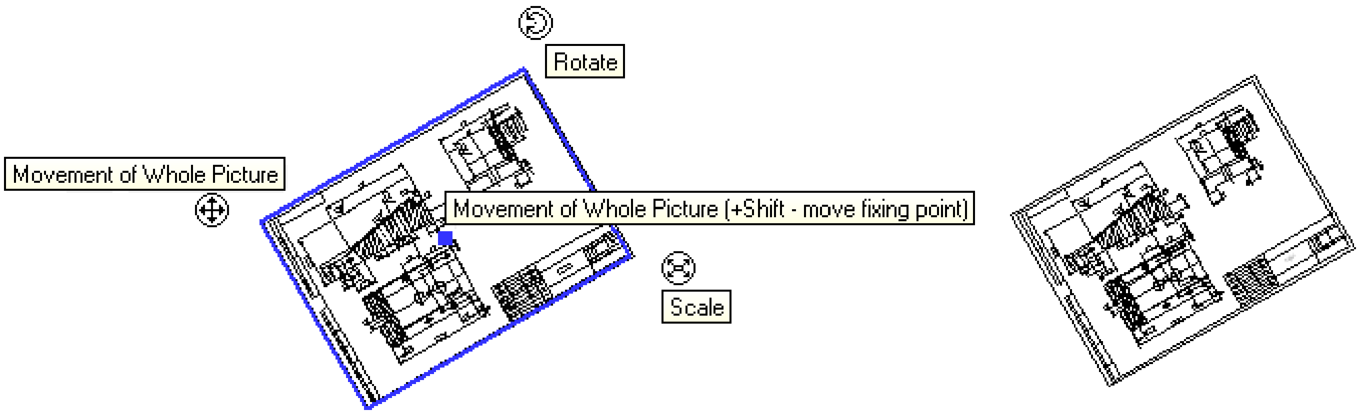
Location of the fixing point can be assigned directly in the drawing's window with the help of or by prescribing exact values of the coordinates in the command's dialog box. With the help of the option it is possible to fix picture to a node.

"Fixing point" indicates which one of the characteristic points of the picture rectangle will be combined with the given fixing point. Selection of another characteristic point from the drop-down list of this parameter allows you to change the final location of the image on the drawing. The same can be done with automenu option:

		Fixing point

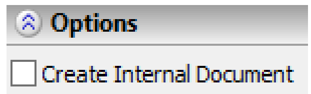
After indicating the fixing point, the picture's contour takes the position in accordance with the parameters (angle of rotation and a scale), prescribed in the command's parameters dialog box (be default, angle of rotation is set to be equal to 0, and scale is 1). At this point it is possible to finish creation the picture by pressing , but it is also possible to preliminarily change the scale, angle and even location of the picture with the help of special markers, located at the corners of the picture's box. Note that upon fixing by a point, angle and scale, rendering the picture can be completed only by pressing .





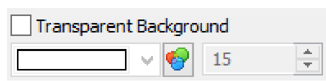
Scale and angle of rotation of the picture can be also prescribed in the dialog box of the properties window (parameters **Scale** and **Angle of rotation**).

Creating Internal Picture with Preserved Connection with Source File



To create an internal picture, right upon its insertion it is necessary to turn on the flag **Create Internal Document** under the group **Options** found in the command's properties window.

Setting Transparent Color



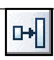
Prescribing transparent color is possible only for bitmap pictures. Transparent color can be set in two ways: with the help of a special list of colors in the command's properties dialog box (this list becomes available only upon activating the flag **Transparent Background**) or directly in the drawing's window with the help of the following option of the automenu:

	<T>	Select Transparent Background Color

This option lets a user select color directly from the screen by pointing at the desired region of the drawing.

Repeated Picture Insertion

For repeated picture rendering it is convenient to use the following options of the automenu:

	<R>	Repeat Last Picture Insertion
	<F>	Select Picture to Insert Copy

The option  enables to repeat multiple insertion of the last picture created in the given document. By default, parameters of the new picture will be assigned the same values as those of the last inserted picture.

The option enables to repeat insertion of any of the previously inserted pictures many times. After calling this option it is necessary to point at the source picture with the help of .

Editing Pictures

A picture's position and size can be modified, or the picture deleted altogether, in the command **EP: Edit Picture**:

Keyboard	Textual Menu	Icon
<EP>	Edit > Draw > Picture	

Upon calling the command, the following options become available in the automenu:

	<R>	Select element from list
	<*>	Select all elements
	<Esc>	Exit command

To select a picture, point to its image and click . Selection of several pictures can be carried out via a window or with the help of the key combination *left* <Shift>+ (adding elements to a list of chosen ones) or left <Ctrl>+ (removing an element from the list). After selecting one or more pictures, the following icons become available:

	<P>	Set Picture Parameters
	<Del>	Delete selected element(s)
	<I>	Select other element
	<Esc>	Cancel selection

Editing Single Picture

If a single picture is selected, it becomes outlined. The appropriate characteristic points and markers will be highlighted with the outline, depending on the fixing mode.

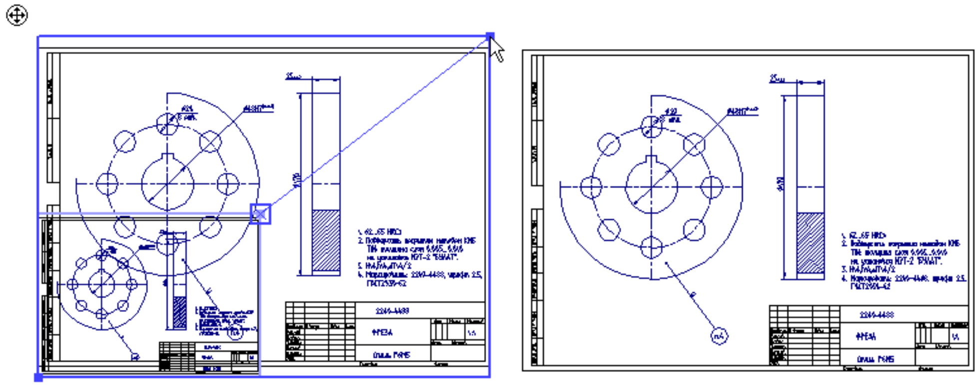
Editing Picture Inserted by Two Fixing Points

If the picture was put on the drawing using two fixing points, then the points defining the picture position on the drawing will be highlighted. By modifying the fixing point positions, you can change the picture location and size. Moreover, at the upper left corner of the picture there will be a picture movement marker, with the help of which a movement of a whole picture can be performed.

To modify the picture size, move the mouse to the desired fixing point and click . The picture's outline will start rubberband. One of the corners of the outline will be "tied" to the pointer, while the other stay still. Next, move the pointer to the desired position and click . After that, the picture will redraw per its new sizes. If the selected fixing point was fixed to a node, this fixing will be broken after the move.

Upon movement of the fixing point, the following option will be available in the automenu:

	<N>	Set Relation with Node

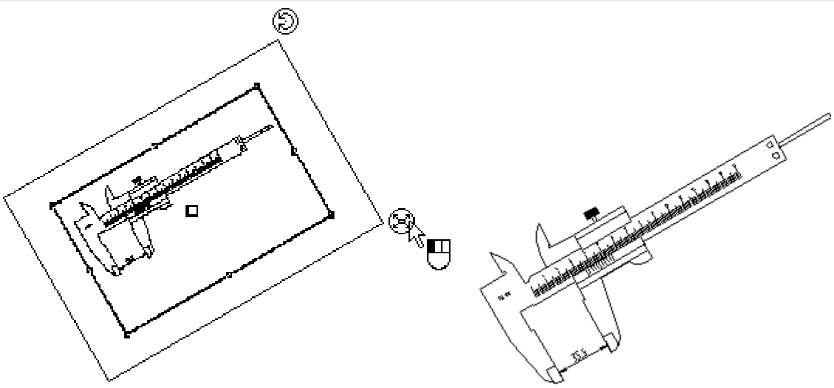


To move the picture with the help of the movement marker, it is necessary to bring the cursor to the pointer, press , drag the cursor to the desired location and press again. If the fixing points were initially fixed to nodes, then, upon moving the picture, their relations with those nodes will be broken.
The picture's rotation angle can also be defined in the two-fixing-points mode in the properties window.

Editing Picture Inserted by One Fixing Point, Angle and Scale

In case when selected picture was put on the drawing with the help of **one fixing point, angle and scale**, the fixing point as well as the markers for setting picture's movement, scale and angle of rotation will be highlighted. The markers enable to change position, angle of rotation and size of the image directly on the drawing.
The location of the fixing point will be determined by the value of the parameter **Fixing Point** in the command's properties window. If this parameter is modified, another fixing point will be highlighted on the drawing (corresponding to the selected value of the parameter).

> Detailed description of working with markers was provided in the chapter "Drawing view. Detail Views".



The exact numerical values of the rotation angle and picture scale can be defined in the picture's parameters dialog box (the option). The same dialog allows modifying the picture's fixing method.

Editing Several Pictures

If multiple pictures are selected, you can modify their parameters (level, layer, priority). You can also delete the selected pictures from the drawing.

Updating Internal Pictures

If upon creating a picture in the command's dialog box the flag “Create Internal Document” was turned on, then such picture will be connected with two links: the first link will refer to the external source file of the picture, the second one – to, created upon insertion, the internal (i.e. stored inside the current document) copy of the file. Upon recalculations, an internal copy of the source file will be used for updating the picture. This means that picture will preserve the form which the source document had at the moment of creating a picture.

For updating the picture from the external file, it is necessary to select a desired picture in the command waiting mode and call the context menu with the help of . For the internal picture, which maintained connection with the external source file, the command **Update File Data** will be accessible from the context menu.

Groups of Elements

Group – is a named logical union of several elements created by arbitrary rules. Grouping allows us to work with heterogeneous elements as with a unique object: select, move/copy as a whole object, specify common properties, etc. Grouping allows us to arrange the structure of a drawing (a 3D model) and avoid undesired editing of its individual elements.

Both 2D and 3D elements can form the groups..

General Principles of Work with Groups

Groups of elements in the documents of T-FLEX CAD can be created in two ways:

- Manually by the user with the help of the **"EU: Edit groups of elements"** command;
- Automatically by the system when various actions are performed, for example, when 2D copies, 2D fragments, and drawing views are unfolded, 2D projections are broken. This method of creation of groups will be described in more detail in the "Automatic Creation of Groups" section.

Each group in the document has a unique name. Groups created by the user by default are assigned the names of the type "Group_0", "Group_1", etc. This name can be changed to a more informative one in the command of editing/creation of groups.

Any object of the drawing or of a 3D model can belong only to one group. Moreover, both 2D and 3D elements can be combined into a single group. The group itself can be included into another group, thus creating a hierarchy of groups of arbitrary complexity.

The system will work with objects combined into a group as with a whole. If the cursor of the mouse is moved across the screen to an element, which belongs to some group, the entire group will be highlighted on the screen. When selecting an element belonging to the group, all elements of the group will automatically be selected.

If necessary, any element of the group can be selected individually. To do so, use +with pressed key <Alt>. For the element of the group selected in this way, it is possible to change properties and even geometric parameters.

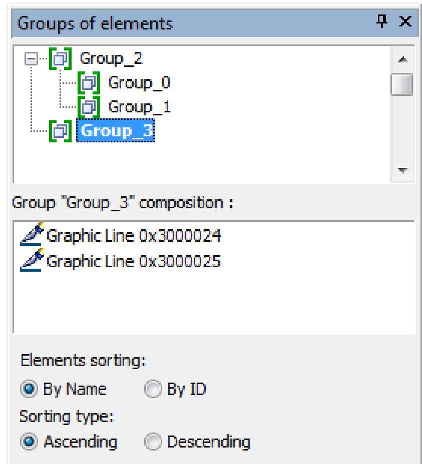
The structure of groups can be edited by adding new elements to them or by deleting already added elements. The elements removed from the structure of the group remain in the drawing or in the 3D model and the user can work with them as with independent objects.

Command for Working with Groups

For creation and deletion of groups, editing its structure, use the **"EU: Edit groups of elements"** command:

Icon	Ribbon
	Draw → Additional → Groups
Keyboard	**Textual Menu**
<EU>	Construct > Groups

The window of properties of this command contains two lists. The upper list – "Groups of elements" – shows the list-tree of groups created in the current document. In the root list of the tree, all main groups of the current document are located. The folders, which can be expanded, contain the embedded groups (groups included into other groups).



The lower list ("Structure of group…") displays the contents of the group selected in the upper list. A group can be selected by pressing on the group's name. The elements included into the selected group are also highlighted on the drawing or in the 3D scene (depending on what sort of windows are opened in the current document).

If in the current document none of the groups were created, both lists will be empty.

In the automenu of the command, the following options for invoking various modes are available:

	<G>	Group
	<U>	Explode group
	<Y>	Set name
	<E>	Edit&Copy
	<Esc>	Exit command

The option allows us to create a new group. The option – ungroup an already existing group, i.e., make all elements included into it independent objects. In this case the group itself will be deleted.

The option is used to specify a new name of the group.

The option allows us to modify the structure (i.e., add or delete elements) of the selected group.

Grouping (Creation of New Group)

After invoking the option, the command will enter the mode of creation of a new group. In the window of properties of the command, in the list of groups, the new group will appear under the standard name "Group_№". When entering the mode of editing, at the header of the list of elements the name of the new group will appear. The list of elements itself will remain empty.

> New group will always be created at the upper level of the structure of groups (at the root of the tree of groups).

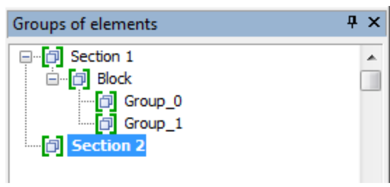
In the automenu of the command, the following options will become available:

	<Ctrl> <Enter>	Finish selection of elements
	<M>	Mode of addition of elements
	<M>	Mode of deletion of elements
	<Y>	Specify name
	<Esc>	Cancel editing

By default, in the automenu the option will be active, i.e., the mode of addition of elements to the group being created will be enabled. The elements can be selected by pointing at them in the window of the drawing or of the 3D model, and also in the tree of the 3D model (for 3D elements). The selected elements will be displayed in the list of elements of the group in the window of the command's properties.

> Note that when selecting the elements, the filters in the system toolbar are active.

To correct a wrong selection, the mode of deletion of elements can be used (the option). The elements which are specified in the given mode are excluded from the list of the selected elements. Only the elements that are already added to the group being edited can be selected. The state of the filters of the selector is also taken into consideration when making a selection.



If necessary, it is possible to change simultaneously the name of the group being created with the help of the option. After invoking this option, the command enters the mode of editing the text of the group's name. The new name of the group can be typed directly in the list of groups. The editing of the name can be completed by pressing outside the box with the edited text.

> The name of the group can also be modified outside the mode of creation/editing with the help of the analogous option in the main menu of the command.

Creation of a group can be completed with the help of the option (in the automenu or in the properties window). After pressing the command returns to the main mode of work.

Ungrouping and Editing a Group

For editing an already existing group, it is necessary to select the desired group in the list of groups and invoke the  option. The work in this mode is carried out exactly in the same way as in the mode of creation of a group.

Note that if in this mode all elements from the structure of the group are deleted, the group itself is not deleted. Later such an “empty” group can be edited again by adding to its structure the new elements. To remove the group, it is necessary to use the .

Working with Groups with the Help of Context Menu

All modes of operation of the **“EU: Edit groups of elements”** command are accessible directly from the context menu upon selection of elements outside the command.

Commands for working with groups are combined into the “Groups” submenu. Depending on the set of elements for which the context menu was invoked, it can contain the following commands:

- **Group…**. It launches the groups editing command in the mode of creation of a new group;
- **Ungroup**. It ungroups (deletes) the selected group/groups;

Edit groups… *(available upon selection of only one group)*. It launches the groups editing command in the mode of editing of the structure of the selected group

Automatic Creation of Groups

Automatic creation of groups is offered by the system in the following cases:

- when creating complex elements of the draft (polygons, rectangles …). Capability of creation of such a group is controlled with the corresponding flag in the window of properties of the given mode of draft creation;
- when creating a 2D array, if a group is selected as one of the initial objects of the array. In this case, by default, the new group is created that includes elements-copies of the array obtained on the basis of elements of the initial group. If two groups were selected for copying, two more groups will be created, etc.;
- when unfolding fragments, projections, drawing views, copies and arrays. In this case, the system generates an additional window-warning with the suggestion to combine the resulting elements into a group. The group will be created in case of positive answer of the user.

When unfolding a 2D array, the number of created groups depends on the command being used – “Explode” or “Explode Keeping Relations”. In the first case, one group is created that includes all elements belonging to the array. In the second case, elements of each copy of the array are combined into an individual group. This implies that the number of resulting groups is equal to the number of copies in the array.

If elements of the array/copy that is unfolded were already combined into groups (for example, upon creation of an array), these groups are removed and the new groups are created.

Drawing Editing

Moving and Copying Drawing Elements. Arrays. Use of Clipboard

T-FLEX CAD provides two main mechanisms for creating new 2D elements based on existing ones.

The first mechanism relies on moving, copying and array creation commands. This group of commands unites all functions related to translating, scaling, using symmetry and rotating arbitrary 2D elements, as well as creating various types of associative and independent copies, including linear and circular arrays. Moving/copying of elements can be done within the current page of a T-FLEX CAD document or from one page to another one.

The second mechanism uses copying via the clipboard that is a somewhat extended functionality in T-FLEX CAD as compared with the standard Windows clipboard management. This mechanism only supports non-associative (independent) copies. However, it supports copying between different documents within the same T-FLEX CAD environment, as well as exchange with other applications.

Any T-FLEX CAD 2D elements can be handled by either of the moving/copying mechanisms, except those specifically mentioned in the respective sections (for example, multi-page text and BOMs cannot be copied or moved).

Moving, Copying and Array Creation Commands

A family of commands is provided in T-FLEX CAD for moving, copying and array creation. These commands are used for transforming existing drawing elements and making various types of copies, including multiple (arrays). All commands have similar interface and underlying mechanism that allow easy switching from one command to another while keeping the same set of selected objects. These commands also interact with object snapping described in the chapter “Sketch. Creating a Non-parametric Drawing”. Upon calling any of the commands, the toolbar is displayed for object snapping management.

The commands are divided into three large groups by their purposes:

The commands for moving (for modifying the existing elements – translation **TM: Move**, rotation **TT: Move and Rotate,** scaling **TA: Move and Scale**, symmetry **TS: Move with symmetry**); **,** translation with rotation **TP: Move with translation and rotation**);

The commands for creating copies (copying with translation **XM: Create Copy**, copying with rotation **XT: Copy and Rotate**, with scaling **XA: Copy and Scale**, creating copies symmetrical to the original elements **XS: Create Symmetry**); and copying with translation and rotation **XE: Copy with translation and rotation**);

Array creation commands (linear **XL: Create Linear Array** and circular **XR: Create Circular Array**).

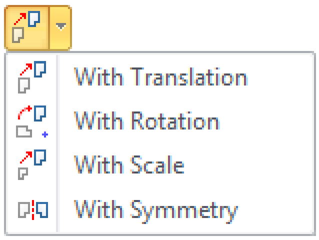
Since the interface of all the commands is identical, the further description will be as general as possible, covering all moving, copying and array creation commands.

Calling the Commands

The commands for moving elements are available under the textual menu entry **Edit**:

Keyboard	Textual Menu	Icon
<TM>	Edit > Move > with Translation	
<TT>	Edit > Move > with Rotation	
<TA>	Edit > Move > with Scale	
<TS>	Edit > Move > with Symmetry	
<TP>	Edit > Move > with Translation and Rotation	

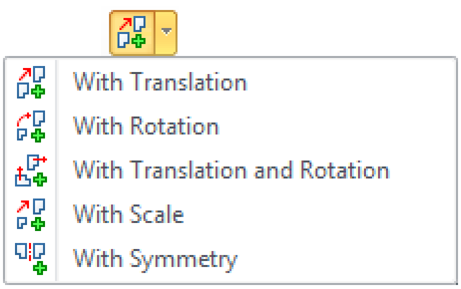
You can find the commands in **Draw** → **Additional** in the Ribbon.



The copying and array creation commands are grouped under the menu entry **Draw:**

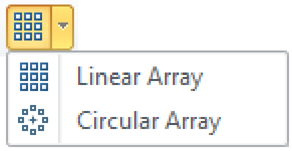
Keyboard	Textual Menu	Icon
<XM>	Draw > Copy > with Translation	
<XT>	Draw > Copy > with Rotation	
<XA>	Draw > Copy > with Scale	
<XS>	Draw > Copy > with Symmetry	
<XE>	Draw > Copy > with Translation and Rotation	

You can find the commands in **Draw** → **Additional** in the Ribbon.



Keyboard	Textual Menu	Icon
<XL>	Draw > Array > Linear Array	
<XR>	Draw > Array > Circular Array	

You can find the commands in **Draw → Additional** in the Ribbon.



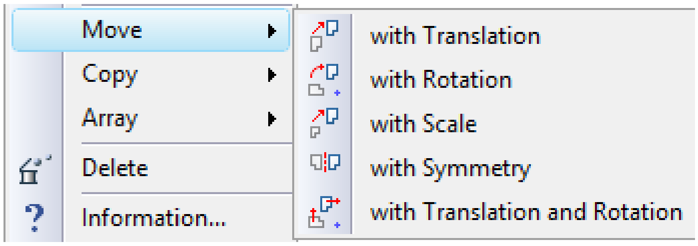
Upon calling any moving, copying or array creation command, the first step will always be selecting the objects for moving or copying. The following options will be provided in the automenu:

	<End>	Finish element selection
	<M>	Add elements
	<M>	Remove elements
	<I>	Select Other Element
	<Esc>	Exit from editing of selected Elements list

Any T-FLEX CAD 2D elements are available for selection. Elements can be selected by box and/or using under the active option . Selection can be canceled in the same way with the active option . To speed up the process, you can use the transparent mode of switching between adding and removing elements: pressing the <Ctrl> inverts the current mode. This means, if the current active option was , pressing <Ctrl>+ will be removing the elements from the selected set, and vice versa, with the active option pressing <Ctrl>+ will result in adding the picked elements to the set. To complete the selection, use the option .

After selecting the objects for moving/copying, the system will switch to the main automenu of the specific command.

The commands can also be called from the context menu when selecting one or several 2D drawing elements. In any case, the main automenu of the selected command will appear at once in this case. All the objects that were selected at the time of calling the context menu will be subject to the transformation.



The Common Options of the Moving, Copying and Array Creation Commands

For working convenience, all moving, copying and array creation commands have similar sets of automenu options.

Such options include the options for switching between the commands, the attachment point selection options and the options that define the action upon the completion of the current transformation.

Options for switching commands

The automenus of all moving, copying and array creation commands include the options, whose composition defines the current command. These options are used for quick switching between the commands while keeping the selected set of objects.

The moving, copying and array creation commands differ primarily in the type of the transformation and its mode. The transformation mode is either moving (modification of the selected elements) or copying (creation of new elements based on the selected). The type of transformation defines the kind of change between the original position and the target position. This can be a translation, rotation, scaling, translation with rotation, creation of a linear or circular array. Thus, for example, the combination of the "Move" mode and "Translate" transformation makes the command **TM: Move**, while the "Copy" mode in combination with the same transformation makes the command **XM: Create Copy with translation**. Note that the array creating transformations always make copies.

The kind of transformation is defined by the first automenu option that contains the enclosed list:

	<Ctrl+M>	with Translation
	<Ctrl+T>	with Rotation
	<Ctrl+Q>	with Scale
	<Ctrl+U>	with Symmetry
	<Ctrl+R>	with Translation and Rotation
	<Ctrl+L>	Linear Array
	<Ctrl+K>	Circular Array

The default automenu option corresponds to the current command.

The current transformation mode is defined by the following options:

	<C>	Move
	<R>	Copy

You can turn on only one option at the time: pressing one option automatically undoes the other one as in a radio group. The current mode is defined by the active option. Setting the option switches to one of the commands in the "Move" group, while setting the option - activates a command in the group "Copy" or "Array".

> If the "Linear Array" or "Circular Array" transformation is active, the options for selecting a transformation mode are not shown in the automenu.

Defining base points of transformation

When defining a transformation, you need to specify two special points - *origin* and *target*. These points define the transformation direction and parameter. Depending on the kind of transformation, either both or just one point is required.

Origin point is the point that marks the original position of the objects to be transformed. This would be the start point of a translation, the center of a rotation, the center of scaling or the start point of a linear array.

The origin point of a transformation can either be defined as an arbitrary point, or be selected as one of the characteristic points of the outlining rectangle. The outlining rectangle is a rectangular area that covers the extents of the set of objects selected for the transformation. Characteristic points of the outlining rectangle are its center, corners and the side midpoints.

The way of defining the origin point of a transformation is selected from the following pull-down list under the option :

	<Ctrl+0>	Left Top
	<Ctrl+1>	Center Top
	<Ctrl+2>	Right Top
	<Ctrl+3>	Left Center
	<Ctrl+4>	Center
	<Ctrl+5>	Right Center
	<Ctrl+6>	Left Bottom
	<Ctrl+7>	Center Bottom
	<Ctrl+8>	Right Bottom

	<Ctrl+9>	Free attachment mode (arbitrary point selection)

The default setting is the free attachment mode . In this case, the origin point is defined by specifying the coordinates in the property window or by clicking in the 2D window, or else by selecting an existing 2D node. To select a node, you can use the option:

	<N>	Select Node

If the free attachment mode was selected, defining the origin point will be the first step of moving/copying.

Target point is the point defining the target position of the objects after the transformation. This can be the end point of a translation, the end point of a linear array (defining its length and step), or the center of the circular array. The target point of the transformation is always defined by specifying its position by mouse clicking , in the property window, or by selecting a 2D node (the option).

Option for selecting action after the current transformation

You can select a desired action upon completing the transformation from the enclosed list of the option :

	<Alt+X>	Exit Command automatically when finished
	<Alt+O>	Repeat Command for selected elements
	<Alt+N>	Repeat Command for created elements (available for copying only)
	<Alt+A>	Repeat Command for selected and created elements (available for copying only)
	<Alt+S>	Select new elements when finished

The selected option defines the system action upon completing the original transformation:

- the system automatically exits the move/copy command.

, , - upon the completion of the first transformation, the command instantly activates the mode of defining a new transformation. This will be indicated by a new set of objects for transformation, rubberbanding with the pointer, and the respective prompts displayed in the status bar (such as, “Set destination point for Move”). Depending on the selected option, the new set of objects for moving/copying will include either the original objects of the first transformation, or the objects created as a result of the first transformation, or else all of the above.

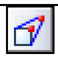
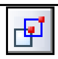
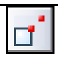
- upon the completion of the first transformation, the command switches to the mode of selecting objects (see the section “Calling the command from menu”).

Common Options of Moving Commands

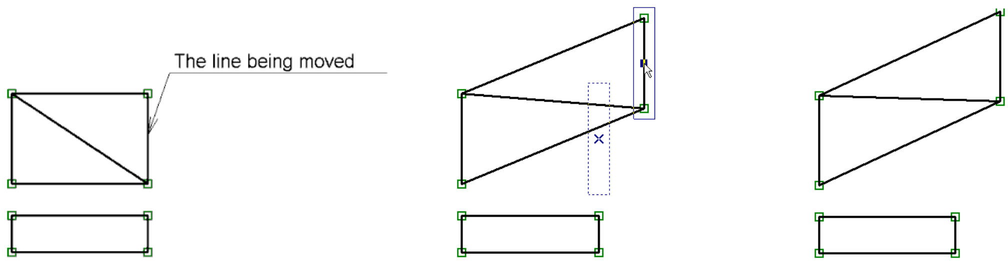
The moving commands include the translation **TM: Move**, rotation **TT: Move and Rotate**, scaling **TA: Move and Scale** and symmetry **TS: Move with Symmetry,** and translation with rotation **TP: Move with translation and rotation**.

Options for selecting moving mode

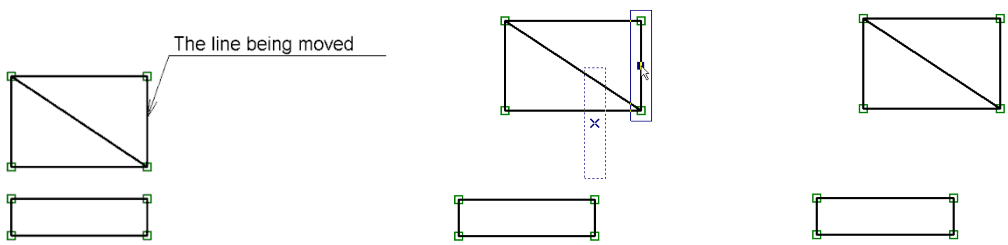
When executing any moving command (translation, rotation, scaling or symmetry), you can set various modes of executing these transformations. The mode selection is done in all cases by one of the automenu options in the list:

	<O>	Change dependent elements
	<G>	Change related elements
	<F>	Change selected elements

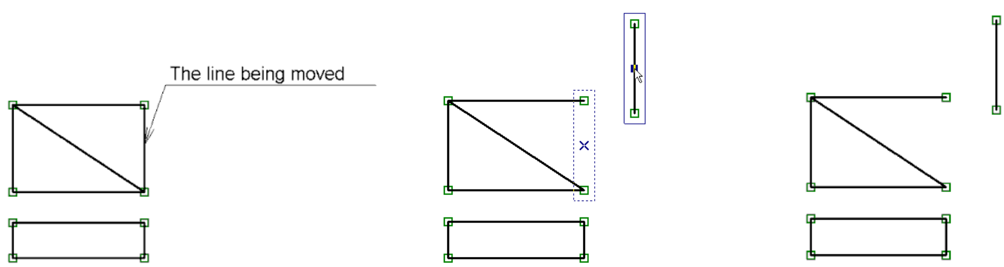
Change dependent elements. The selected elements are moved together with their parents and immediate dependents only.



Change related elements. The selected elements are moved together with all objects related via parents or dependents.



Change selected elements. Only the selected elements are moved, separated from parents and dependents.



In complex cases, when it is impossible to break a relation between the element and its parents and/or dependents, the following exceptions from this rule are possible:

If the selected object can't be separated from a parent, a copy of the parent is created, that moves together with the selected object. The original parent element is left unchanged;

If the selected object cannot be separated from a dependent, then the object is moved, while its copy is created in the original position to which the dependent is attached;

If the selected objects have a common parent construction entity that cannot be moved (such as a node or a construction line), and the parent does not have any other dependents that cannot be moved, then the parent is moved along.

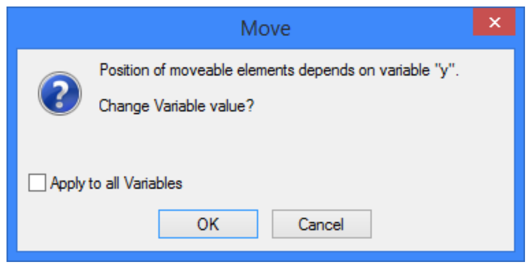
Dynamic model regeneration mode

When defining various types of moving in the modes “Change dependent elements” and “Change related elements”, you can turn on the dynamic model regeneration mode to view the modifications of the elements related or attached to the selected ones (the selected elements themselves are always updated dynamically). To turn on this mode, use the option .

Options for handling variables

If the original position of the selected elements (or the elements used for defining the selected ones) was driven by variables, use the options and when moving the elements. These options cannot be used together. When one is engaged, the other becomes inaccessible.

If none of these options is active, then after selecting the new position for the elements being transformed, a message is displayed on the screen, prompting the user for modifying the value of the respective variable. Pressing the **[Ok]** button completes the transformation with updating the variable. Pressing **[Cancel]** aborts the transformation. In the case when several variables are affected, the prompts will be displayed subsequently for each of the variables. Checking the item “For all variables” in one of the prompt windows confirms adjustment of all the rest of the variables without querying.



With the option turned on, the values of the respective variables are adjusted without querying.

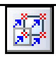
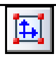
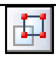
The option replaces all the variables used for defining the coordinates of the transformed elements by their respective values. The variables themselves are not deleted from the model. The variables related to other element parameters are not affected by moving.

Common Options of Copying and Array Creation Commands

This section covers all the commands that allow creating copies of the selected objects. Those are the commands for copying with translation **XM: Create Copy**, copying with rotation **XT: Copy and Rotate**, copying with scaling **XA: Copy and Scale** and creating symmetrical objects **XS: Create Symmetry**, and copying with translation and rotation **XE: Copy with translation and rotation** as well as the array creation commands **XL: Create Linear Array** and **XR: Create Circular Array**.

Options for selecting copying modes

When executing any copying command, one can use various copying modes. The mode selection is done in all cases by one or the automenu options in the list:

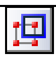
	<F>	Create Associative Copy
	<J>	Create Copy on Associated Constructions
	<G>	Explode Copy keeping relations
	<O>	Explode Copy

Create Associative Copy. This option creates an associative copy, whose elements maintain the relation with the original parent elements. The copied elements will automatically adjust, as the original elements are modified.

Create Copy on Associated Constructions. For selected construction elements the associated copies are created. Copies of the drawing elements are detached from the original parent elements and snapped to the copies of the construction elements. If this mode is used while creating rectangular or circular arrays, the drawing elements will be created only at the moment of array creation. Increasing the number of elements in the array does not lead to appearance of new graphics lines and other drawing elements.

Explode Copy keeping relations. The created copy operation is automatically destroyed. The elements created by this operation become independent from the original parent elements. However, the internal relations are maintained between the resulting objects similar to those that existed between the original elements. The relations with variables are maintained if the variable values are not affected. Otherwise, the variables are replaced by constants.

If a copied object has a parent that was not among the set of the objects to be copied, then upon destroying the copy operation, a copy of the parent element will be created that will move together with the selected object. To prohibit this, turn on the additional option:

	<Alt><T>	Copy only selected elements

With this option turned on, the system tries to separate the objects being copied from their parents that were not included in the set of the objects to copy (similar to the mode “Change selected elements” in the moving commands).

Explode Copy. The created copy is automatically broken up into the separate unattached objects. The copied construction elements become stand-alone objects, regardless of the ways of their parent element creation. All variables that were driving the parameters of the original elements are replaced by constants in the new elements.

For successful copying of the detailing elements (dimensions, leader notes, roughness symbols, tolerances, etc.) in the modes “Create Associative Copy” and “Explode Copy”, make sure that the set of the objects to be copied includes the elements and their parents as well. Otherwise, the copy will not be created. To avoid this, let the system automatically append the selection with the required parent elements by including those in the set of objects to be copied. This mode is turned on by the additional automenu option:

	<K>	Auto selection of required parents

Attachment node creation option

When creating associative copies and arrays, selecting an existing 2D node (the option) as a base point of the transformation automatically establishes a relation between this base point and the selected node. As a result, the position of the base point will change according to modifications to the node position, causing repositioning of the whole copy. If the necessary nodes do not exist, those can be created automatically using the option:

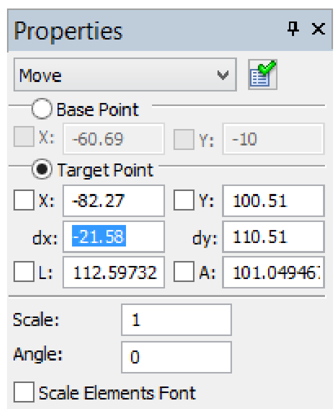
	<T>	Create Node at point

When the option is turned on, a 2D node is created automatically at the point of the mouse click when defining the base points of the transformation. The base points become attached to this node.

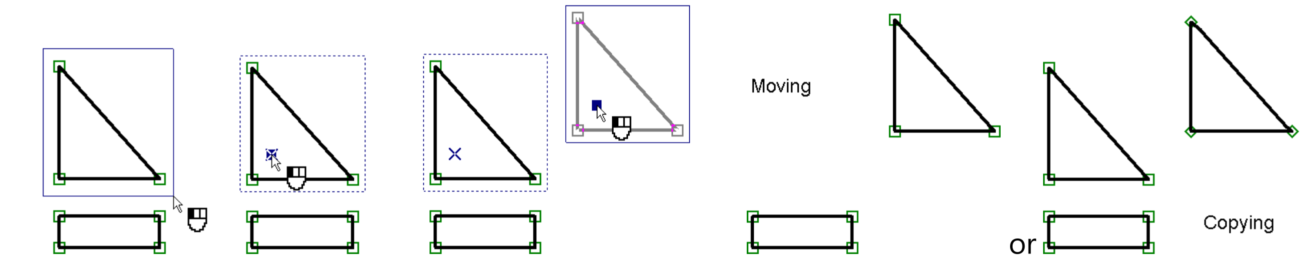
Translation

To define translation or copying with translation, specify the start point (the origin point of the transformation) and the end point (the target point of the transformation). If using the mode of automatic origin point location based on characteristic points of the outlining rectangle, you shouldn't define the start point. The transformation is performed by carrying the start point over the end point.

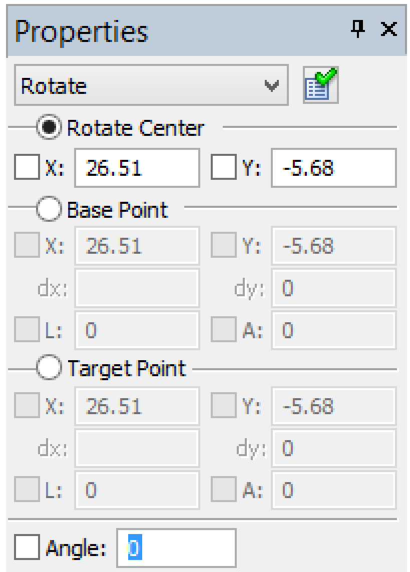
Additionally, you can change the scale and the rotation angle of the translated image in the property window. Option **Scale Elements Font** controls scaling of the drawing elements added to the copy set. When the flag is on font size of all elements being copied will be scaled according to the assigned scale factor. When the flag is off, font size remains unchanged. The flag does not affect font size of the text being copied – it is always scaled.



When defining the end point of the transformation, you can restrict pointer movements by snapping to the coordinate axes. This helps defining a vertical or horizontal transformation. The related options are and . Turning on either option allows pointer movements along the respective axis. When turned off, the option prohibits movements in the respective direction. With both options turned off, the moving (copying with translation) can only be done among the pages without changing the coordinates of the object being translated.



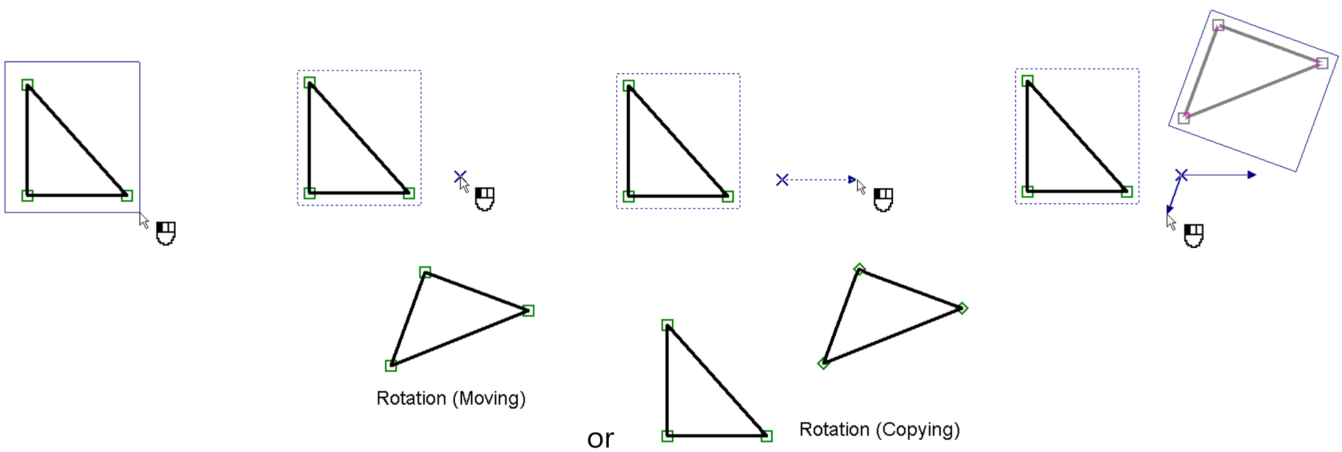
Rotation



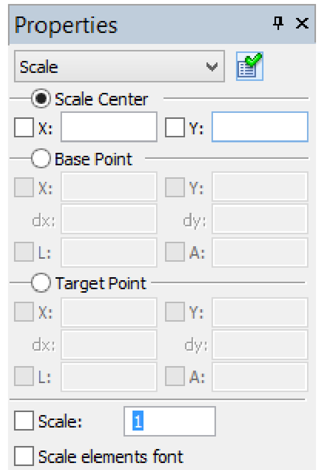
When defining a rotation, three points are specified in a general case: the center of rotation (the attachment point), the start point and the end point. Rotation is done about the specified center. The rotation angle is defined as the angle between the vectors constructed from the rotation center to the start and end points.

The value of the rotation angle can be specified numerically or by a variable in the property window.

When using the mode of the origin point automatic definition based on characteristic points of the outlining rectangle, the rotation center is defined automatically.



Scaling



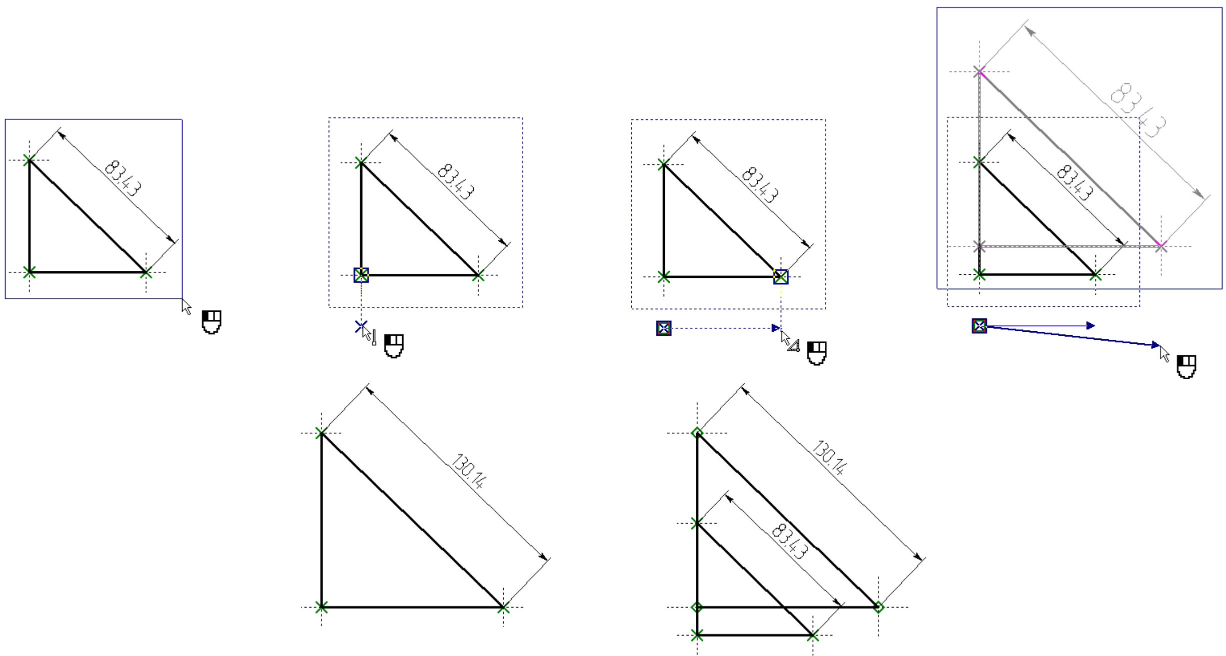
When scaling and image, you need to specify the three points, just as in the case of rotation: the center of scaling, the start point of scaling and the end point of scaling.

The center of scaling is determined automatically in the case of using the mode of automatic definition of the origin point based on the characteristic points of the outlining rectangle.

The scaling factor is computed as the ratio of the distances between the center and the end point and between the center and the start point.

The scaling factor can also be defined numerically or by a variable in the property window.

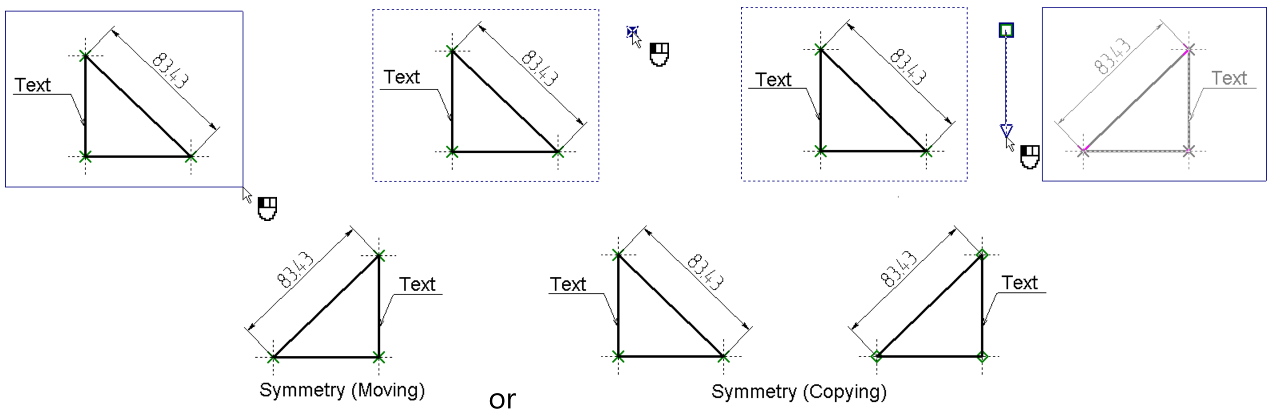
Scale elements font flag controls scaling of the drawing elements (analogous to the flag with the same name for Translation).

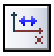
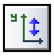
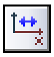
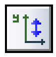


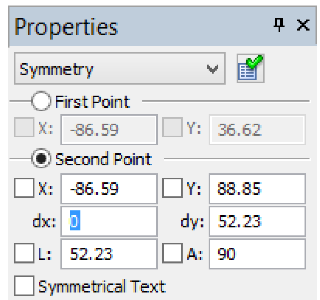
Scaling (Moving) or Scaling (Copying)

Symmetry

To define symmetry, specify just the symmetry axis to mirror the selected object about. The axis can be defined by either selecting an existing straight line or segment using the option , or by defining two points for the symmetry axis to pass through. Existing 2D nodes can be used as the points.

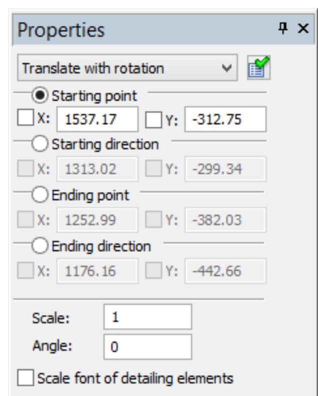


The options  and  help speed up definition of horizontal/vertical symmetry axes. Both options are active by default. This means, an arbitrary axis can be defined (you need to specify two arbitrary points or an arbitrary straight line/segment). Undoing one of the options turns on the mode of creating a horizontal/vertical axis. The remaining active option defines the axis being:  -horizontal,  -vertical. In this case, simply select one point for the symmetry axis to pass through.



When mirroring text, you can check the additional flag "Symmetrical Text" in the property window. When the flag is cleared, only the text position is affected by the symmetry. When set, the text contents are also mirrored symmetrically.

Moving with Rotation



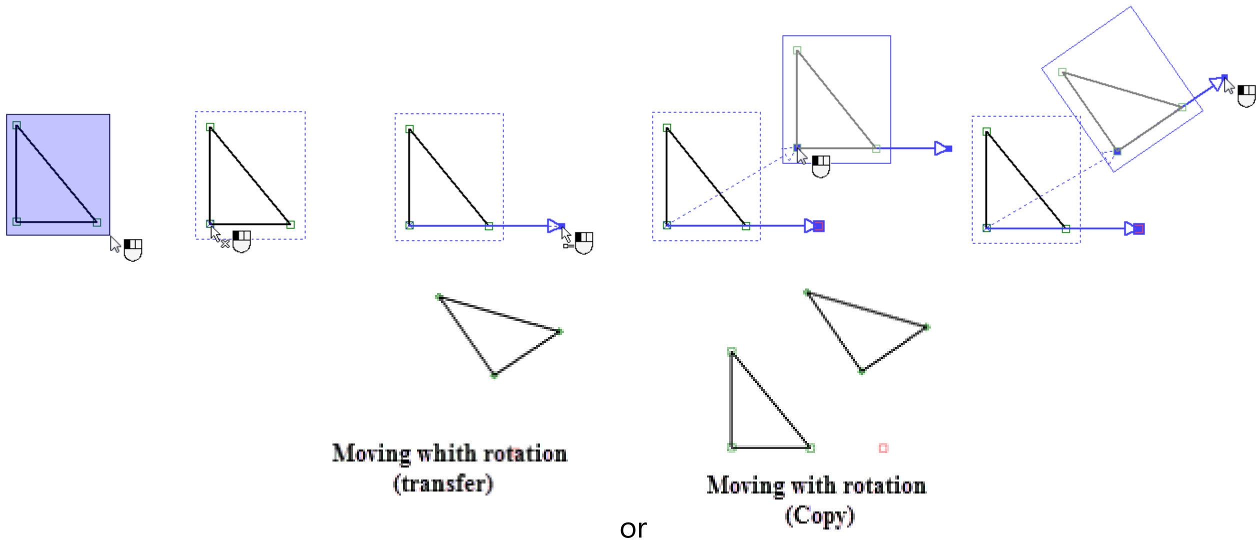
For creation of translations with rotation or of analogous copying it is required to specify:

- initial point (this is an initial point of transformation and the first point of the initial direction vector);
- initial direction vector (the end point of a vector is specified);
- end point (this is a target point of transformation and the first point of the final direction vector);
- final direction vector (the end point of a vector is specified).

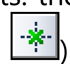
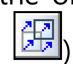
If the mode of automatic determination of the initial point on the basis of one of the characteristic point of the encompassing rectangle is enabled, the initial point is not specified.

Transformation is carried out by translating the initial point to the end point (target) and rotating with respect to the end point at an angle between the initial and final direction vectors. In the properties window you can specify the magnitude of angle of additional rotation (it is added to the angle specified by the vectors).

In addition, in the properties window it is possible to change the scale of the image being translated. The **Scale font size of formatting elements** flag allows us to control the scaling of the formatting elements that are included into the set of copied objects.



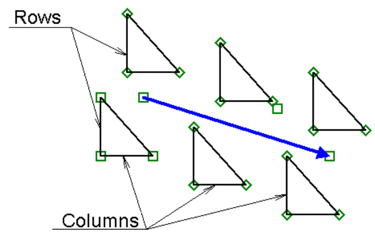
Linear Array

To create a linear array, you need to specify two points: the start (origin) and the end (target). 2D nodes can be used as points: the existing ones (the option ) or the ones automatically created using the option . When creating an associative array (the option ), it will be attached to the nodes automatically. The specified points define the direction vector of the array and its step, length or the number of copies, depending on the way of defining the array. The copies will be positioned along the array direction vector.

After defining the first point, the array elements start rubberbanding on the screen. Their number depends on the default setting of the number of copies in the property window. To complete the array creation, simply select the position of the end point.

The start point of the array is determined automatically in the case of using the mode of automatic definition of the origin point based on the characteristic points of the outlining rectangle.

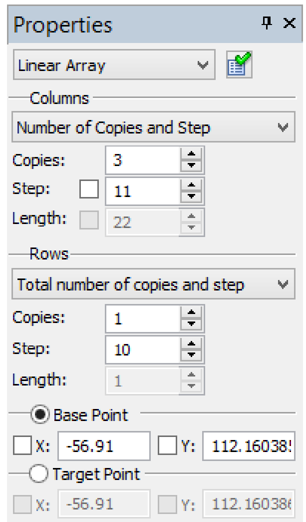
The array can be two-dimensional, that is, composed of several rows. The rows are created in the direction orthogonal to the direction vector. The number of copies in the direction orthogonal to the specified vector (that is, the number of rows), and their parameters (the step or the total length) can be defined among the array parameters in the property window.



The group **Columns** defines the parameters of the columns in the linear arrays (the copies positioned along with the direction vector):

Mode. Sets the definition mode of the linear array: “Number of Copies and Step”, “Length and Step”, “Number of Copies and Length”.

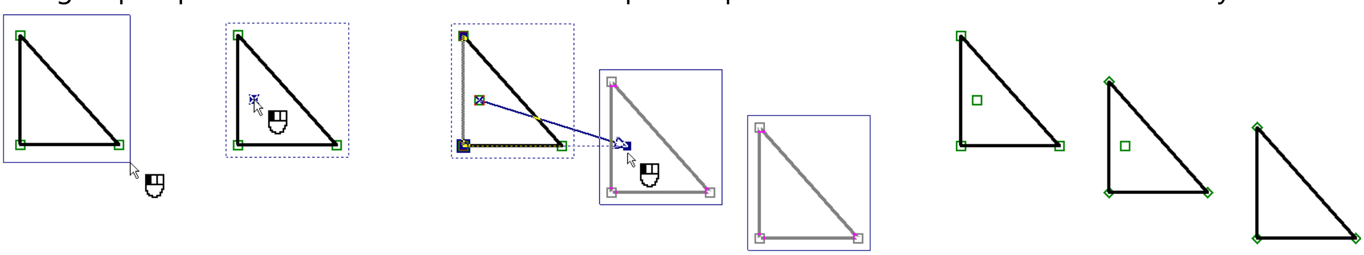
Depending on the selected definition mode, the distance between the points specified at the array creation can be defining, respectively: the step (the number of copies is always defined by a numerical value); the step or the length, at user’s choice; the total length of the array (the number of copies is again defined by a numerical value).



The values of the step and the total length of the array can be defined by numerical values in the property window as well (the parameters **Step** and **Length**). To do this, check the flag next to the desired parameter. The specified points will in this case define the array direction only.

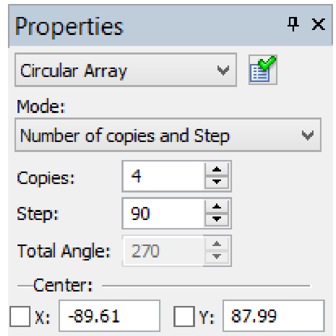
The number of copies is always defined by a numerical value using the parameter **Copies**. The original elements are included in the count of copies. The accessibility of parameters **Copies**, **Step** and **Length** is determined by the selected mode of defining the array.

The group of parameters “**Rows**” defines the respective parameters of the rows of a linear array.



Circular Array

To create a circular array, after selecting the objects to copy you need to specify the center point of the array. You can specify a 2D node as the center using the option . The option  allows creating a node in the specified point with automatic attachment to that node (for the associative array). After selecting the objects to copy and before specifying the center of the array, the array elements rubberband on the screen. The number of elements and the angle swept by the elements are defined by the default values. The array parameters can be modified in the property window. Here you can define:

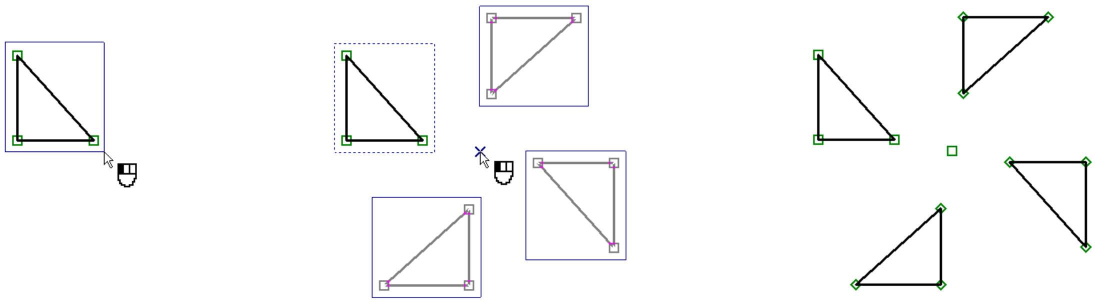


Mode. Sets the definition mode for the circular array: **Number of copies and total angle**, **Total angle and Step** or **Number of copies and Step**. Depending on the selected mode, some parameters may be inaccessible (computed automatically):

Copies. Defines the total number of copies in the array, including the count of the original elements.

Step. Defines the angle between the copies of the array.

Total Angle. This parameter allows defining the total angle that will be swept by the array elements.



Calling the Commands in Transparent Mode

The command **TM: Move** can be called in “transparent mode” when one or several T-FLEX CAD 2D elements are selected. The elements can be selected by box and/or using <Shift>+, <Ctrl>+. After selecting the elements, simply point the mouse at one of the highlighted nodes or the border lines of the object being moved. The pointer then assumes the shape (when pointing to a line) or (when pointing to a node), that indicates readiness of the command **TM: Move**.

Next, two ways of acting are possible:

Click and release without moving the pointer. Then move the pointer to the target point of copying and once again click or press the <Enter> key. This way is convenient when moving objects from one page to another.

Depress and move the pointer while holding down the mouse button. Moving will be completed when the button is released or the <Enter> key is pressed.

In either case, after first pressing of the both of the automenu and the property window will appear as appropriate in the “**TM: Move**”. command. If necessary, you can switch to another moving, copying or array creation command, using the automenu or the property window.

EDITING COPY OR ARRAY

All elements obtained by moving or non-associative copying can be edited as normal construction or graphic elements.

The associative copies created using the copying commands are edited by the command **EY: Edit Copy Operations**. The command can be called by one of the following means:

Keyboard	Textual Menu	Icon
<EY>	Edit > Draw > Copy	

Upon calling the command, select the copies to be edited. To select one copy, you can use . Multiple selection can be done by box selection or selection by <Shift>+, <Ctrl>+ or by the automenu option:

	<*>	Select All Elements

When the copies are selected, both the selected and the original elements are highlighted, as well is the transformation vector (or the attachment point in the cases of scaling, rotating or creating a circular array).

Upon multiple selection of copies or arrays, the option is available in the automenu:

	<Del>	Delete selected Element(s)

Selection of a single copy or array makes the following options available in the automenu:

	<Enter>	Finish Editing
	<P>	Set selected Element(s) Parameters
	<O>	Explode Copy
	<G>	Explode Copy keeping relations
	<H>	Select Clipping Hatch (available only for translated, rotated and scaled copies)
	<K>	Cancel selection of clipping Hatch (available only for copies clipped by a hatch)
	<S>	Edit copied elements list
	<I>	Select Other Element
	<Del>	Delete selected Element(s)

When selecting a copy (an array), the property window displays the dialog for editing the copy parameters, similar to that used at the time of the copy creation. It allows modifying the copy parameters. You can use the parameters dialog box for this purpose as well, called using the option .

The two options - and - are provided for breaking up copies, that is, for converting those into a set of independent elements. The copy (array) itself as a T-FLEX CAD element is deleted at this moment. The result of applying these options corresponds with the respective modes of the copy (array) creation.

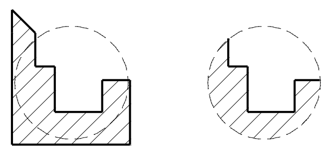
Elements created as a result of breaking up a copy can automatically be combined into a group. By default the group is assigned the name of the broken copy. It is possible to refuse the automatic creation of the group.

To delete a copy altogether, you can use the option .

The option is provided for editing the list of the original elements of the copy (array). In this mode, you can add the new elements to the list of the objects to be copied and delete some of the elements from the list of selected.

The option is accessible for copies created by the commands **XM: Create Copy**, **XT: Copy and Rotate**, **XA: Copy and Scale, XE: Copy with Translation and Rotation.** It allows clipping the image of the copy by a hatch. To do this, upon calling the option, select the desired hatch by clicking .

If the clipping hatch is used for this purpose only, we recommend setting its flag “Invisible”. The diagram shows a copy image before and after selecting an invisible hatch for clipping. In addition to the described capabilities, the attachment points can be redefined for the selected copy (array).

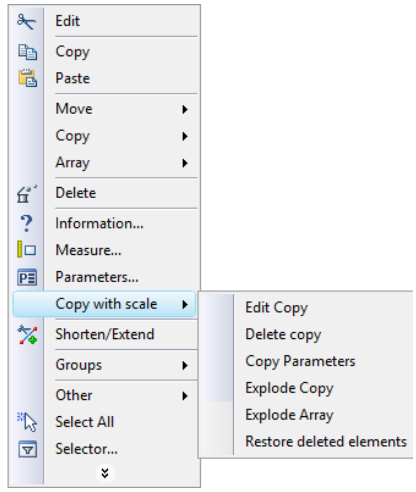


To do this, select one of the transformation vector ends or an attachment point by clicking it (in the cases of scaling, rotating and the circular array). After that, specify the new attachment point(s). At this moment, the following options will be available in the automenu:

	<T>	Create Node at point
	<N>	Select Node

After defining the new attachment point, the element being edited will be drawn according to the applied changes.

The commands for editing a copy or an array, or their elements, are also accessible via the context menu. Simply select one of the elements of a copy/array and right click. In the coming up context menu, the editing commands will be provided for the selected element, as well as for the whole copy or the array.



You can change the properties of the selected element of a copy/array (the default properties are copied from the parent element). You can also delete the selected element of a copy/array (without deleting the copy itself). In this case, the copy/array itself maintains the information about the deleted element, so that it can be restored in the future.

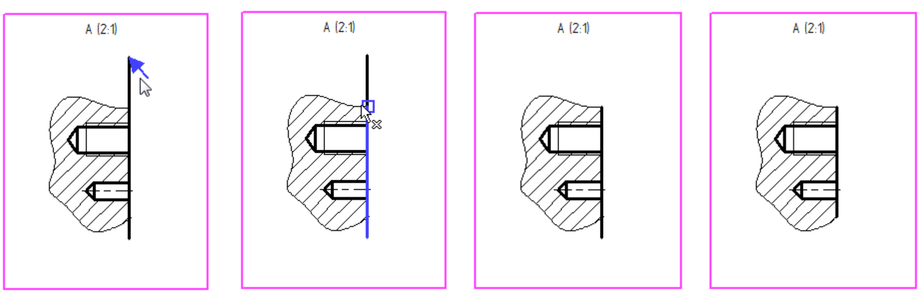
In the context menu of graphic lines obtained by copying, the **Shorten/extend** command is also available. This command allows us to modify the length of the visible part of the line. This function is described in more detail below.

The editing commands for the copy/array itself are grouped in the context menu into a submenu, named according to the copy type (such as, “Move Copy”, as shown on the diagram). An additional command is provided among the copy/array editing commands, **Restore deleted elements**, specifically for restoring all deleted elements of the copy.

Changing length of visible part of a line obtained by copying

The **Shorten/extend** command, which is available in the context menu of graphic lines obtained by copying, allows us to modify the length of the visible part of the line. It is possible both to shorten and extend the line of the copy. However, the function of extension is only available for segments and arcs.

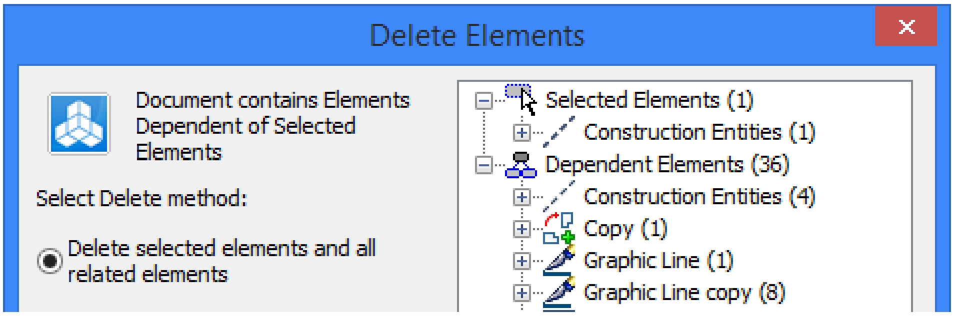
The part of the line that is to be changed is indicated by 2D nodes. At first when you work with the command it is required to indicate the part of the line which must be modified, then – the node which defines the new end point of the line. For closed lines it is necessary to indicate two nodes and the part of the line which must remain. The 2D nodes, which are being used, can be located on the line itself or at some distance away from it. In the latter case the end point of the line is defined as a point on the line that is the closest to the selected node.



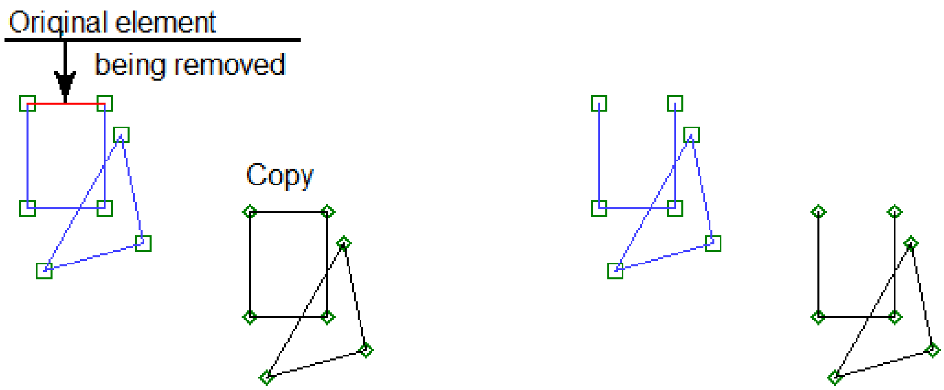
The context menu for the line modified with the help of the **Shorten/extend** command contains the **Reset** command. It allows us to quickly restore the original appearance of the line-copy.

Removing original objects of copy/array

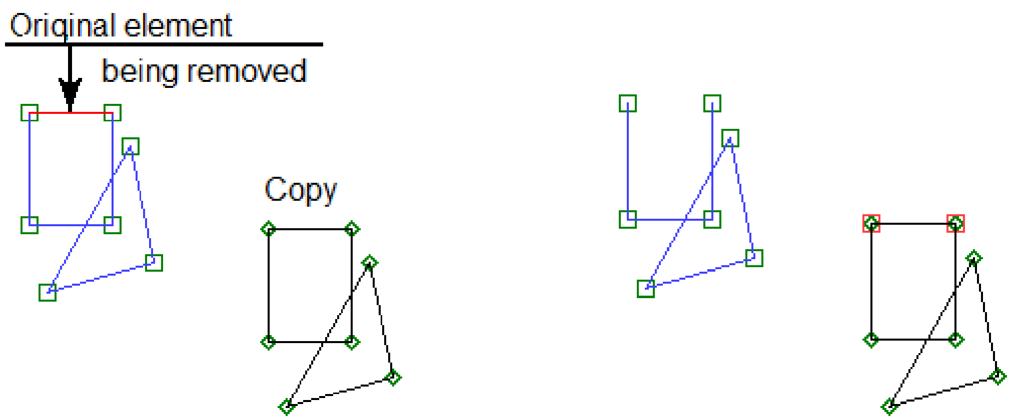
When deleting an original element of the copy/array, two variants of the system's response are possible depending on the option selected in the dialog of the command of deletion.



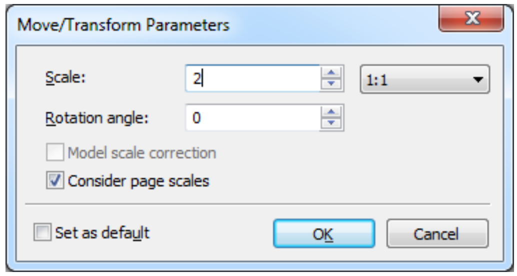
If the "Delete selected and dependent elements" option is chosen, the element that is being removed is simply excluded from the set of original elements of the copy/array. As a result, the view of the copy/array will change: the elements obtained by copying of the element that has been deleted will disappear.



When the "Delete only selected elements by changing the way of specifying the dependent elements" option is selected, the element that is being removed is also excluded from the original set of the copy/array. But, in addition, instead of copies that have been disappeared, identical externally free objects are created. This means that the result of copying visually does not change.



Copying between pages with different scales



The dialog of parameters of the already created 2D copy (by translation, rotation, scaling – except symmetry) contains two additional flags: **Correction for a scale of model** and **Compensate scales of pages**. These flags are taken into account when copying between pages is performed.

The **Correction for a scale of model** flag is available only for translations created automatically upon construction of taken out elements.

Description of this flag can be found in the "Drawing views. Taken out elements" chapter.

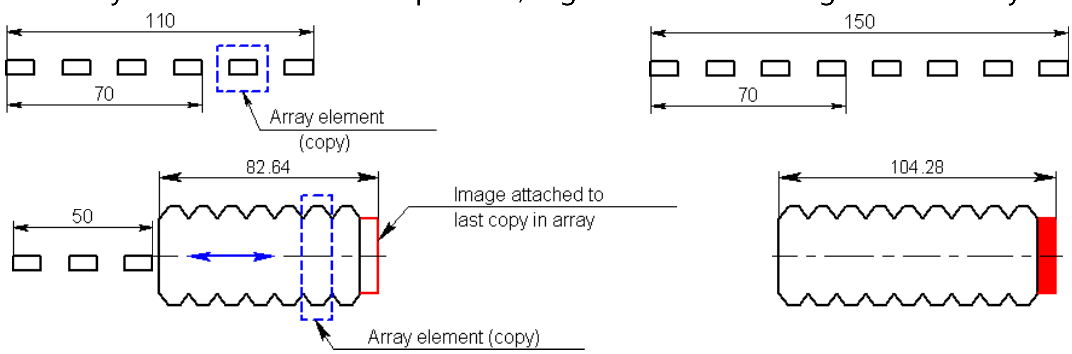
The **Compensate scales of pages** flag allows us to achieve the correct ratio between the dimensions of the elements on the original page and the resulting page of copying when the scales of these pages are different. When this flag is enabled, to obtain the scale of the copy, the scales of both pages are taken into consideration:

$$\textit{Resulting scale of the copy} = \frac{\textit{Given scale of the copy* Scale of target page of copying}}{\textit{Scale of original page of copying}}$$

By default, this flag is enabled.

Particulars of handling variable arrays

When modifying the number of copies in a linear or circular array, adding or deleting copies occurs in the position immediately before the last copy in the array. This feature helps, for instance, keeping the dimensions between the outer elements of the array throughout modifications to the number of copies in the array. The dimensions set on "inner" copies of the array may disappear as the total number of copies is reduced. The same is true for any 2D constructions: the elements attached to the outer copies of an array will always maintain the correct position, regardless of the changes to the array.



When creating a hatch based on an array elements, hold on to the following technique: before creating the hatch, set the maximum necessary number of copies in the array. The hatch should be defined using the automatic contour search mode. In this case, the hatch will behave correctly in the future and will not "break" under any modifications to the number of copies in the array (within the initially defined range).

In the three-dimensional modeling mode, a 2D array placed on a workplane can be used for creating a 3D profile. The profile can be created based either on the graphic lines directly, or on the hatch constructed by those graphic lines. When using an array with a variable number of elements, use of an intermediate hatch (based on the lines of the array) for the profile creation makes sense only in the case when you know the maximum number of copies in the array. In all other cases, the 3D profile should be created based on the graphic lines of the array. In the course of future construction, keep in mind that indexing of geometrical items within the resulting 3D element (such as edges, vertices, etc.) will be changing, as you modify the number of copies in the array that defines this 3D element.

COPYING VIA CLIPBOARD

Besides the moving, copying and array creation commands, T-FLEX CAD also supports the mechanism of copying via the clipboard. Its function is mostly similar to the copying command "**XM: Create Copy**", yet has some additional capabilities. The clipboard allows copying any 2D elements, except for the drawing views and multipage text and BOMs. This mechanism is recommended for use in the following cases:

For copying across several documents within the same T-FLEX CAD application;

For exchanging data with other applications.

Copying via the clipboard includes four commands available in the context menu and in the menu **Edit** after selecting the objects to copy: **XC: Copy**, **XI: Copy with Insertion Point**, **XP: Paste**, **XE: Paste Special**.

> The standard key combinations are used with the clipboard: <Ctrl><C>**,** <Ctrl><V>**,** <Ctrl><Ins>**,** <Shift><Ins>**.**

The command **XC: Copy** - places the selected object on the clipboard:

Icon	Ribbon
	Edit → Edit → Copy
Keyboard	Textual Menu
<XC>, <Ctrl><C>	Edit > Copy

To use this command, simply select the elements, and then call the command. No additional actions are required. When later pasting the copied objects into the T-FLEX CAD document, you can use the characteristic points of the copied object (center, upper left corner, upper right corner, etc.).

The command **XI: Copy with Insertion Point** - places the selected object on the clipboard with the specified attachment point:

Icon	Ribbon
	Edit → Edit → Copy with Point
Keyboard	Textual Menu
<XI>	Edit > Copy with Point

Upon calling the command and selecting objects, you need to specify an arbitrary point (2D node), to which the objects will be attached when pasted in the T-FLEX CAD document. The following options will be available in the automenu in this case:

	<N>	Select Node
	<A>	Set absolute coordinates
	<Esc>	Cancel selection

Object snapping is active when defining the point, similar to that used in sketching. Upon pasting the copied objects in the T-FLEX CAD document, the attachment can be defined by specifying a point or by the characteristic points of the copied object.

When copying, the data are put on the T-FLEX CAD clipboard in a specific internal format. Besides that, to ensure interaction with external applications, the selected drawing elements are placed on the clipboard in the image format Enhanced Metafile (EMF). However, if a single element of the type "Text" was selected for copying, then additionally the textual data is placed on the clipboard in the following formats:

1. T-FLEX Paragraph Text (except the string text),
2. RTF (except the string text),
3. Unformatted text.

This supports data exchange both within the same T-FLEX CAD application and across several different applications. To insert data from the clipboard, use the commands **XP: Paste**, **XE: Paste Special**.

> These commands are available only if there is some data in the clipboard.

The command **XP: Paste**:

Icon	Ribbon
	Edit → Edit → Paste
Keyboard	Textual Menu
<XP>, <Ctrl><V>	Edit > Paste

In this command, the format selection upon pasting the clipboard contents is done by the application itself. The program scans through the clipboard, searching for the appropriate format among the clipboard data. The data will be pasted in the first appropriate format found. The order of the format search is as follows:

1. Internal T-FLEX CAD format (used only when copying within one T-FLEX CAD application)
2. T-FLEX Paragraph Text
3. RTF
4. Unformatted text
5. EMF

6. BMP (bitmap image)

Upon calling the command, the elements being pasted will be rubberbanding with the pointer. The attachment point for the elements being pasted is defined by clicking or by using the option .

The following options are provided in the automenu:

	<N>	Select Node
	<E>	Use Variables when names are coincident (only when pasting in another T-FLEX CAD document)
	<Alt><T>	Copy only selected elements
	<U>	Move along X axis (only when pasting in the same T-FLEX CAD document)
	<V>	Move along Y axis (only when pasting in the same T-FLEX CAD document)
	<Ctrl+4>	(Selection of the attachment point)
	<Alt+x>	(Selection of the action after pasting)
	<Esc>	Exit command

The options and are used for blocking pointer movement in the directions of the respective coordinate axes. To specify the exact position of the copied elements or the offset with respect to the original object, use the command property window. This option is available only when pasting the copied elements in the same T-FLEX CAD document.

The group of options for selecting the attachment point allows specifying the point to which the pasted object should be attached. The characteristic points of the object are available for selection. If the objects were placed on the clipboard by using the command "**XI: Copy with Insertion Point**", then an additional attachment point can be defined in the command being discussed (the option).

When using the clipboard copying mechanism, the system forces copying of the parent elements of the objects being copied (in the cases, when the former were not explicitly included in the set of the copied elements). This feature is similar to that used in the copying mode "Explode Copy keeping relations". In the current case, you can also prohibit this system behavior by using the option . With this option activated, the elements, whose parents were not included in the set of elements to be copied, will be converted into independent objects.

The copied elements can have relations with variables (except for the variables describing the position of these elements). To replace the respective parameters of the elements being copied by the variables defined in the target T-FLEX CAD document (the document where the elements are copied to), use the option . This option is only available when pasting the copied elements in another T-FLEX CAD document within the same active application. If the option is turned on, the names of the variables are compared in the target document versus those in the copied objects. If the names coincide, then the copied elements take on the relations with the respective variables of the current document. The variables, for which no match is found, are replaced by their values. If this option is turned off, the relations with all variables are broken, the variables being replaced by constants.

The group of options is used for the same purpose as in the moving, copying and array creation commands. Those define the action that will be automatically performed upon completion of pasting the clipboard contents. The following choices are provided:

- the system exits the command after pasting the clipboard contents;

- repeated pasting mode - upon pasting the first instance, the copied object starts rubberbanding with the pointer. The system waits for the user inputting the attachment point for the next copy. Upon selecting a second point, another copy is pasted, and so on. The copy creation can be interrupted by right clicking or pressing ;

- the copying command **XI: Copy with Insertion Point** is automatically executed over the pasted object;

- is similar to the previous choice; however, before calling the copying command, the system turns on the mode of editing the list of selected elements.

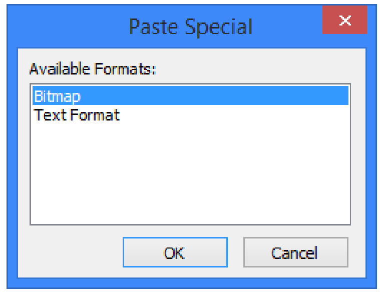
When pasting data from different applications, either a textual or a graphic format is used. If a textual format is used, then a " Paragraph text" element is automatically created, and then its editing command is launched, **ET: Edit Text**. When inserting images, the command is launched for the "Picture" element creation, **IP: Insert Picture**.

The dialog box will appear when pasting clipboard data with AutoCAD objects into a T-FLEX CAD document (see more details in "Exporting and Importing Documents" chapter). This dialog is used for specifying the general import parameters of an AutoCAD document. Then **XP: Paste** command will start for setting location parameters of the elements being inserted – position, angle of rotation, scale factor.

The command **XE: Paste Special**:

Icon	Ribbon
	Edit → Edit → Paste Special…
Keyboard	Textual Menu
<XE>	Edit > Paste Special…

This command allows the user to manually select a format for the clipboard contents to be pasted. The command dialog displays the list of formats present in the clipboard at the time of calling the command. Depending on what format the user selects, the system goes into the T-FLEX CAD object pasting mode, or pasting pictures in the EMF or BMP format, plain text or formatted text (RTF).



ELEMENT REPLACEMENT

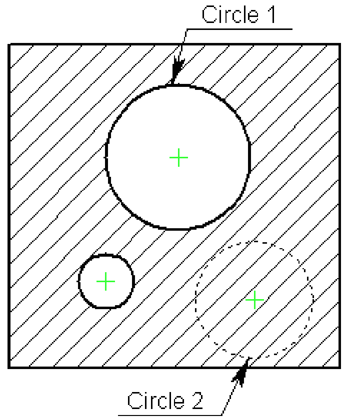
Let us reiterate that parametric properties of T-FLEX CAD models are managed by applying drawing elements over the construction entities. Meanwhile, all construction elements are defined using various geometrical relations with several base elements. This array of references from one element to another makes the model parametric. The command "RL: Replace Element" allows replacing a construction element by an element of the same type, that is, replacing all references to the original element in the model by references to another construction element. If necessary, the original element can be automatically deleted from the model upon the replacement.

The command "RL: Replace Element" can be called by one of the following means:

Icon	Ribbon
	Edit→ Edit → Replace
Keyboard	**Textual Menu**
<RL>, <Ctrl>+<H>	Edit > Replace

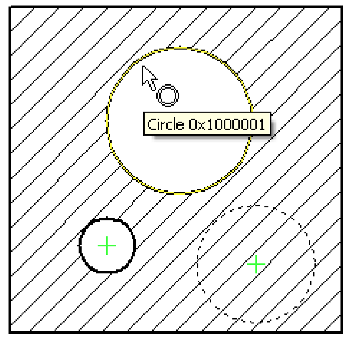
Upon calling the command, the following options become available in the automenu:

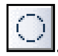
	<L>	Select Line
	<C>	Select Circle
	<E>	Select Ellipse
	<S>	Select Spline
	<N>	Select Node
	<I>	Select Other Element
	<Del>	Delete Source Element After Replacement
	<Esc>	Exit command



Let's review the use of the command on a simple example. A hatch is created in the drawing, whose contour was defined by a circle. We need to modify the model so that a different circle, “Circle 2”, is used in the future instead of “Circle 1”. That's use the command "RL: Replace Element" for this purpose.

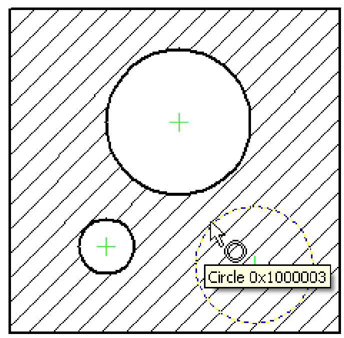
When using this command, the first step is selecting the source element.



In the example, it is a circle. It can be selected using the option . The selected element will be highlighted in the drawing.

When working with drawings crowded with elements of the same type, one can miss the selection. The option  helps selecting a neighboring element in such a case. It cancels the last element selection and highlights the element of the same type nearest to the previous.

Next, you need to select the target element of the same type for replacement.

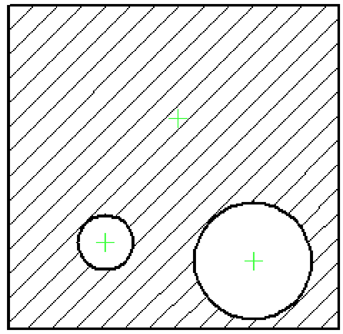


When selecting such element, only the elements of the same type as the source element will be pre-highlighted in the drawing. The only option available in the automenu will be the option for selecting elements of the respective type.

In this example, select the “Circle 2”.

Upon selecting the source and the target construction element, confirm the element replacement using the option:

	<End>	Replace Element

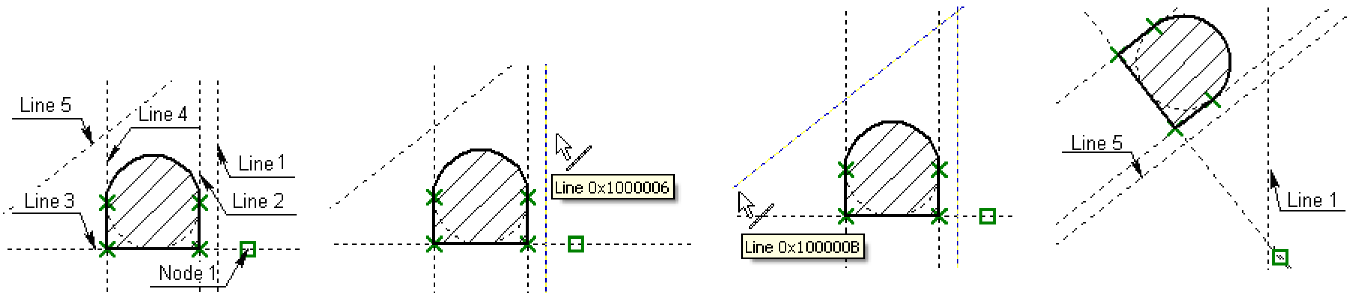


In the course of the replacement, all the elements constructed relative to the source element are rearranged with respect to the target one. In this example, it was the hatch.

If necessary, the source construction element can be deleted from the model upon the replacement. The mode of deleting the source element is turned on by the icon .

In the following example, the construction began with creating a vertical line – “Line 1”. All later constructions were done relative to this line. The “Line 2” was constructed as parallel to the first one, and the “Line 3” - orthogonal, through the “Node 1”. A tangency circle and the “Line 4” were later constructed relative to the two latter lines.

Let's replace the base line “Line 1” by a new line “Line 5”.

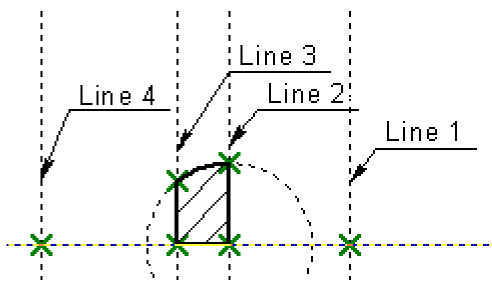


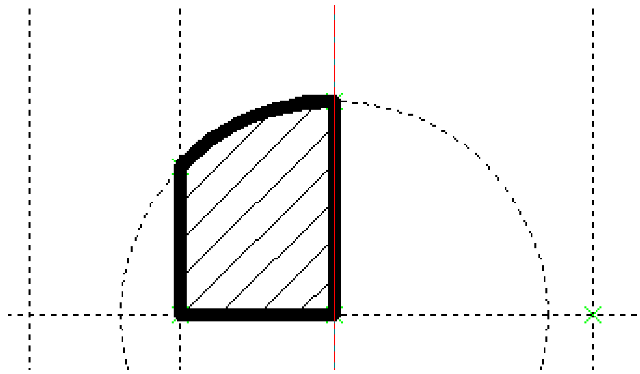
Upon the replacement, all construction elements that were defined relative to the “Line 1” are rebuilt relative to the “Line 5”.

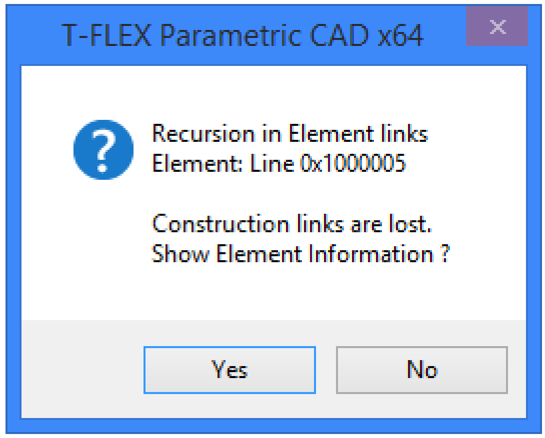
When using the command **RL: Replace Element,** the target element should not be a child of the source one. Otherwise, a message is output about the encountered recursion.

In the next example, the “Line 2” is constructed relative to the base line, “Line 1”. In turn, the lines “Line 3” and “Line 4” are constructed relative to the “Line 2”. A circle is created with the center at the intersection between the “Line 2” and the horizontal line in such a way, that it intersects the “Line 3” and not the “Line 4”.

An attempt to replace the “Line 1” by the “Line 2” in this model causes the message about recursion.

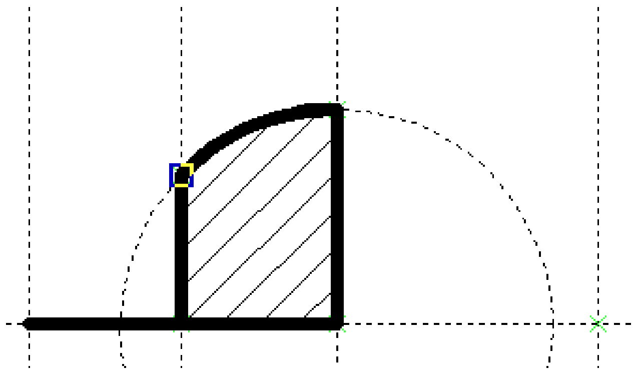


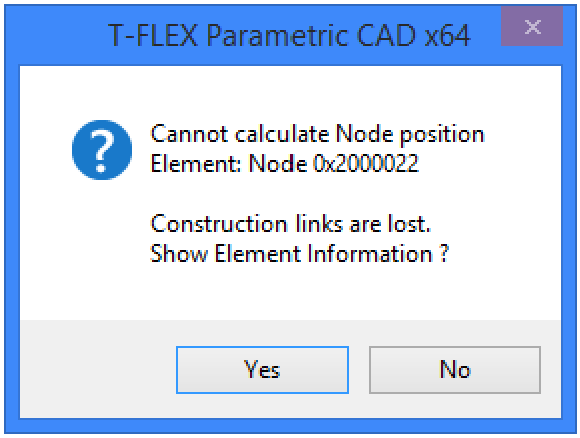




The command may also fail when some drawing element position cannot be defined relative to the target element.

For example, when creating the drawing of the above example, one of the nodes was defined as the node at the intersection between the circle and the “Line 3”. An attempt to replace the “Line 3” by the “Line 4” will cause an error message, since the system will not be able to define the position of this node after the replacement.





DRAWING MODIFICATION VIA DIMENSIONS

T-FLEX CAD has the capability of modifying the model (3D model or 2D drawing) by using the dimensions. The user specifies the new dimension nominal value, and the system automatically rebuilds the 3D model or drawing based on parametric dependencies. Such editing is supported for both 2D dimensions (that is, dimensions in a drawing) and for 3D dimensions (dimensions in a 3D model), as well as their corresponding dimensions on 2D projections.

Decides that, there is a provision for automatically calculation of all dimensions to the middle of the tolerance range.

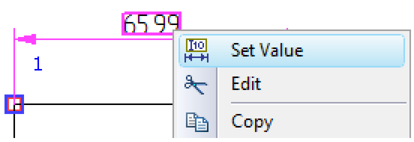
DIMENSION VALUE MODIFICATION COMMAND

In T-FLEX CAD it is possible to edit both the drawing and the 3D model by modifying the nominal values of dimensions created therein. The command **PE: Set Dimension Values** is used for this purpose. The command can be used either in the “transparent mode” or explicitly called by one of the following means:

Icon	Ribbon
	Parameters → Tools → Set Value
Keyboard	Textual Menu
<PE>	Parameters > Dimensions > Set Value

Besides that, this command is available in the context menu when selecting a dimension (whether a 2D dimension or a 3D dimension).

To enable the command in the transparent mode, you need to set the flag **Dimension edit is transparent** in the command **SO: Set System Options**, the tab **2D**. After that, selecting any dimension value string by launches the command **PE: Set Dimension Values.** The selected dimension value is highlighted, ready for editing.



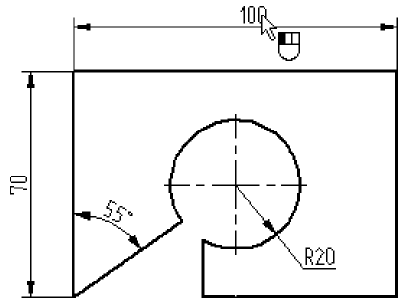
If the transparent mode is not set for this command, dimension values can be selected for editing only after the explicit command call.

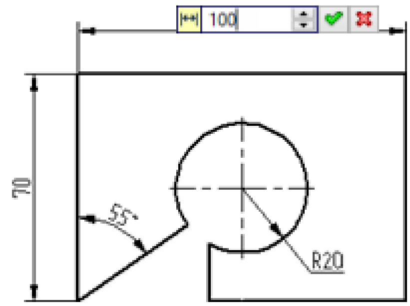
When a dimension value is modified, the system tries to find one construction element, whose modification affects the dimension value. If such element is found, it will be modified according to the requested dimension value. Otherwise, the drawing will stay unchanged. If several such construction elements are found, then the system will select the one of them, whose modification will affect as few other dimensions as possible. When the position of a construction element is modified, other 2D elements will be adjusted as well (including construction lines and/or graphic lines, dimensions, leader notes, etc.), that are related with it. If the position of the elements being adjusted was driven by variables, then the system will query you for the automatic adjustment of their values.

Upon calling the command, the following icons appear in the automenu:

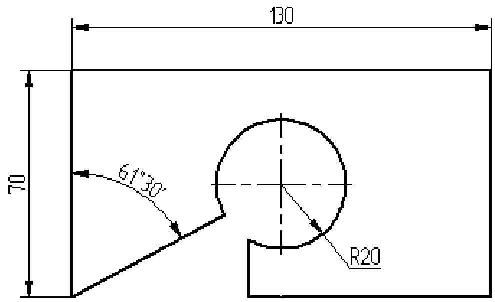
	<Enter>	Finish Value input
	<Esc>	Exit command

Select the dimension and enter the new nominal value:





Then confirm the input by pressing the <Enter> key or . As was already mentioned, the drawing changes only when it is possible.



The command will not work correctly in the following cases:

The flag “Manually” is set in the dimension parameters, or the parameter “Dimension text” is assigned the value “No parameters”.

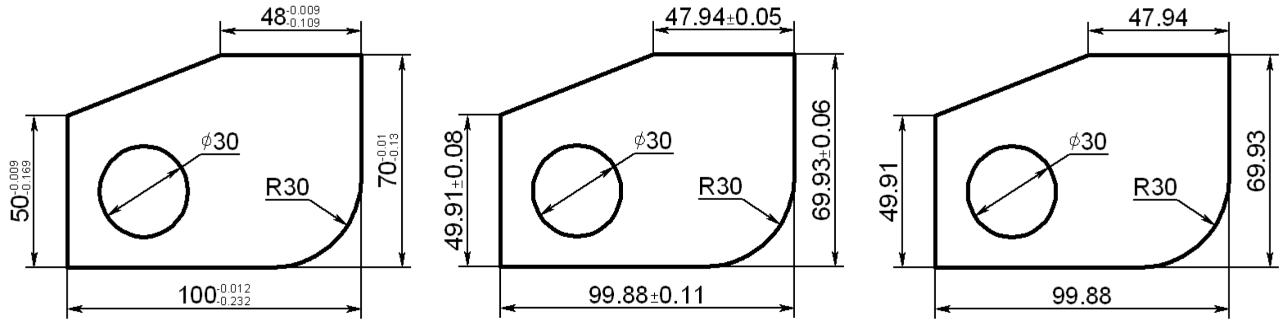
The referenced construction elements definition does not allow their position modifications. For example, you cannot change the value of a circle radius if the circle is tangent to three entities.

Changes in the construction entity positions caused by the dimension modification make the system unable to define the position of some element with respect to other affected entities.

In all of these cases, the dimension reverts to the original value, with no changes done to the drawing.

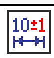
"Recalculate Dimensions to middle of tolerance field" command

Besides the command that allows arbitrary modifications to dimension values, there is also a command in T-FLEX CAD to automatically recalculate dimensions to the middle of the tolerance range. This command performs a correction of nominal values for all selected dimensions in such a way as to make their values balanced about the median of their own tolerance range. The new tolerance range (also referred to as "tolerance field", or "zone") becomes symmetrical. This functionality can be used when adapting a 3D model to the toolpath calculation in the CNC module.

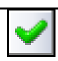


> Warning: this action is not reversible. That means, after recalculating all dimensions in the model to the middle of their tolerance range, the reverse model recalculation is impossible. To save the original state of the drawing, you can use the command **UN: Undo changes** right after recalculating the dimensions to the middle of the tolerance range, or save the original and the recalculated models in different files.

The command can be called by one of the following means:

Icon	Ribbon
	Parameters → Tools → Recalculate to Middle of Tolerance
Keyboard	Textual Menu
<PN>	Parameters > Dimensions > Recalculate to Middle of Tolerance

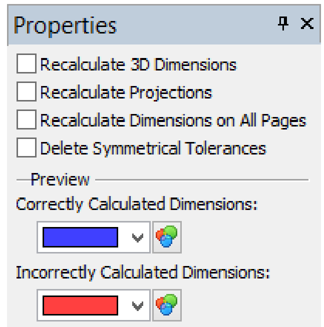
The following options become available in the command automenu:

	<End>	Recalculate
	<F5>	Preview Operation Result
	<Esc>	Exit

Dimensions are recalculated upon clicking the icon . Before that, you can review the expected result ofthe recalculation with the option .

In the command properties window there are flags that control the recalculation process. When all flags are disabled, then only the dimension on the current 2D page will be recalculated (except for the dimensions on 2D projections, if such exist on that drawing page). The flags in the properties window serve to make the following adjustments:

Recalculate 3D Dimensions. If this flag is set, then 3D dimensions will be recalculated along with 2D dimensions (and, therefore, with the 3D model).



Recalculate Projections. By having this flag set you allow recalculating the dimensions on 2D projections. If this flag is cleared, then the dimensions created on 2D projections are not recalculated. The projections themselves are not updated either (even if the 3D model changed as a result of recalculating 3D dimensions).

Please note that projections recalculation is also affected by the state of the flag **Recalculate Dimensions on All Pages**. If this flag is set, the dimensions are calculated on all 2D projections of the given model. If the flag is cleared, then recalculated are the dimensions of 2D projections located on the current 2D page only.

Recalculate Dimensions on All Pages. This flag allows performing recalculation of 2D dimensions on all pages of the given document.

As a result of recalculation, dimension tolerances become symmetrical. The manual method of defining tolerances will be set in the parameters of recalculated dimensions. The tolerance values can be assigned either symmetrical values resulting from the recalculation or zero values. The choice is made by setting/clearing the flag "**Delete Symmetrical Tolerances**". When the flag is cleared, the parameters of recalculated dimensions are assigned symmetrical tolerances, whereas when the flag is set – zero tolerances.

When previewing the recalculation result (the option ), the system highlights with different colors the dimensions that it successfully recalculated to the middle of the tolerance range, along with those that couldn't be recalculated, or those recalculated incorrectly. By default, the correctly recalculated dimensions are marked blue, the incorrectly recalculated ones – red. If necessary, you can specify your own highlight colors in the properties window.

RELATIONS

To quickly track geometrical dependencies in drawings and manage them, T-FLEX CAD has a special type of 2D elements – Relations. Relations serve to visually render on the drawing screen the types and parameters of geometrical dependencies between construction elements. Using the relations one can modify parameters of those dependencies without calling the editing commands of the respective construction elements.

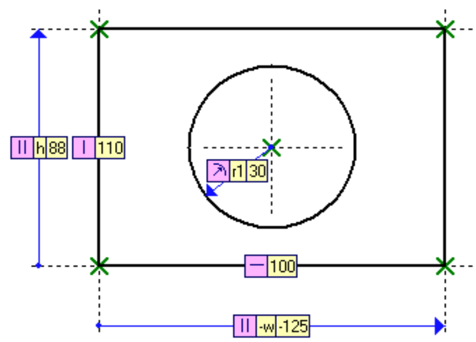
Relations are auxiliary objects that are displayed in the drawing field in the way of special marks. Those are not printed and are not exported. The information about the type of the geometrical dependency and the numerical parameter of the object, to which a relation pertains, is displayed on the relation mark. If a parameter is related with an expression or with a variable, then the relation mark displays both the expression and its current value.

Relations can be created automatically by the system ("temporary Relations") or manually by the user.

The automatic creation is done in the command for editing construction lines and graphic lines created based on construction lines. The system creates relations and displays them on the screen: if a construction line is edited – then for that line itself, if a graphic line is edited – then for the construction line, on whose base the graphic line is created. When exiting the editing command, such Relations are deleted automatically. Temporary Relations can be used to modify construction line parameters in the transparent mode.

Manual creation of Relations is done by the user in the command "**REL: Element Relations**". In this case, it is possible to create Relations either for individual construction elements or for all construction elements in a given drawing. Relations that are explicitly created by the user exist in the drawing up until the user deletes them by the same command. By default, those are permanently displayed in the drawing. Using those you can analyze parametric relations in the model, as well as modify construction line parameters. If necessary, all or specific existing Relations can be hidden from the current 2D window or completely over the entire document (such hidden Relations will not be displayed in any 2D windows opened for this document).

USING RELATIONS WHEN WORKING WITH DRAWINGS



Relations serve the two main purposes:

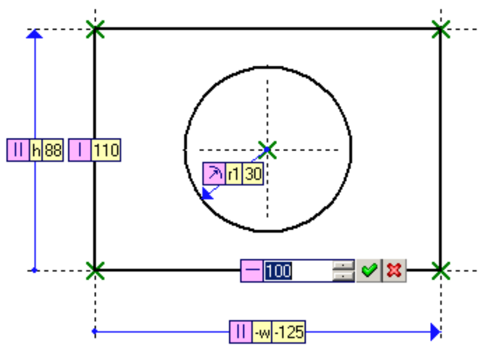
- Visualizing geometrical relations in the model;
- Modifying geometrical parameters of the model in the transparent mode.

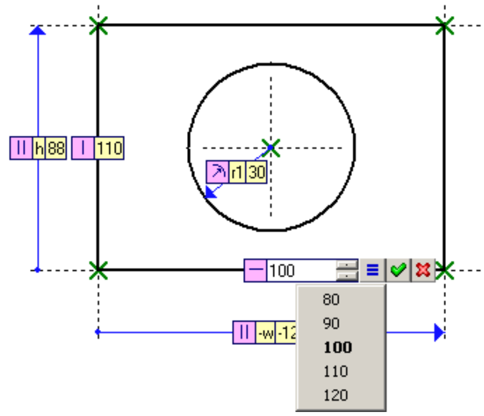
The first goal is achieved by creating Relations, when the user can visually examine geometrical relations without using the command "**Help|Information...**".

To modify an existing geometrical parameter using Relations, point the mouse at the parameter value in the Relation mark and click . The mark will turn to an edit box displaying the parameter value it controls.

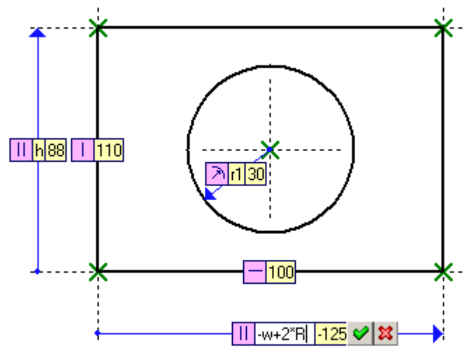
Just like in other system fields for editing values, the user can create here the list of frequently used parameter values and use it with the help of a special button for selecting the value from the list. This list is created with the commands of the context menu called by in the editing mode of selected Relation. "**Font**" command can modify font that will be used for displaying the Relation marks.

> Font parameters are common for all "marks": Relation marks, dimension marks used for editing dimension values in transparent mode, dragger marks in 3D operations.



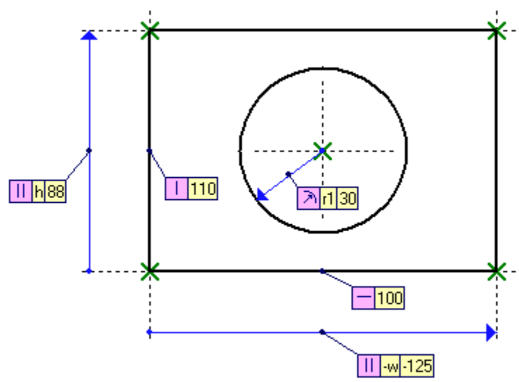


The new parameter value can be fixed by clicking the button on the mark itself or in the automenu.

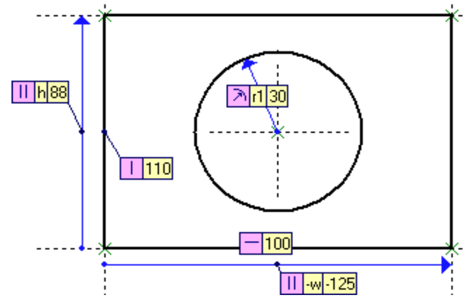


If the parameter described by a Relation is bound to an expression or a variable, then the Relation mark displays both the expression and its current value. The user can edit the expression itself in the same way as its value.

Sometimes, a Relation mark may obstruct working with the model by overlapping a portion of the drawing. To fix the situation, you can delete or hide from display such Relation (how to do it will be described a bit later), or simply move the Relation mark aside.



To move a Relation mark, point the mouse at the Relation icon. Once the cursor changes to , depress and, while holding the mouse button, drag the mark image to the desired position. The moved mark will be connected with the Relation image by a leader line.



You can move not only the relation mark, but also the Relation image that appears as an arrow connecting the 2D element, for which this relation is created, and its parent element. To do this, simply move the cursor to the Relation arrow and click . After that, the Relation image will rubberband along with the cursor. Move it to the new position and fix by clicking again.

CREATING RELATIONS WITH THE COMMAND "REL: ELEMENT RELATIONS"

A special command "**REL: Element Relations**" serves to create, as well as hide/show and delete existing, Relations:

Icon	Ribbon
	Parameters → Tools → Relations
Keyboard	Textual Menu
<REL>	Parameters > Relations

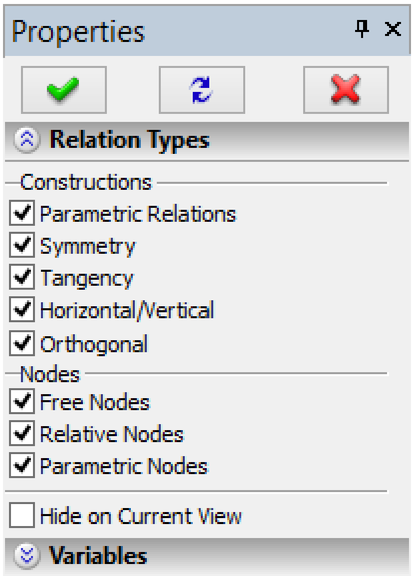
The following options are available in the command automenu:

	<*>	Create Relations for All Elements
	<Del>	Delete All Relations
	<C>	Create Relations for selected Elements
	<P>	Create Relations for parent Elements chain
	<D>	Delete Relations mode
	<Esc>	Exit command

There are three ways to create Relations:

1. Automatic creation of Relations for all construction elements in the current drawing. For this, you just need to push the option after starting the command;
2. Manual creation of Relations for individual construction elements with the option . After calling the option, select the construction elements in the 2D window, for which you need to create relations.
3. Manual creation of Relations for chains of dependent elements with the option . After calling the option, select the element in the 2D window, which will be the last in the chain. Relations will be created for the specified element through the entire chain of parent elements up to the base ones (those that are independent of other construction elements).

The created Relations will be always shown on the drawing, whether in the command waiting mode or inside any 2D command. Some of the existing Relations can be hidden from display, if you specify in the command properties window, which Relations shall be visible. This is done with the help of flags defining the visibility/invisibility of each Relation type (all flags are enabled by default):



Relations for *construction lines*:

Parametric Relations – Relations for construction elements that use geometrical parameters, which can be defined by variables. Such geometrical parameters include, for example, an offset line parallel to another line, an angle of a line inclined with respect to another line or to the horizontal, a circle radius etc. The exception is vertical and horizontal lines, for which Relations visibility is controlled by a special flag (see below).

Symmetry – Relations for lines constructed as the symmetry axis for two other lines.

Tangency – Relations for elements constructed with the tangency condition.

Horizontal/Vertical – Relations for vertical and horizontal lines.

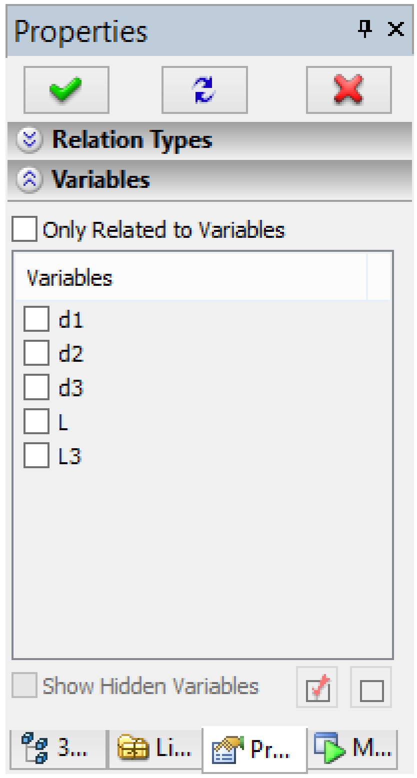
Orthogonal – Relations for the lines constructed as perpendicular to other lines.

Relations for *nodes*:

Free Nodes – Relations for the free nodes, meaning those defined by two coordinates - X, Y.

Relative Nodes – Relations for nodes defined by an offset relative to another node.

Parametric Nodes – Relations for nodes having one numerical parameter. Such nodes include the nodes on construction lines (circles, splines, functions etc.), as well as a node relative to another node on a line.



The flags in an additional **Variables** section serve to control the visibility of Relations depending on whether variables were used to define geometrical parameters of those elements (by default, those flags are disabled):

Only Related to Variables. When this flag is enabled, the 2D window will display the Relations only for those construction elements, which were defined using variables.

The list below shows all numerical variables in the current document. An additional flag **Show Hidden Variables** serves to display in this list also the hidden numerical parameters of the current document. Using the list, you can specify the variables that shall be considered when determining which Relations to show. To select a variable, you need to set the flag beside its name (using).

All variables in the list can be quickly marked with the button . Checks can be cleared off all variables in the list by the button .

After exiting the command, the Relations hidden by the above-described flags will stay hidden from the drawing. Nevertheless, they exist in the model. To make them visible, you need to call the **REL: Element Relations** command again and set the respective flags in its property window. You do not need to create Relations anymore in this case.

Hide on Current View. When enabling this flag, all created Relations become invisible in the current 2D window. However, those will be visible in other 2D Windows of the same drawing (if any are open). Just like in the case of using the previously described flags, to alter the relations visibility/invisibility once out of the command, you would have to call it again and change the status of the given flag in the property window.

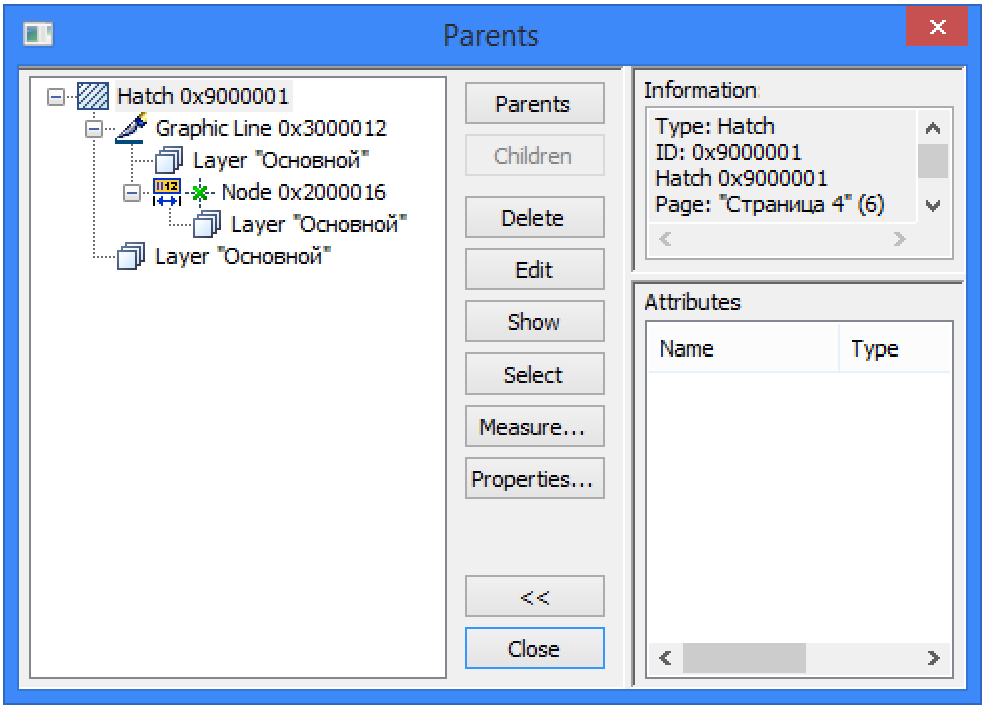
Relations are deleted with the options and . When clicking , all previously created Relations will be automatically deleted. The option is used to delete individually selected Relations. After calling this option you need to sequentially select any Relations to be deleted.

Managing Relations Visibility Outside "REL: Element Relations" Command

You can manage the visibility of created Relations even without calling the command **REL: Element Relations**. The "View" toolbar provides the  icon to call the special command **Show/Hide Relations**. The effect of this command is same as setting/clearing the "Hide on Current View" flag in the property window of the command **REL: Element Relations**.

Displaying Relations in the "Info" Command Window

When the "Info" dialog is activated for selected or all model elements, the relation objects are displayed in this dialog in a special way. An existing relation icon is displayed beside the element, to which it pertains.



Variables and Related Parametric Tools

VARIABLES

This chapter describes the uses of variables in T-FLEX CAD, the ways of defining parametric relations between the drawing elements and the idea of parameterization without programming. The variables allow extending the concept of parameterization on a deeper level. This chapter describes how to perform complex mathematical calculations within a drawing, how to define relations between construction entities, and other very useful capabilities of the system.

MAIN CONCEPTS

The T-FLEX CAD variables – are auxiliary elements of the system which enable to specify different types of non-geometrical interconnections between the elements of a drawing.

For example, the variables can serve as parameters of construction lines. In this case, the value of the construction line parameter will be determined by the value of the variable. If the value of the variable changes, then the value of the construction line parameter connected to it will be automatically modified (for example, radius of a circle or line location). With the help of variables, it is possible to assign color or visibility of the elements of a drawing, parameters of hatches, the text content, various parameters, etc. The variables can be also used upon creating a 3D model.

By assigning interconnections between the values of the variables, which determine parameters of the drawing's construction elements and drawing's image elements, it is possible to achieve automatic modification of the entire drawing when changes in the values of one or several basic variables are made.

Creating Variables

T-FLEX CAD system provides various ways of creating variables:

- using variable editor;
- while creating and editing construction line parameters and also other elements of the drawing or 3D model;
- using text editor;
- while defining textual strings for parameters of certain elements;
- while defining practically any of the numerical parameters of the system elements (levels, priorities, etc.)

The main tool for handling variables is the variable editor. It allows a user to perform any manipulations over the variables. Thus, we will commence describing the work with the variables exactly with the description of the variables editor. All other methods for creating variables will be described later, in the section “Using variables in the T-FLEX CAD”.

Variables Characteristics

Before getting to description of the variables editor itself and how to work in it, let's consider the main characteristics of any variable of the T-FLEX CAD.

Upon creating any variable of the T-FLEX CAD, it is necessary to indicate:

- a unique *name* of the given variable which enables to uniquely identify it in the document and also determine the *type* of the given variable (*text* or *real*);
- *expression*, based on which the system will calculate the current value of the variable.

Moreover, there is also a number of additional characteristics of variables, which can be specified in case of need. Several of them enable to impart additional properties to the variables (for example, the indication of *external variable*). Others are used exclusively for simplifying the work with a large number of variables in the document (*comment* of variable, *group* of variable).

Rules for assigning variables' names

The name of any T-FLEX CAD variable must represent itself a string of characters. The letters, numbers and the character "_" (underlining) can be used in the name. There is no limitation on the length of the variable's name.

The variable's name determines the *type* of the variable: *real* or *text*. The type of the variable shows what sort of values the given variable may take. The type is determined by the first character in the name of the variable. The name of the real variable must start with the letter, text variable – with the symbol *$*.

Examples of correct variable names:

VAR1; VVVVVVVVVV; VAR_1; $TEXT; WIDTH; width;

Note that the two last variable names are considered different, as the names are case-sensitive. Local language extensions of US ASCII are supported for the names. Local language users shall keep in mind that some language characters (particularly, Cyrillic and Greek) resemble the standard US ASCII, while their system codes may be different. Therefore, care should be taken in entering names, as the system will not recognize a name with the same appearance yet actually composed of different characters.

Examples of inappropriate variable names:

1_VAR (the first character is not a letter);

!_VAR! (inadmissible "!" character is used);

V A R (the name may not contain "space" characters).

Expression for variable

An *expression* is specified for each variable so that the system could calculate the value of the variable at any moment of time. An expression – is a mathematical formula, containing standard algebraic operations, logical operations, conditional operations, calls to mathematical functions and the T-FLEX CAD functions, various constants (real or character, depending on the type of variable), the values of other variables. As a result of the expression calculation, the value of the variable is obtained.

> The rules for composing expressions for the T-FLEX CAD variables and description of the functions that can be used inside the expressions are described in the Attachment I of this chapter.

Upon specifying the expression for the variable, the type of the variables should be taken into account. Real variables can take only numeric values (**12; 125; -234; 781.234; 3.834e+6**), text variables – only character values (“**Text**”; “**String**”; “**Name**”).

An expression can represent itself just a constant (numeric or character depending on the type of variable).

Variables-functions

Besides various mathematical and special functions, predetermined in the system, upon compose the expressions for the variables, it is possible to define and use user's own functions. For example, if, upon defining the variables, many similar, bulky expressions, differing only in separate arguments, are used in the expression, it is possible to define user's own function, a call to which can replace the expressions. User's functions are defined with the help of variables of a special kind – *variables-functions*.

Variable-function represents itself the definition of the user's function. The expression for the variable-function is composed according to the same rules as those for the standard variable. The entry of the arguments of the function into the expression is denoted in the following way (the number of the arguments is not limited):

#1 – the first argument,

#2 – and second argument and so on.

The function defined in such way can be used in the variables editor upon specifying the values of other variables. Upon calling this function, the names of the variables and numeric expressions serve as the arguments. The number of actual arguments must be equal or more than the number of formal arguments.

For example, if the variable-function was defined in the following way:

```
FUNC=(#1+#2)*10,
```

the call **FUNC (L, 20, 30)** will not be a mistake.

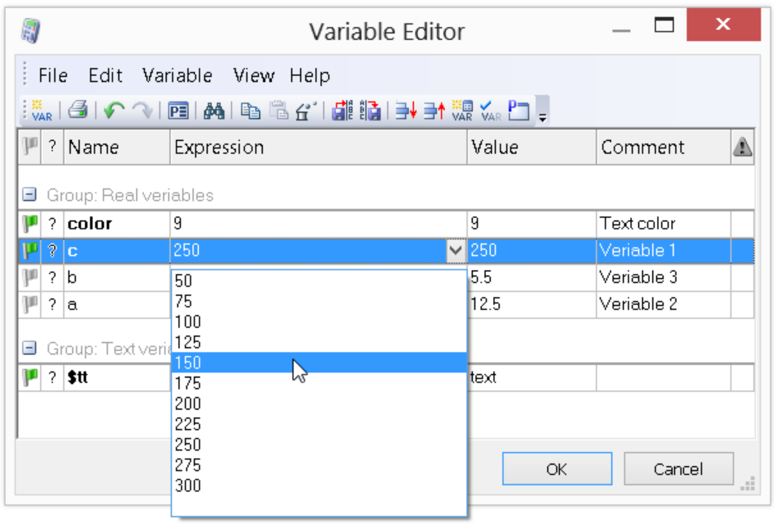
> In the list of the variables of the current document, the empty brackets “**()**” are automatically added to the name of the variable-function.

The list of variable's values

For any T-FLEX CAD variable it is possible to define a list of values. To be more precise – a list of expressions since the list for the variable can include any values, not necessarily the constants. The list of various expressions can be defined even for a variable-function. After that, the value of any variable, more precisely, of any expression, determining this value, can be selected from the created list.

The list is created as a set of lines, containing required constants or expressions. Moreover, the list can be created on the basis of already existing file, internal database, the list of materials in the 3D version or the calendar two last items are available only for text variables.

The list defined for the variable will appear in all places in which the value of the given variables is specified (in the variables editor, in the command **M: Edit model parameters**, upon assigning the variables of a fragment). The field for the entry of the values of such variable will include the graphic button which enables to call the list of values. To access the list of values, it is enough to point at this button with the cursor of the mouse and press . As a result, the list will pop up on the screen, and the new value (expression) can be selected from the list.



Comment of variable

If necessary, for any variable the *comment* can be specified. It represents itself an arbitrary text string. The comment enables to “attach” certain clarifications to the variable. For example, the comment may clarify the variable's designation (the width of the part, the radius of the circle, etc.) or the range in which the value of the variable is changed.

The comment of the variable, if it is specified, will appear in the dialog for inserting a variable (this dialog can be called, for example, from the context menu of all fields of the system dialogs), and also from the list of the values of the fragment upon its insertion into a drawing or 3D model.

Group of variables

If necessary, for any variable the *group* can be identified. When the variables are broken down into the groups, it becomes easier to control the large list of variables of the complicated drawing.

The *Group*, as well as the comment, is an additional characteristic of the variable specified for simplifying the work with a large number of variables. The fact that the variable belongs to a certain group does not affect in any way the use of this variable.

External Variables

Any variable, the value of which is specified by a constant (numeric or character), can be given an attribute *"external"*. External variables are used for organizing parametric connection between the assembly document and the fragments. The values of the external variables, defined in the fragment, can be modified from the assembly document.

The variables, marked as external, can be exported to the external text file with the possibility of reading from this file afterwards. This enables to use external variables for organizing connection between the T-FLEX CAD and other systems and application programs.

Hidden Variables

For regulating the work with a large number of variables, the mechanism of hidden variables can also be used.

Any variable, created in the T-FLEX CAD document, can be marked as *hidden*. By default, such variables are not displayed in the window of the variables editor or in the windows of other T-FLEX CAD dialogs dealing with the variables. Thus, it is possible to hide various auxiliary variables.

All standard templates of the T-FLEX CAD documents already contain a list of hidden variables providing automatic cross-reference between the fields of the drawing format (the title block) and the BOM data. In other words, the textual strings of these variables are substituted in both the appropriate title block fields and the respective entries of the BOM data.

Used and used variables

In order that the value of the variable could affect the structure of the drawing (or 3D model) of the given document, it is not sufficient just to create this variable in the T-FLEX CAD document. The given variable has to specify a characteristic of the elements of the drawing or 3D model: location of a line or a node, the radius of a circle, the level of visibility of the image line or 3D bodies, etc.

The variables the values of which take part in specifying characteristics of other elements will be further called *used*. Also, the variable is considered to be used, when its value is used for evaluating the value of another variable.

Consequently, the variables the values of which are not used anywhere on the drawing or in the 3D model and also upon calculation of the values of other variables are considered to be unused. Such variables, being equal variables of the T-FLEX CAD document, do not have any influence on its content.

WORK IN VARIABLES EDITOR

Window of Variables Editor

The work with the editor can be carried out in two ways. The first way – the work in the main window of the variables editor, called with the help of the command “**V: Edit variables**”. The dialog box of the given window enables to use the entire functionality of the variables editor and possesses a convenient interface. However, all changes made in the given window will be applied to the model only after closing the window of the dialog box.

The second way – is to use a special service window of the system – the window “Variables”. This window enables to work with the variables in the transparent mode.

In this chapter the description of work with the variables editor will be further presented by taking the standard window of the variables editor as an example. However, the same operations can be performed in the window “Variables”. The work with the window “Variables” will be discussed in the section “Working with variables editor in transparent mode” in a more detailed manner.

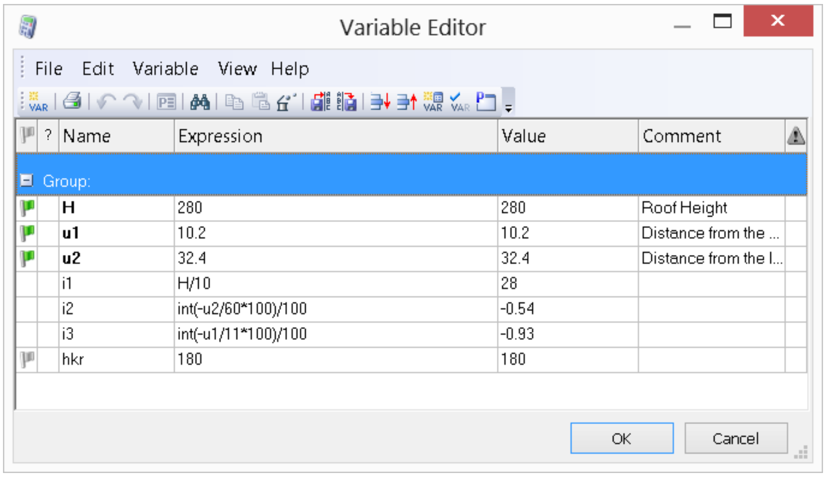
To call the main window of the variables editor, the command **V: Edit variables** is used:

Icon	Ribbon
	Title Block → Additional → Variables
Keyboard	Textual Menu
<V>	Parameters > Variables

The window of the variables editor contains the list of all variables of the current T-FLEX CAD document no matter in what way they were created (recall that the new variables can be created not only in the variables editor). Upon calling this command, the window of the variables editor will be empty if no single variable has been created in the document.

In the editor window, the variables list is displayed as a table, the form of which can be freely edited by a user. It is possible to modify the number and content of the displayed columns, the parameters for grouping and sorting the rows of the table, parameters of the grid of the table.

The variables editor has its own textual menu and the toolbar containing the main commands for working with the variables.

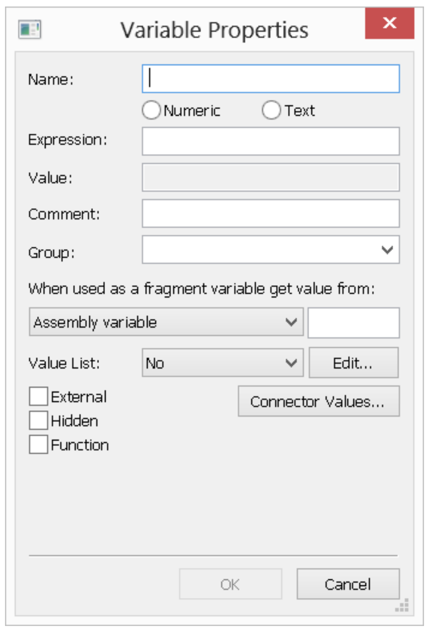


Creating Variable

In the variables editor the new variable can be created by using the command **New Variable**:

Keyboard	Textual menu	Icon
<Ctrl><N>	Variable > New	

After calling this command, the window for specifying properties of the variable being created appears. For creating a variable it is necessary to indicate the name, type of the variable (real or text), and also specify an expression, which will determine the value of the given variable.



The field "Name" and the toggle "Real/Text" work in a synchronized manner. For example, if the specified name of the variable starts with the symbol "$", then the type toggle is automatically switched to the value "Text". And vice versa – upon changing the type of the variable, the symbol "$" is automatically added/removed to/from the name of the variable.

Parameters **Comment** and **Group** do not have to be specified.

> By default, the parameter "Group" takes the same value as the variable selected in the table of variables upon calling the command **New Variable**.

The flag **External** is turned on if the variable being created has to be external.

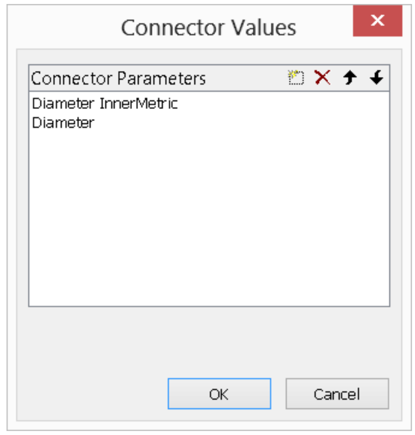
> In the table of variables the names of the external variables are marked with bold font.

The flag **Hidden** is turned on if the variable being created has to be hidden.

The flag **Function** is set on only in case it is necessary to create a variable-function. In this case the expression has to be made up with the use of the notation for the arguments of function (#1, #2, #3…).

If the current document is going to be used as a fragment, then for its external variables, the *assembly variable name* and/or the list "*connector values*". can be also indicated in the dialog "Variable's Properties". What that means will be discussed in detail in the chapter "Creating assembly drawing". The name of the assembly variable is defined in the field of a single-named parameter, and the list of connector values – in the dialog box "Connector values" emerging upon pressing the button **[Connector values…]**.

> The list "Connector Values…" is filled up for external variable of a document, used as a fragment with fixing by connector. Upon inserting such fragment into an assembly, the system has to automatically change the value of the fragment's external variable in accordance with the given ("values") of the indicated connector. The system selects the name of the required value of the connector in the list "Connector values" for the external variable. Upon fixing the fragment to the connector, the system will be first looking for the first name from the list, among the named connector values, and if it is not found – the second name and so on.



The list of connector values may contain arbitrary number of elements.

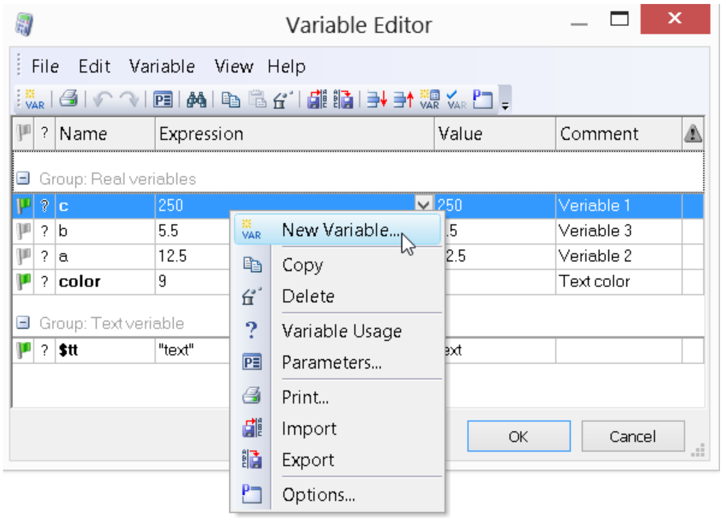
For creating a new element of this list, the button is used in this window, for removing already existing element – the button . The buttons and enable to move elements up and down along the list (the order of the elements in the list is set by a priority of elements in the list upon searching for the coincidences with the connector values).

The group of parameters **Value List** enables to create and edit the list of values for a variable. If a given variable does not have the list, then the value **No** will be specified in the drop-down menu of the given group. For creating the list it is necessary to select from the menu the required creation method: **Text**, **Database**, **File**, **Data**, **Materials**. More detailed description of various methods of creating the list of variable's values is presented below in the section "Creating List of Variable's Values".

After pressing **[OK]** the created variable appears in the list of variables.

> It is recommended to use uncomplicated names for the variables in order not to write lengthy expressions. It is a good thing to write a comment for each variable.

The command "**New Variable**" can be also called from the context menu at any place of the list of variables.



Besides the use of the aforementioned method, it is possible to create a new variable by other means. It is enough to put the name of yet non-existing variable into the expression of some variable. After recalculation of the given expression the system will find that such variable (for example, the variable "C") has not been defined, and a message will pop up on the screen: `"Create variable "C"?"`. If the question is answered positively, the new variable will automatically appear in the list of variables, and the focus of input will be moved to its field "Expression" – for specifying the expression for this variable. In case of a negative answer, the new variable is not created and the error message is generated.

Creating List of Variable's Values

The list of variable's values is created with the help of the parameter "Value List" in the variable's properties window. The method by which the variable's list is created is chosen from the pull down menu of this parameter: **Text**, **Database**, **File**, **Data**, **Materials**.

After selecting the list creation method on the basis of **text**, the window of the text editor will appear, in which the necessary list of values can be formed. Each value has to be located in a separate line. Upon creating the list, all options of the text editor become available.

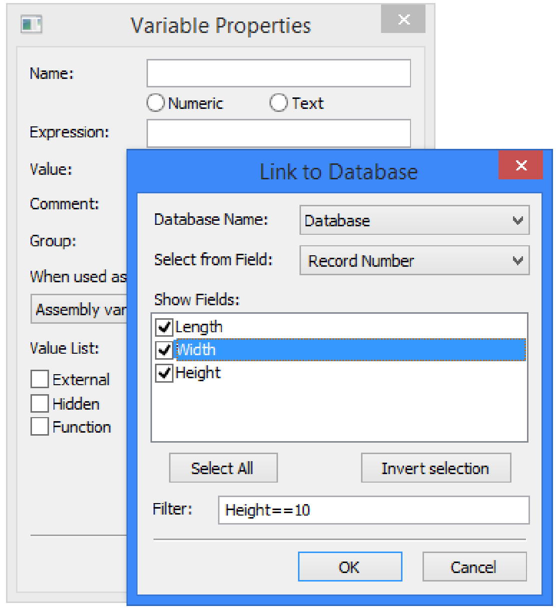
When the list is created on the basis of **database**, already existing internal database is used. For example, let's suppose there is database shown on the picture. After calling the command for creating the list on the basis of database, the dialog window will appear. The parameters for forming the list have to be indicated in this window.

№	Length	Width	Height
1	10	20	10
2	20	25	10
3	40	30	15
4	75	25	20

One has to choose:

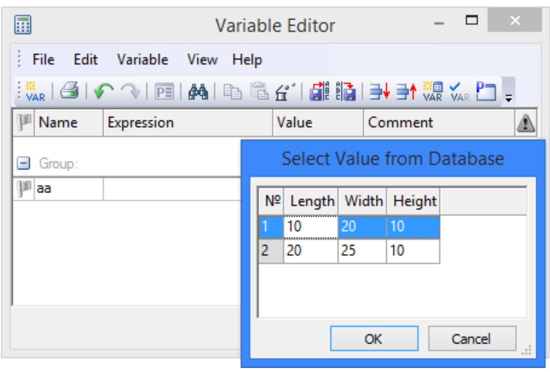
- the name of the database from the list of databases for the current document;

- the column in the database from which the values will be selected. The first line of the list – "Record number" can be also chosen. In this case, the selected record number will be a value being returned;
- columns which will appear upon creating the list.

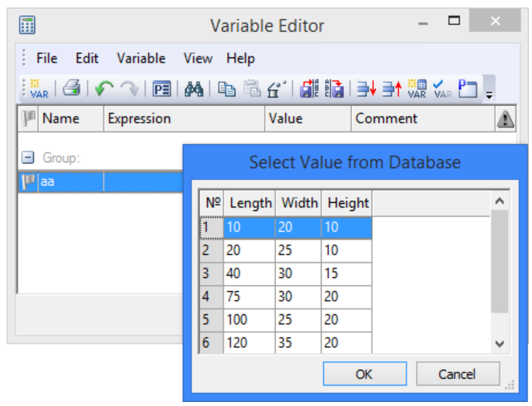


The field "**Filter**" enables to specify conditions for the values selected from the database (upon creating the list of values). These conditions are specified with the help of logic expressions, which are made up by the same rules as those for the expressions for the variables (see Attachment I to this chapter).

For example, the use of the expression, shown on the picture above, will lead to the result shown on the picture below (for comparison, the list obtained in the same example without specifying the condition in the field "Filter" is shown as well).

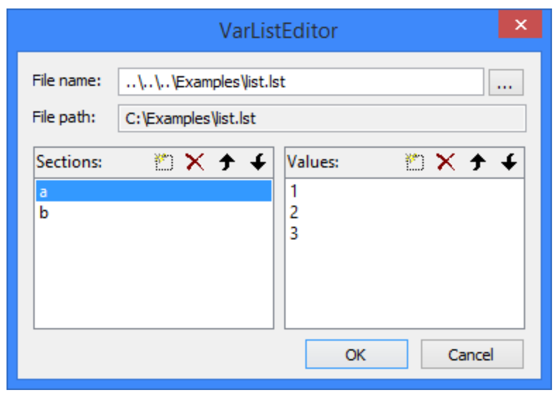


List on the basis of database obtained with the use of filter



List on the basis of database obtained without using the filter

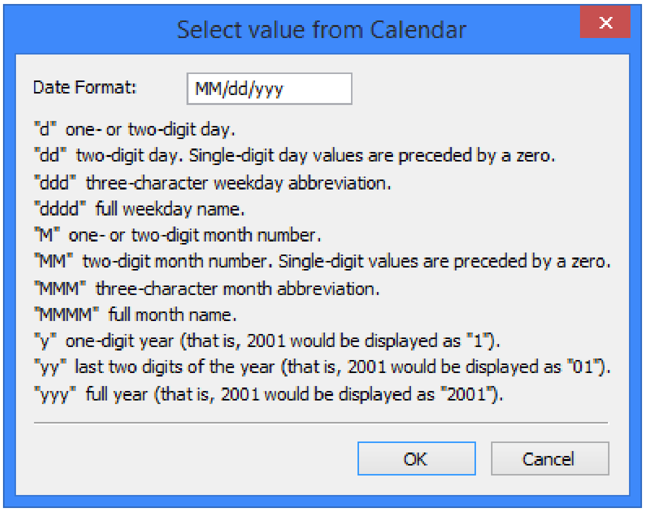
When the list is created on the basis of **file**, the dialog window for selecting already existing or creating the new file appears. The data in the file must be stored in the form of sections with the lists of values. Upon creating the list, the required section is indicated. Also, it is possible to create the new section or remove already existing section from the file with the help of buttons  and  (buttons in the left pane of the dialog).

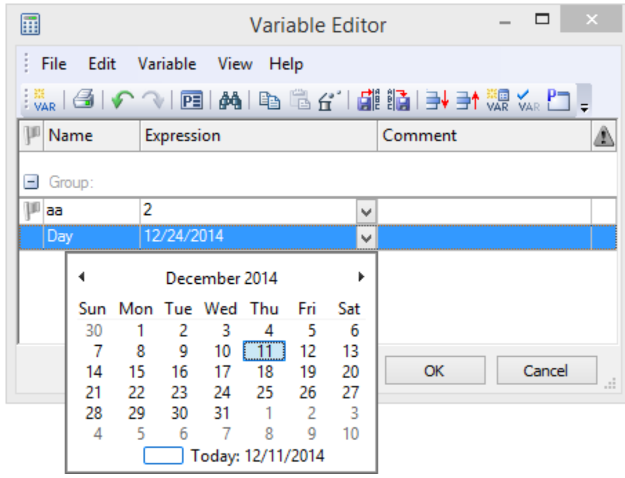


The list of values for the selected section is shown in the field on the right. The buttons , ,  and  in this pane of the window enable to edit the list of values of the selected section.

The created file can be used while working with other T-FLEX CAD documents.

When the list is created on the basis of **date**, the dialog window pops up, in which the date representation format can be specified, for example, “DD.MM.YY”. The variable, for which this list is specified, has to be a text variable. After that, upon making a selection from the list, the window in the form of a calendar will emerge, in which any required date can be chosen.





Also, it is possible to form the list **on the basis of materials list** (only in 3D version of the system). Such list can be formed only for the **text** variable. Upon creating the list, the window of the text editor appears, in which all materials, used in the 3D model of the current document, are included by default. If necessary, the list can be edited manually.

In order to edit the created list, it is necessary to use the button **[Edit...]** of the group “Value List”. For each type of the list, the corresponding edit method will be called.

For removing any list of values, it is enough to put again the value “No” in the drop-down list of the group “List”. If the list is created on the basis of database, then only connection with the database will be broken, the database itself will be preserved.

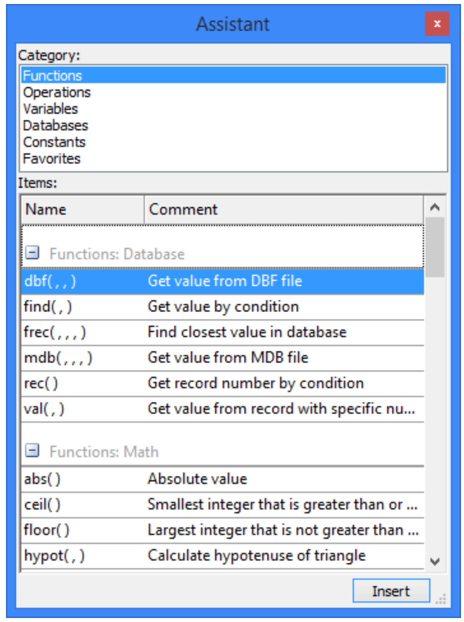
Window “Assistant”

To specify the values of the variables, the user can use a special auxiliary window “Assistant”:

Keyboard	Textual menu	Icon
<Ctrl><H>	View > Assistant	

Assistant is used for quick insertion of functions' names, records from the databases, variables already existing in the document, frequently used constant, etc. into the expressions of variables

For convenience the contents of the Assistant is divided into several categories. The list of available categories is displayed in the upper part of the Assistant window in the field **Category:**. The contents of the category selected in the field "Category" is displayed in the lower part of the window. For example, on the picture to the right the list of standard mathematical function of the T-FLEX CAD is shown in the Assistant window. For each function there is a short description.



To insert a function (a link to a field in the database, an operation, a variable, etc.), the user needs to select a desired category in the Assistant window, then select a required line in the category contents list and finally press or button **[Insert]**. The item is inserted in the main window of the variables' editor into the space corresponding to the position of the cursor before activating the Assistant window.

When working with the variables' editor, the window "Assistant" can be placed on the screen permanently.

Properties of Variable

For changing the name, expression or other characteristics of variables, the variable's properties dialog box, called with the command **Properties**, is used:

Keyboard	Textual menu	Icon
	Variable > Properties ...	

The command **Properties** can be also called from the context menu, activated with the upon choosing the variable in the list of variables. Or it can be called just by pointing with the cursor at the variable's name in the list of variables and pressing .

After calling this command, the window of the dialog **Variable's Properties** will appear. Inside this window it is possible to modify the *name* of the variable, its *expression*, *comment* and *group*, by using corresponding fields of the dialog. It is possible to mark this variable as external or hidden (flags **External** and **Hidden**).

The command **Properties** can be also accessible when several variables are selected simultaneously. In this case, when this command is called, the window emerges in which for selected variables the only parameter: *group* can be specified. Selection of several variables is carried out with the help of <Ctrl>+, <Shift>+.

Removing Variable

It is possible to delete the variable with the help of the following command of the variables editor:

Keyboard	Textual menu	Icon
	Edit > Delete	

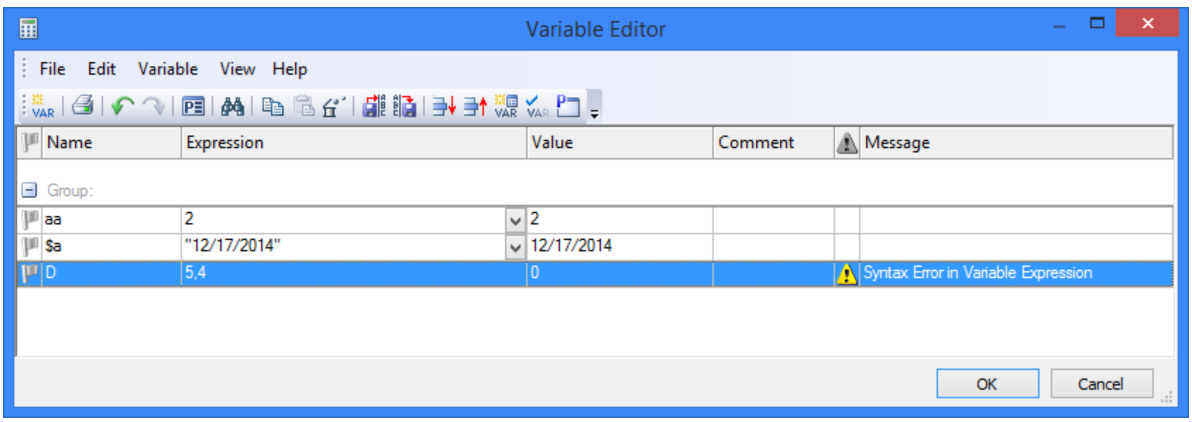
The command **Delete** can be also called from the context menu upon choosing the variable in the list of variables.

After calling this command, the variable, for which this command was called, will be removed. Note that, only unused variables can be removed (in the column "Not used" a symbol "?" will be standing next to such variable). The command **Delete** is not available for used variables. In case if the variable is currently used, the user has an option of either deleing the chain of dependent elements or replacing the selected variable with a constant value.

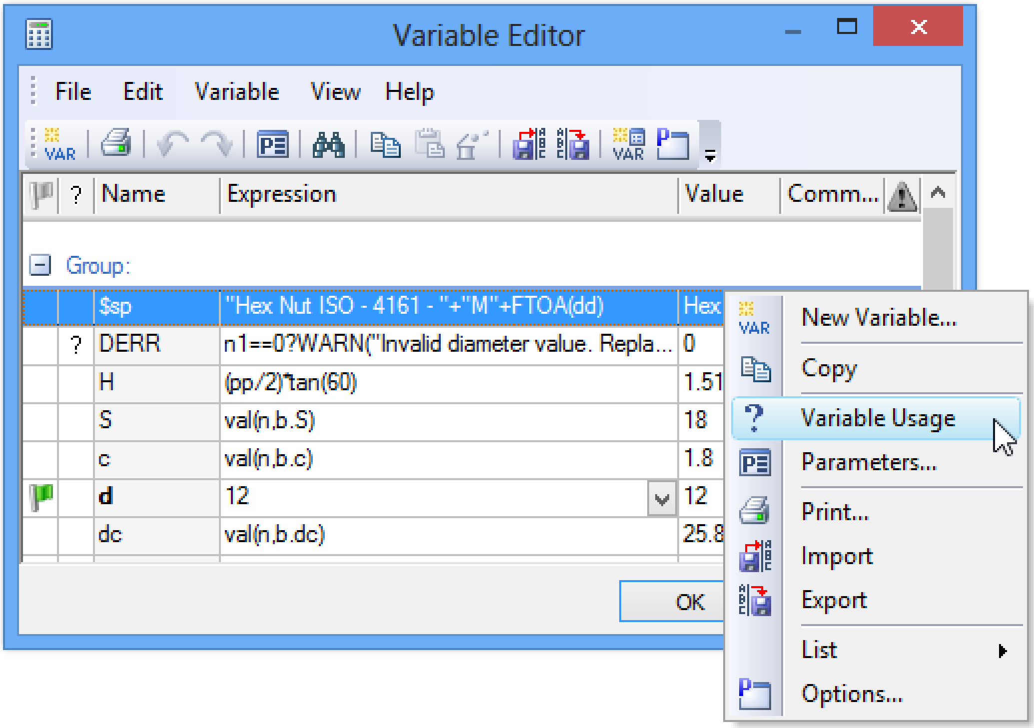
Diagnostics of Errors

Upon creating the new variables and also the further work with the variables, various errors can arise. Usually these are the syntax errors in the expression of the variable. In this case, for the problem variable the sign ⚠ will be shown in the column "**State**". Upon bringing the cursor to this sign, a tooltip about the type of the error will appear. At the same time, the color of the field **Value** will be changed to red manifesting the existence of the error. In addition, for the given variable a detailed description of the arisen error will appear in the column **Message** (by default this column is turned off).

In spite of the existence of errors, it is possible to finish the work in the variables editor, and get back to their correction later. The existence of errors in the variables editor does not affect the recalculation of the 2D/3D model elements of which depend on the incorrect variables.



If necessary, you can track the use of any variable in the model with the help of the "The use of a variable" command found in the context menu of the variable.



Canceling Operations in Variables Editor

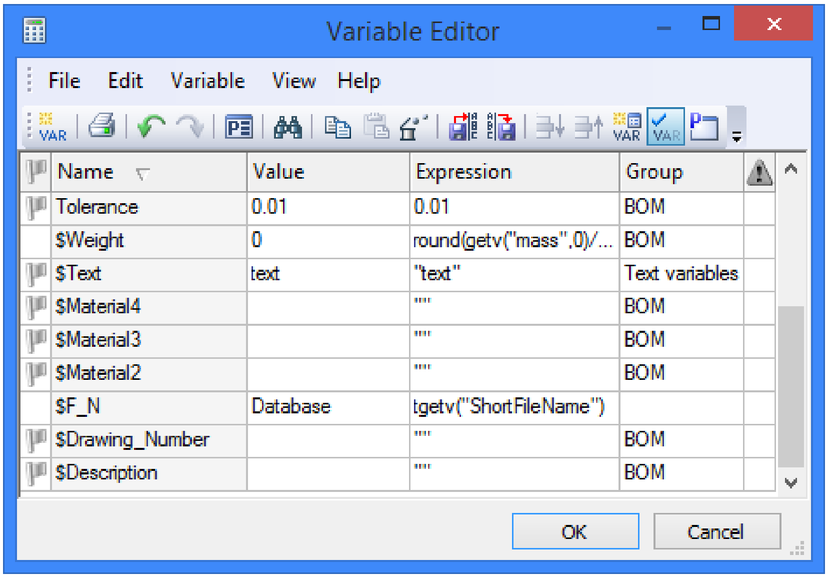
Any actions in the variables editor can be canceled/repeated with the help of step-by-step commands:

Keyboard	Textual menu	Icon
<Ctrl><Z>	«Edit > Undo	
<Ctrl><Y>	«Edit > Redo»	

The number of cancellation steps is limited only by general setting of the system (parameter "Undo/Redo Buffers" in the dialog of the command **Customize** > **Options...**, the tab "Preferences").

Show/Hide Hidden Variables

Command «Show/Hide hidden variables»  shows or hides hidden document variables in one click. State is maintained in user settings.

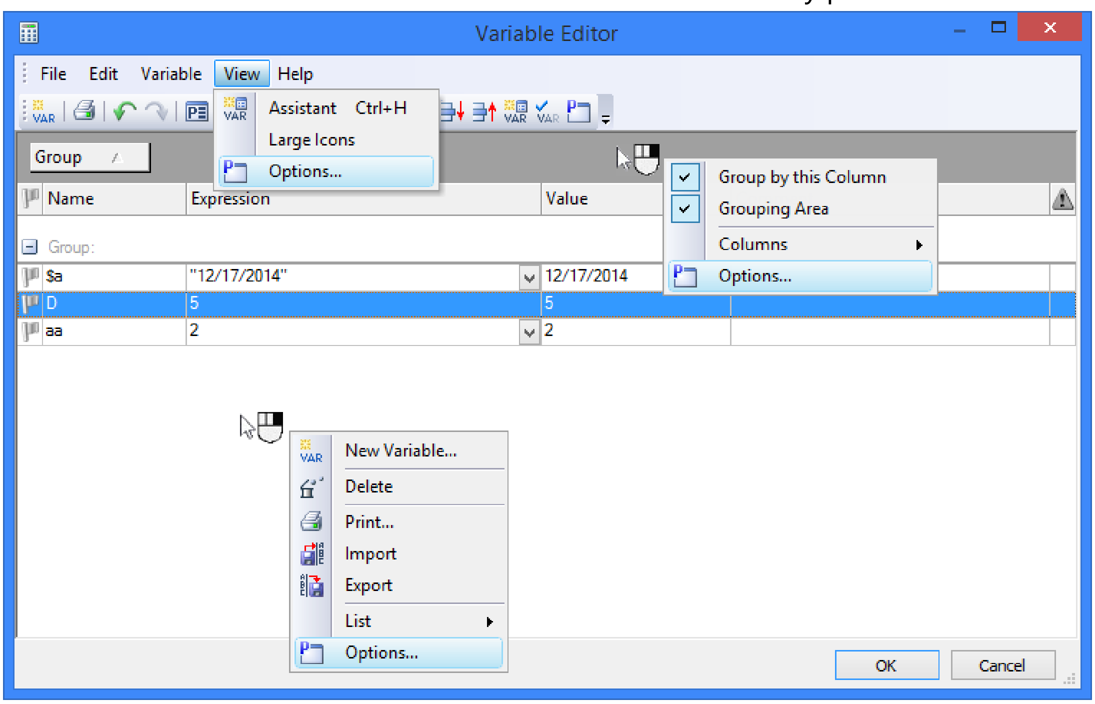


Customizing Window of Variables Editor

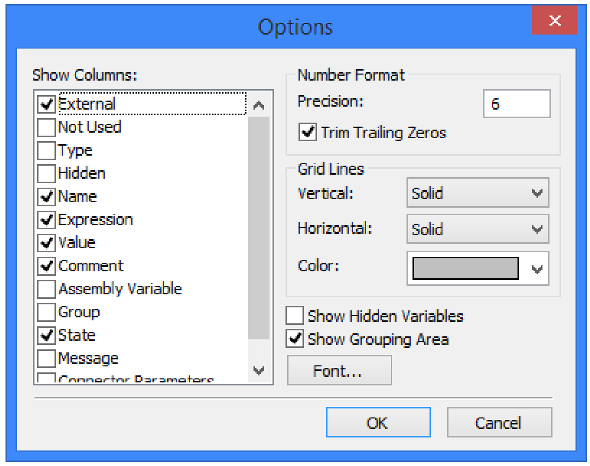
The appearance of the table of variables in the window of the variables editor can be customized with the help of command:

Keyboard	Textual menu	Icon
	View > Options...	

Moreover, this command can be also called from the context menu at any place of the variables editor.



After calling this command the window **Options** opens up. At the left pane of the window, the list of all possible columns from the table of variables will be displayed.



The majority of the columns correspond to some characteristic of the variable (**name**, **type**, **usability**, **expression**, current **value**, and so on). The columns **State** and **Message** are used for output of system messages about errors upon evaluation of the value of variable.

The columns displayed at the current moment are marked with a tick before the name. To add the column into the table, it is enough to select it in the list and with the help of put a tick before its name. For removing a column from the table, it is sufficient to take off a tick next to its name.

In the right pane of the window "Options" there are other various parameters of the table of variables:

The group **Number Format** sets the format for the real numbers in the column **Value**: **Precision** and **Trim Trailing Zeros**.

The group **"Grid Lines"** defines the appearance of the grid of the table of variables:

Vertical. The appearance of vertical lines of the grid of the table: **No**, **Small dots**, **Large dots**, **Dashed**, **Solid**.

Horizontal. The appearance of horizontal lines of the grid of the table: **No**, **Small dots**, **Large dots**, **Dashed**, **Solid**.

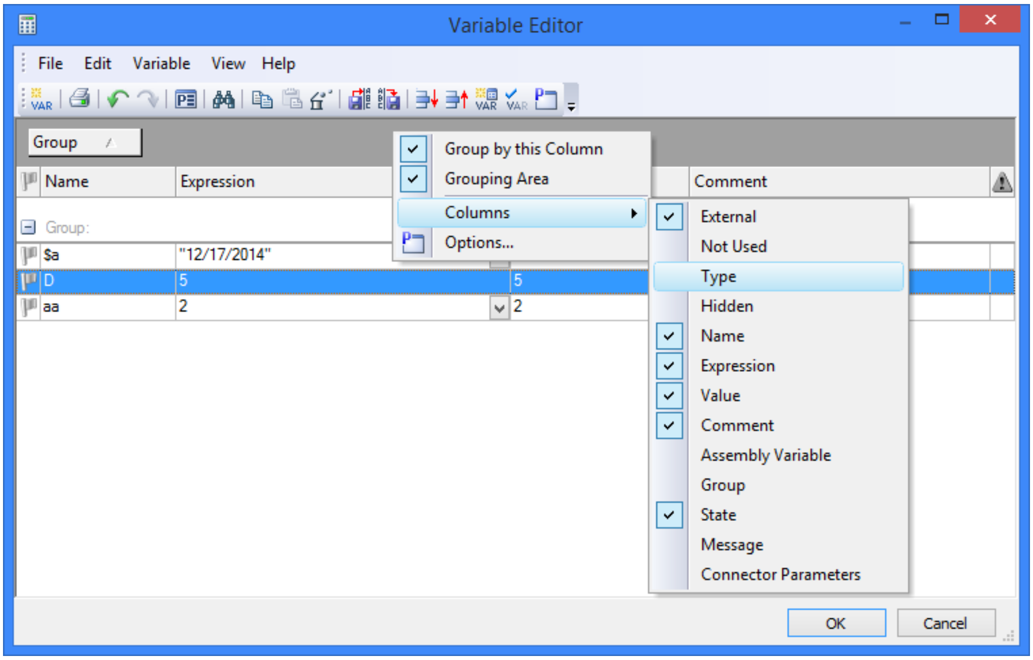
Color. The color of grid lines of the table of variables.

Show Hidden Variables. This flag controls the view of hidden variables in the table of the variables editor.

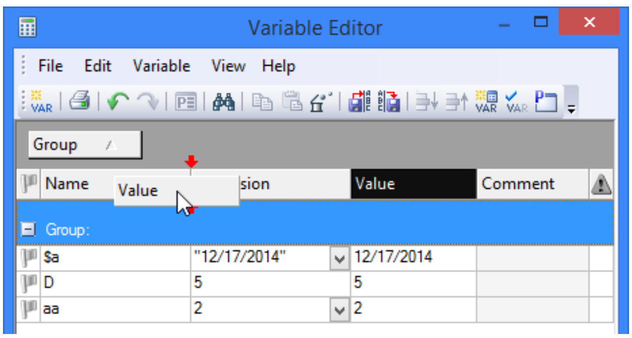
Show Grouping Area. This parameter controls the view of the **grouping area** over the table of variables (see below).

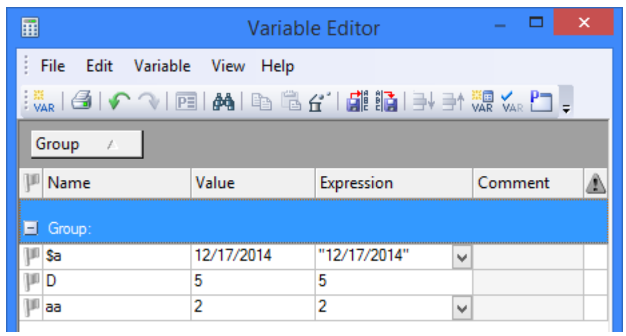
The button **[Font...]** enables to set the font used upon displaying the table of variables in the window of the variables editor.

The visibility of single columns of the table can be customized without calling the command **Options**. To do that, it is enough to call the context menu from any place on the header of the table of variables. Submenu **Columns** enables to quickly turn off/turn on the display of the columns.



The order in which the columns are displayed in the table of variables can be easily changed just by pulling over the columns into the required places. To do that, it is enough just to bring the cursor to the column, hit and without releasing the mouse button, drag the header of the column to the required place. Red arrows on the screen suggest where the column will be inserted.

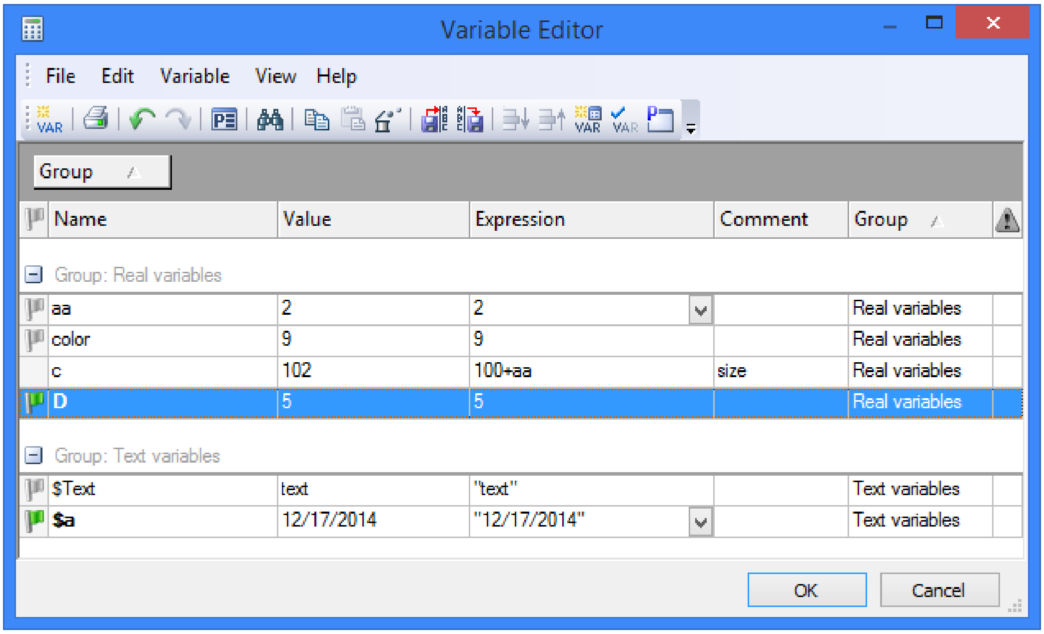




Grouping Area. Grouping of Variables

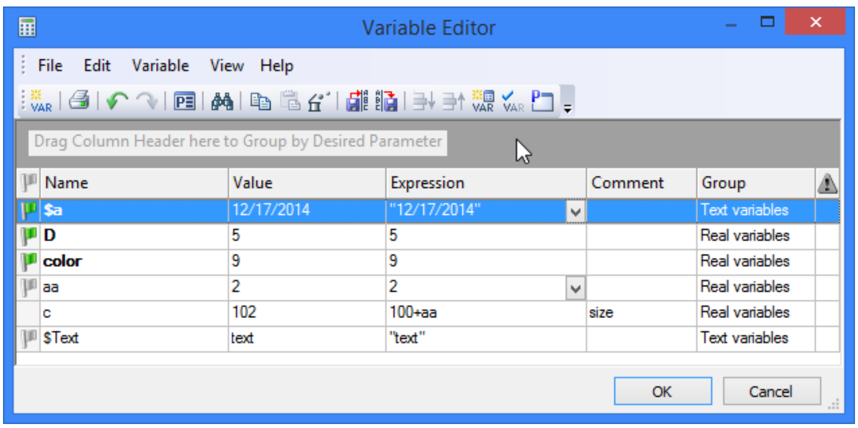
By default, in the variables editor all variables are grouped by the characteristic “group” (if this characteristic is specified for variables of the current document). However, it is possible to use other parameters of variables for grouping, including several parameters at the same time.

For customizing the grouping parameters it is convenient to use the **grouping area**. It is turned on via the dialog of the command **View > Options…** or via the context menu in the header area of the table of variables. The grouping area is situated over the table of variables. By default, the label “Group” is displayed in the grouping area. This means that the variables are grouped by the characteristic “group”.

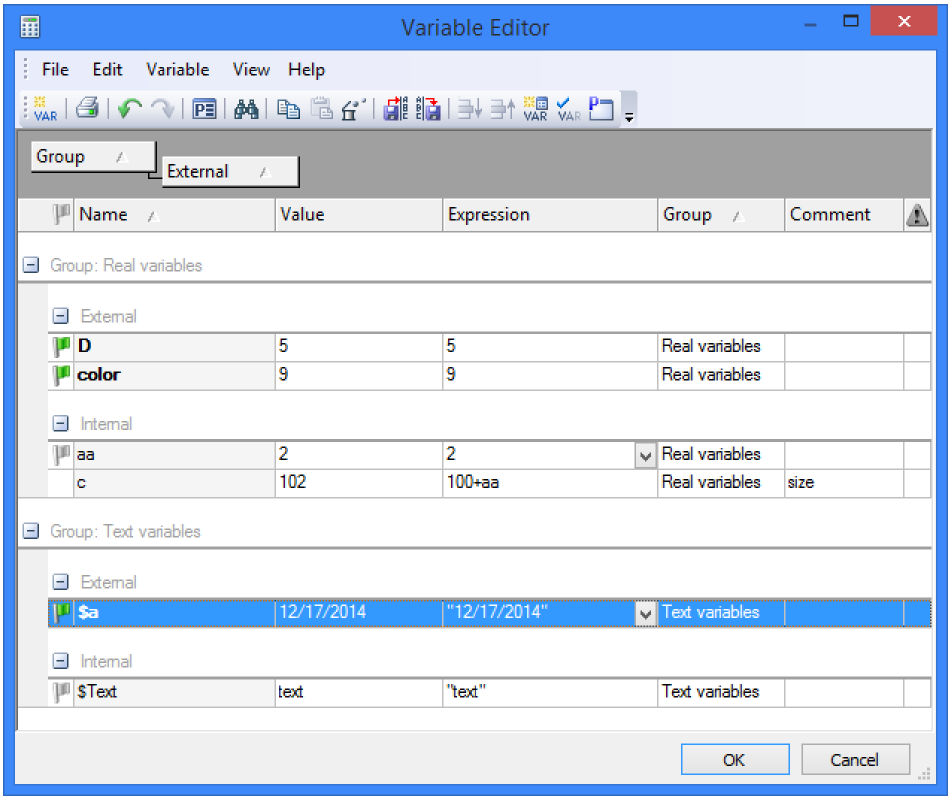


In order to turn off the grouping, it is enough to point with the cursor at the label in the grouping area, press and without releasing the mouse button, drag the label to any other place outside the grouping area. If the label is moved to the headers' bar of the table of variables, the column with the same name will be added to the table. If the label is moved while holding the key <Ctrl>, the label will be copied – it will remain in the grouping area, and at the same time the corresponding column will appear in the table.

When the mode of grouping is turned off, the message **Drag Column Header here to group by Desired Parameter** is displayed in the grouping area.

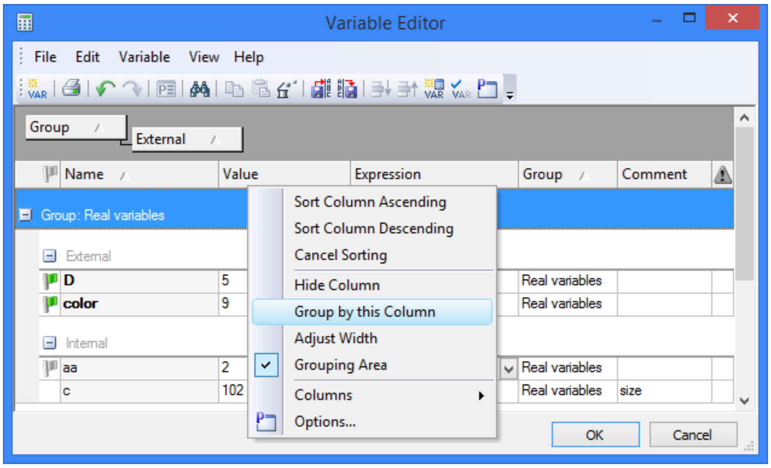


To activate the mode of grouping by some characteristic, it is enough to point at the header of the corresponding column of the table of variables, press and without releasing the mouse button, drag the header of the column to the grouping area. After that, the label of the chosen characteristic will appear in the grouping area, and the variables in the table will grouped by this characteristic. If the headers of two columns of the table are moved to the grouping area, the grouping will be carried out by two characteristics simultaneously. Location of the labels in the grouping area shows the order of grouping.



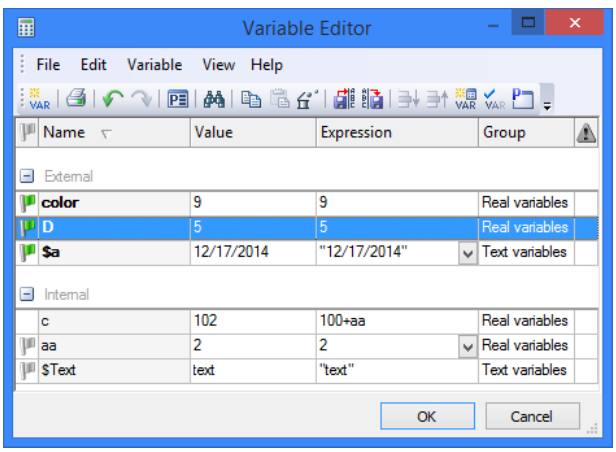
Note that upon moving the header of the column into the grouping area, the column is removed from the table. In order to keep this column in the table, it is necessary to hold the key <Ctrl> while moving the header.

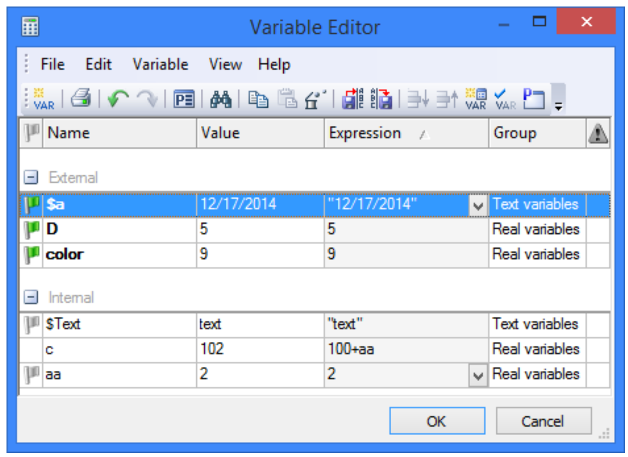
It is also possible to control grouping with the help of the context menu. It is sufficient to point the cursor at the column header of the table of variables and call the context menu with the help of . The flag **Group by this Column** will be accessible in the context menu. In order to activate grouping by this column, this flag should be set on. To cancel grouping by selected column, it is enough to take this flag off.



Sorting Variables

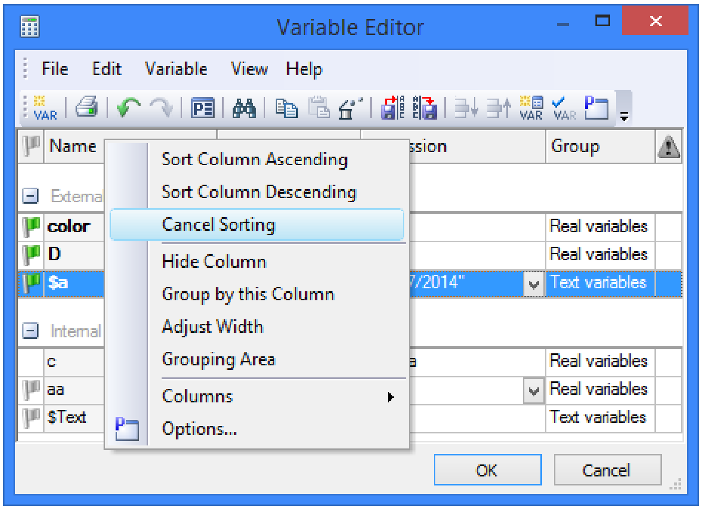
In addition to grouping, for the list of variables it is possible to specify sorting by any column. By default, the sorting is turned off. To turn it on, it is enough to bring the cursor to the header of that column of the table of variables by which the variables are to be sorted. The tooltip **Sorting by: ...** with the name of the selected column will appear on the screen. If one hits pointing at the header of the column, then the rows of the table of variables will be sorted by the selected characteristic. In the header of the column, by which the sorting is carried out, an additional symbol in the form of a triangle will appear, showing the direction of the sorting: – for sorting *in ascending order*, – for sorting *in descending order*.





By default the mode of sorting in an ascending order is turned on initially. Pressing on the header of a column repeatedly turns on the sorting in a descending order. For sorting by another characteristic, it is enough to press on the header of the corresponding column.

Sorting can be canceled with the help of the command “**Cancel Sorting**” in the context menu called from the headers' bar of the table of variables (i.e. in the area of the header of any column of the table). This command is available only when the mode of sorting is turned on. Also, in the context menu the commands **Sort Column Ascending** and **Sort Column Descending** are available. They activate sorting by the column for which, in the column header, the context menu was called.



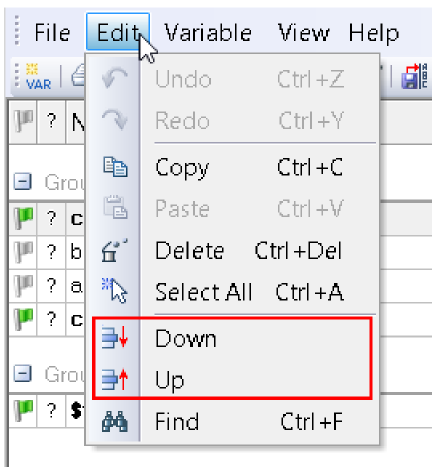
Sort parameters are saved in the file of the document and restored upon reentry into the variable editor.

When sorting capability is turned off the user can also utilize <Ctrl>+<ArrowUp>, <Ctrl>+<ArrowDown> to change the order of the lines in the variable editor. To change the order of rows in a table the buttons can also be used:

 - Move downwards.The current row will be moved to the row below.

 - Move upwards.The current row will be moved to the row above.

These commands are also accessible via the text menu of the variables' editor:

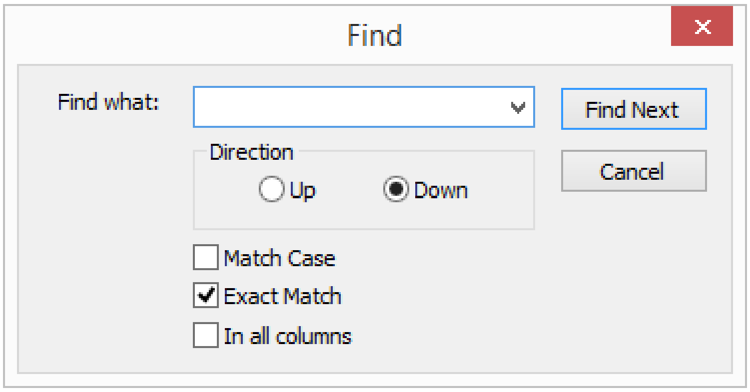


The changes in the sequence of the lines are saved in the document's file separately for the main window of the variable editor and for the auxiliary window "Variables" (see section "Working with variable editor in transparent mode").

Finding Variables

Upon working with a large list of variables, it is sometimes convenient to use the command of searching for variable by name:

Keyboard	Textual menu	Icon
	Edit > Find	



After calling this command the dialog window pops up, in which the search parameters are required to be specified. After specifying the parameters, the button **[Find Next]** has to be pressed. If the search process was completed successfully, the cursor is moved to the column **Expression** for the found variable. If the variable was not found, the cursor stays at the same place, and in the message line of the editor the message appears: **Cannot find specified string**.

Upon specifying parameters it is important to pay attention to the state of the flag **Match Case**. By default, this flag is activated. In this case the system looks for the variable, the name of which coincides exactly with the specified string in the field “Find what”. When this flag is off, the system searches for the variable for which the text, specified in the field “Find what”, enters the variable's name as a substring.

In all columns. The option allows finding of the specified string in all columns.

Copying Variables

The T-FLEX CAD makes it possible to copy variables from one document to another with the help of the clipboard.

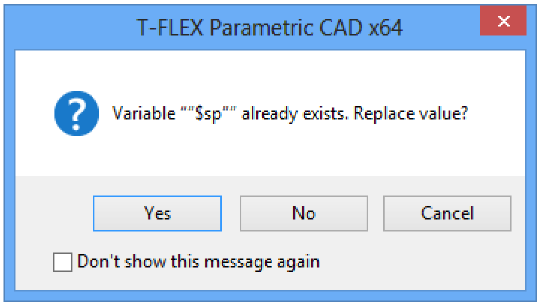
For copying one variable to the clipboard it is necessary to select it in the table of variables with the help of , and after that call the command **Copy**:

Keyboard	Textual menu	Icon
<Ctrl><C>	Edit > Copy	

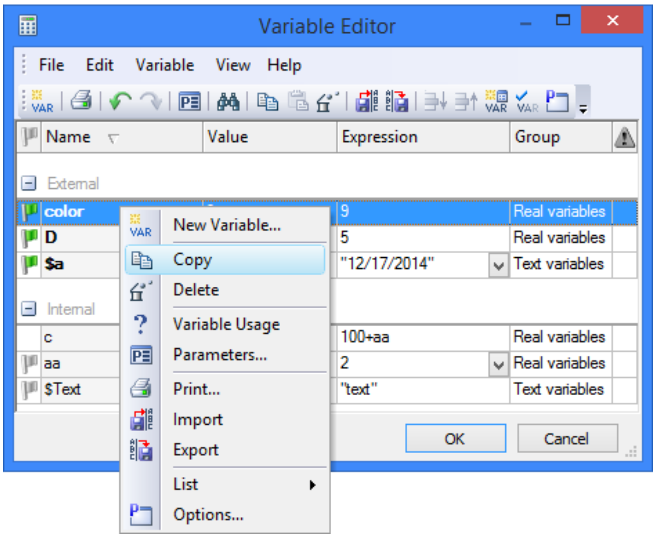
For inserting the already copied variable, the command **Paste** is used:

Keyboard	Textual menu	Icon
<Ctrl><V>	Edit > Paste	

If the document, in which the insertion is performed, already contains the variable with the same name as that of the variable being inserted, the message **Some Variables have equal Names** pop up.



The commands of copying/insertion can be also called from the context menu for the variable:



For copying several variables simultaneously, a multiple selection with the help of <Shift>+ and <Ctrl>+ is used. For selecting all variables of the given document at once, the following command is used:

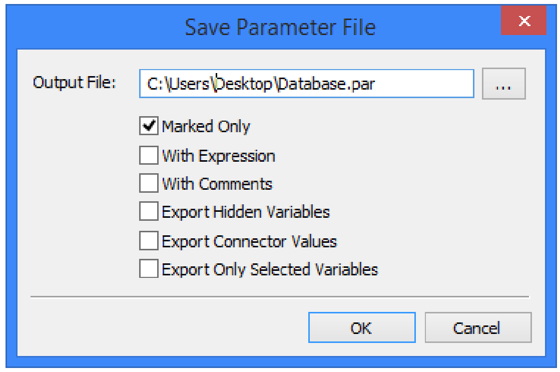
Keyboard	Textual menu	Icon
<Ctrl><A>	Edit > Select All	

Writing Variables to External File

The values of variables can be written into the file of parameters by using the command **Export Parameters**:

Keyboard	Textual menu	Icon
	«File > Export»	

Upon calling this command the dialog window appears. One needs to specify the name of the file, into which the information will be written. By default, the filename coincides with the name of the current drawing, file extension is – "par". It is possible to specify an arbitrary filename.



Each variable is written in a separate line. The format of the record is the following:

<name of variable> = <value> [/*<comment >*/]

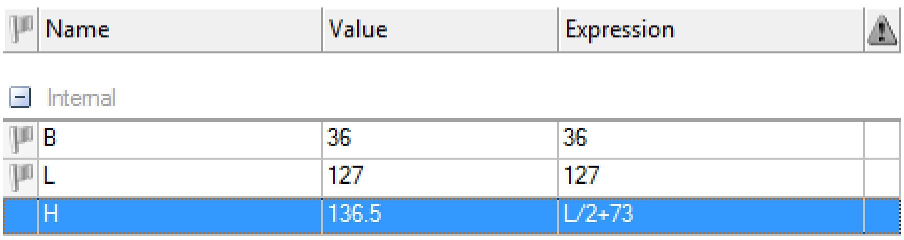
The name and the value of the variable are always written down.

The comment is written on condition that one of the following parameters has been set on: **With Expressions** or **With Comments**.

If in the export dialog box the flag **Marked Only** is activated, then only external variables of the given document will be written into the resulting file. When the flag is taken off, all visible variables are exported.

Hidden variables are not exported by default. In order to write them into the external file as well, it is necessary to set on the flag **Export Hidden Variables**.

For example, suppose there is the following list of variables in the editor:



Upon writing the file, the following parameters have been activated: output all variables except hidden ones, with expressions and comments. This has to result in the file with the following content:

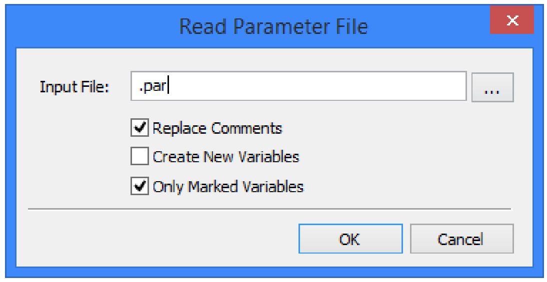
```
L = 127 /* 127 ; Length */
B = 36 /* 36 ; Width */
H = 136.5 /* L/2+73 ; Height */
```

If you select **Export connector values**, the output file will have values added from the corresponding column.

If you select **Export only selected variables**, the output file will have only selected rows.

The values of external variables can be read from the file of parameters by using the command **Import Parameters**:

Keyboard	Textual menu	Icon
<Ctrl><R>	File > Import	



After calling this command the dialog window pops up.

If the variable present in the file of parameters is absent in the current drawing, it will not be read. Also, the variables that are not defining in the drawing as external will not be read.

How can files of parameters be used?

Files of parameters can be used when you need to save several versions of the same drawing. In this case you save several files of parameters with the values of external variables and, if necessary, read the values from a specific file. As a result, a finished drawing with the required parameters is obtained.

It is convenient to use the files of parameters for connection of the T-FLEX CAD with other computational software. In the system you can create a parametric drawing with certain set of parameters. Your computational procedure receives the values of these parameters through the file. You calculate the remaining parameters of the drawing in your software and create either a new file of parameters or update the old one. From the T-FLEX CAD you read the file and obtain a modified drawing on the basis of the parameters calculated by you. Thus, the file of parameters serves as an intermediate link for connection between the T-FLEX CAD and your software.

Printing List of Variables

For printing the content of the variables editor the command **Print** can be used:

Keyboard	Textual menu	Icon
<Ctrl><P>	File > Print…	

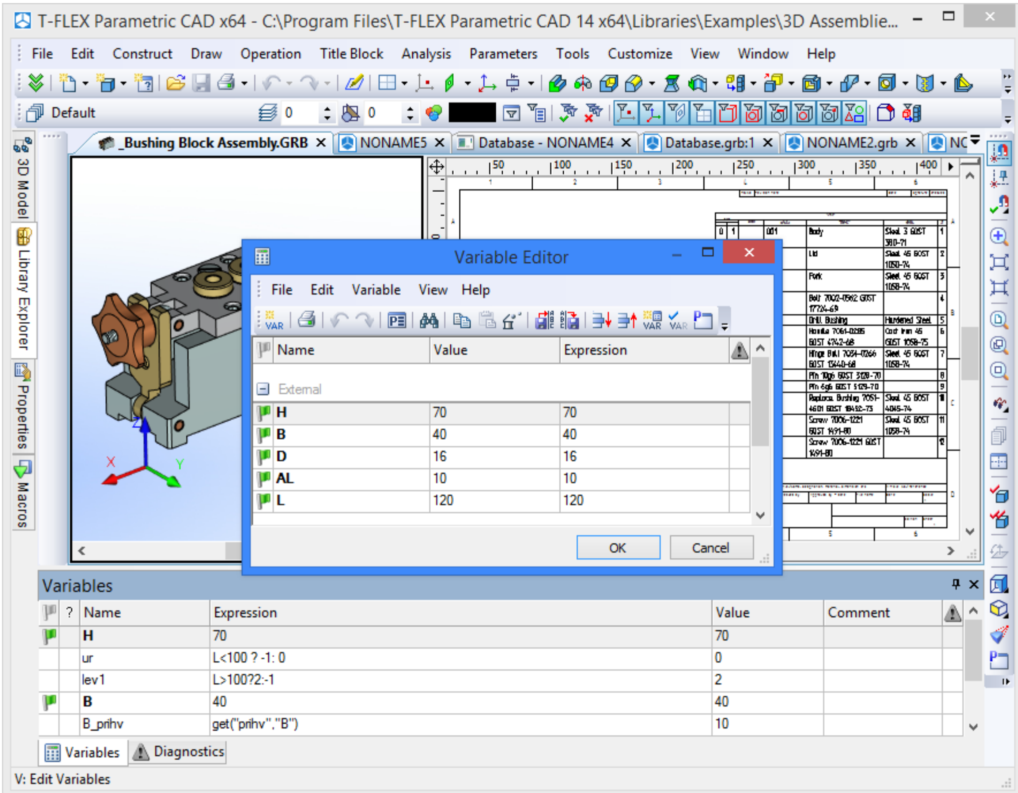
After calling this command the standard printing options dialog appears. As a result, all content of the variables editor will be sent to printer in the same form as it was displayed on the screen.

Before calling this command, separate lines from the table of variables can be picked out in the window of variables editor with the help of , <Ctrl>+, <Shift>+. Then by setting on the flag **Print range > Selection** in the printing options dialog, it is possible to print out only selected lines and not the entire content of the editor.

Working with Variables Editor in Transparent Mode

To work with the variables editor in the transparent mode (while working simultaneously in the drawing or 3D model window), the service window of the system “Variables” can be used. Similar to other service windows of the system, the window “Variables” can operate in floating and popup mode. Also it can be placed along one of the sides of the main window of the system.

By functionality the window “Variables” duplicates the main window of the variables editor, called with the command **V: Edit variables**. But the textual menu and the toolbar are absent in the window “Variables”. All operations with the variables in the window “Variables” can be carried out only with the help of the context menu and hot buttons. Such simplification of the interface is completely compensated for by the transparent operation with the variables. With the help of the window “Variables”, it is possible to edit variables while being in any command. Upon changing the expression of the variable in the window “Variables”, automatic recalculation of the drawing (or 3D model) is carried out. All changes are immediately displayed in the window of the system.



EDITING EXTERNAL VARIABLES

External variables are usually widely used in the T-FLEX CAD documents for organizing parametric connection between assembly document and fragments, and also for organizing connection of the T-FLEX CAD with other systems and applications.

For editing variables, marked as external, besides the variables editor, the command **M: Model Parameters** can be used:

Icon	Ribbon
	Parameters → GUI Control → Model
Keyboard	Textual Menu
<M>	Parameters > Model

In contrast to the variables editor, only those variables of the current document which are marked as external are displayed in this command. This command does not allow a user to create new variables. Thus, if there are no external variables in the drawing, the message **No external variables** is produced and the command is not called.

The kind of the dialog box used in this command depends on the attribute setting **External variable editor** in the command **ST: Set Document Parameters** (the tab **External variables**). This attribute can assume one of the following settings:

Internal Editor. Upon calling the command **M: Model Parameters**, the **Variable Editor** dialog box comes up for editing external variables, that looks similar to the normal variable editor window;

External Program. In this case, the appearance of the dialog box is defined by the external custom application;

Control. The user might have created a custom dialog box using interface elements (see the chapter "Control Elements. Creating User Defined Dialog Boxes"). In such a case, calling the command **M: Model Parameters** brings up this dialog box.

The command **M: Model Parameters** can be used for modeling the process of editing external variables of the fragment in an assembly. One more possible way of using this command – when too many expressions are specified in the current drawing. In this case it is possible to mark the variables on which the remaining variables depend as external ones, and, if necessary, modify their values in the external variables editor.

Upon changing the values of the external variables with the help of the command **M: Model parameters,** only constants can be used as admissible values of variables.

After completing the command, all variables are recalculated per the changes to the external variables, and the drawing is regenerated with the new parameters.

Use of Variables in T-FLEX CAD

The variables and expressions created within the variable editor do not affect the drawing in any way per se. The variable editor in itself is merely a powerful calculator. However, the variables can be used in T-FLEX CAD system in many various ways.

Variables and Construction Lines

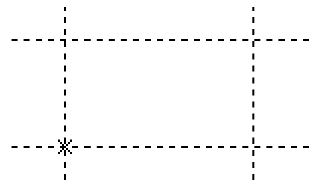
The main application for numerical variables is their use as construction line parameters.

A variable can be assigned as a parameter to a construction line in the following two ways:

When creating construction lines in the commands **L: Construct Line**, **C: Construct Circle** and **EL: Construct Ellipse**. This can be done in the dialog of the properties window for the given commands or in the dialog of the construction line parameters, called with the help of the option <P>.

When editing construction lines in the command **EC: Edit Construction**. To do that, it is necessary to choose a line in the edit command. After that, it is possible to specify the variable as a line parameter in the dialog of the properties window or in the dialog of the line parameters, called with the help of the option <P>.

Your actions for defining a construction line parameter via a variable are the same in both cases. Therefore, consider only the example of editing construction lines.

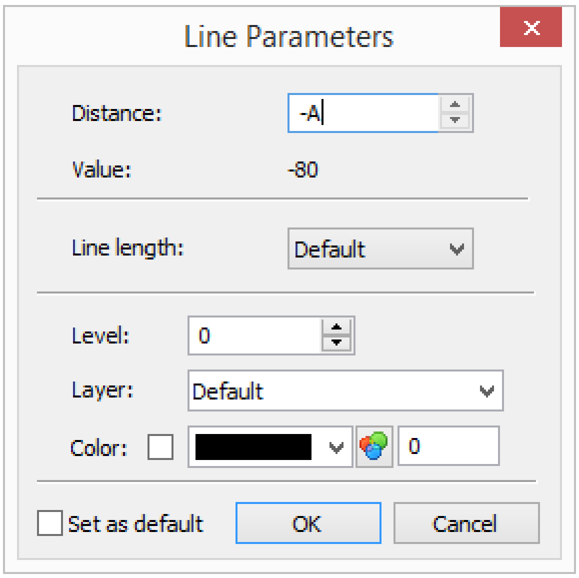


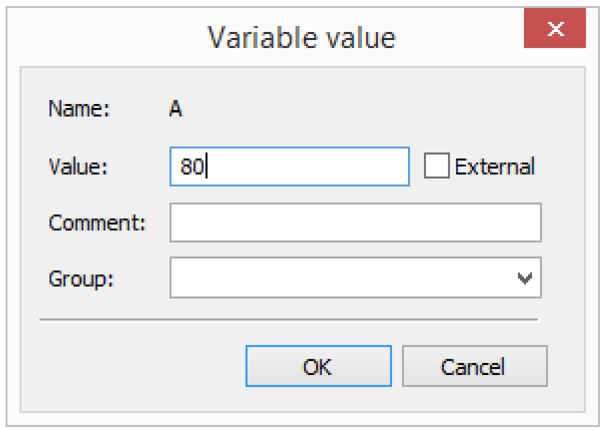
In this example, the left and bottom lines are constructed as vertical and horizontal respectively. The right and top lines are constructed as parallel, accordingly.

> Save this drawing with a certain name, say, TEST, as it will be further used for describing variable uses.

Call the command **EC: Edit Construction**. Select the right vertical line In the dialog of the properties window place the cursor into the field of the parameter "Distance".

The distance from the reference line is a parameter of a parallel line, and, by default, this distance was specified as constant. Replace the value by the expression *"-A"*.





Hit at any place of the drawing. Two outcomes are then possible:

- if the variable *A* exists, then the construction line will adjust to the variable value.

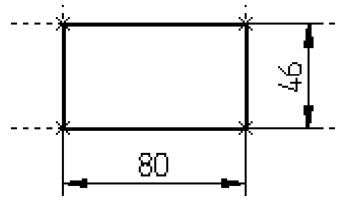
- if the variable does not exist, then the dialog box will appear on the screen for defining the value of the new variable. You can then also mark the variable as external. After pressing **[OK]** the construction line will be adjusted in accordance with the value of variable.

By doing the same with the top horizontal line and defining its parameter by a variable *B*, you establish relation between the variables and the construction lines. From now on, the construction line positions will be driven by modifications in the variables *A* and *B*.

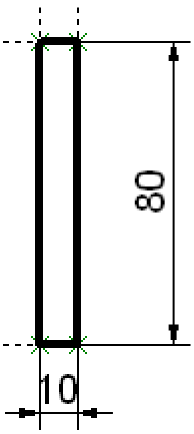
Note that construction line parameters can be defined by variables only when the option or <P> is available. (Refer to the commands **L: Construct Line**, **C: Construct Circle, EL: Construct Ellipse**).

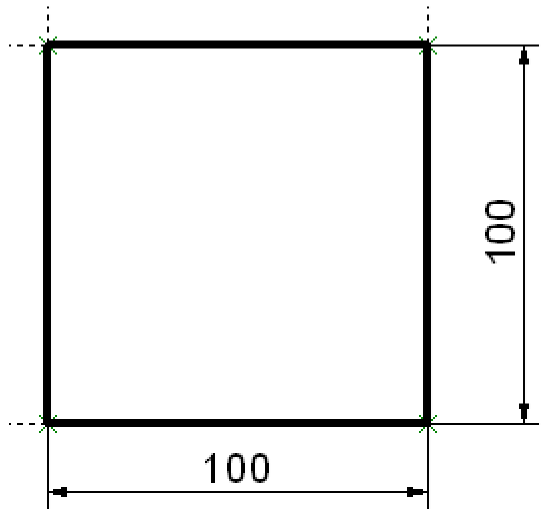
To clearly witness the relation between the construction lines and the variables, let's complete the drawing.

Draw the graphic lines using the command **G: Create Graphic Line** and apply the dimensions between the vertical and the horizontal lines using the command **D: Create Dimension**.



After that, in the window "Variables" or in the main window of the variables editor (called with the command **V: Edit Variables**), change the value of variables *A* and *B*.





Name	Value	Expression	
B	10	10	
A	80	80	

Name	Value	Expression	
B	100	100	
A	100	100	

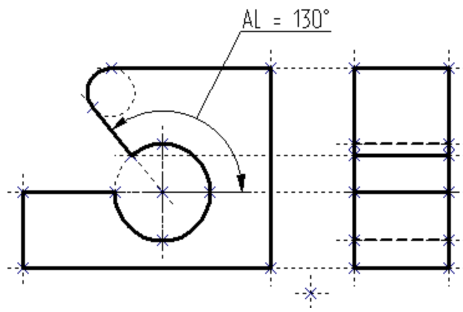
Note that modifications in variables and expressions driving the construction lines, also affect the dimension display and dimension values.

The dimensions themselves can be also used for modifying position of the construction lines, with which these dimensions are connected, and as a consequence, values of the variables determining parameters of these lines. To do that, it is necessary to point the cursor at the dimension value and hit . Chosen dimension value is selected for editing. Upon modifying the dimension value, the system automatically changes position and parameters of construction lines, on which the given dimension is based. If position of the given line was determined by a variable, the value of this variable will be also changed.

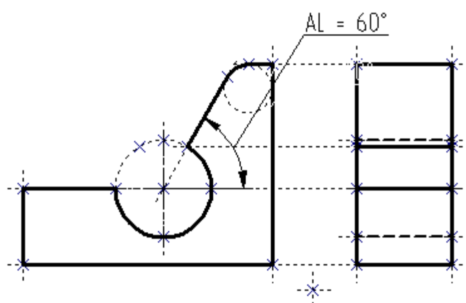
When using variables as construction line parameters, try not to use complicated expressions. The recommended approach is defining construction line parameters via a standalone variable or a simple expression. All complicated mathematical relations can then be defined within the variable editor. This helps keeping definitions in one place, without need of visiting all possible commands and searching through all elements for handling.

Variables and Visibility Levels

It is often convenient to define visibility level values by variables. This helps covering a wider variety of configurations by a single parametric model. Consider, for example, the parametric drawing on the diagram.



The two views are interdependent. The slanted line was constructed as passing through a node, at a specified angle to the horizontal. A variable *AL* is introduced as the parameter of the line. The configuration on the diagram corresponds to the value of *AL = 130*. Let's modify the variable value to *AL = 60*.



The upper dashed graphic line on the side view stays after modifications, which is wrong. This graphic line was originally created on top of a construction line parallel to the horizontal line and tangent to the circle. This flaw can be fixed by using the variable *LEVEL* as the value of the visibility level of the dashed graphic line.

The value of this variable can be defined by the following expression:

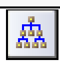
```
LEVEL = AL > 90 ? 0 : -1
```

The visibility interval for graphic lines is set from 0 to 127 (inclusive). In our case, if the variable *AL* is greater than 90, then *LEVEL* equals 0, which is within the visibility interval. Therefore, the graphic line will be drawn. If *AL* is less than 90, then *LEVEL = -1*, which is outside the visibility interval, and the graphic line will not be drawn. This drawing sample can be found in the directory *"Documentation samples/2D Design\Variables\ Variables and visibility on levels.GRB"*.

This approach allows creation of models representing a family of product modifications. An example could be a drawing of a bolt with various head styles.

Variable Dependency

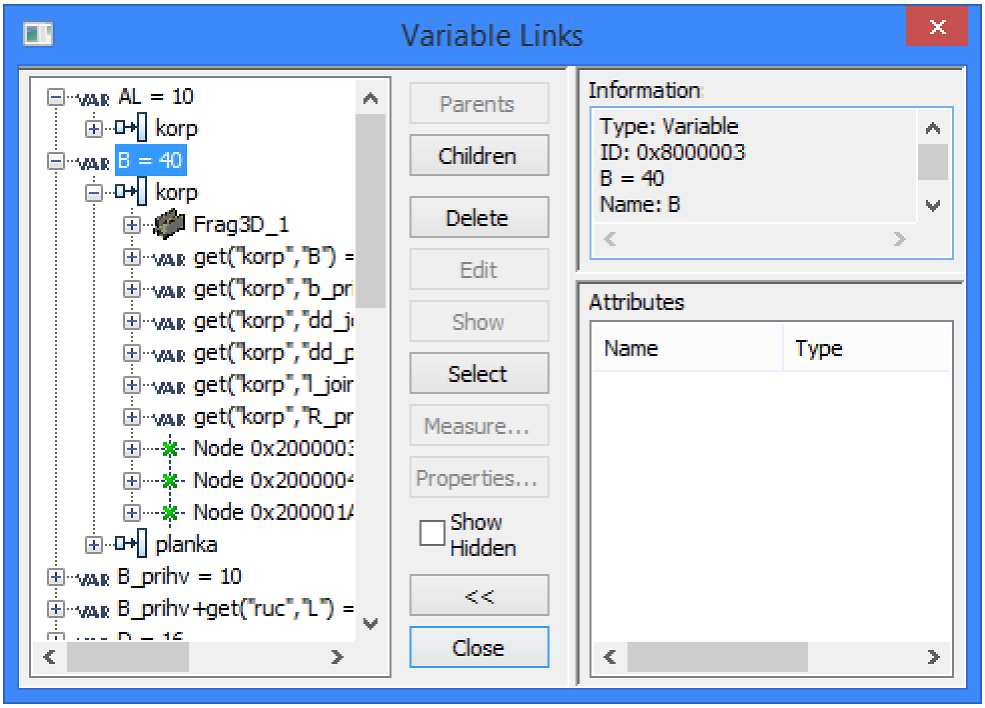
Variable interdependencies with other elements can be examined with the help of the command **SR: Show Variable links**. The command is called as follows,

Icon	Ribbon
	Parameters → GUI Control → Links...
Keyboard	Textual Menu
<SR>	Parameters > Links...

As a result of calling the command, the dialog box appears, listing all the variables of the drawing and their values in a tree layout. By default, only the visible variables are in the list. To view all variables of the given document (including hidden), set the flag **Show Hidden Variables**.

The "+" glyph in a box before a variable indicates a collapsed branch. Such a branch contains a list of one or more drawing elements that rely on the variable. To expand the list, point and click the left mouse button over the box. The list contains element types and Ids.

The listed elements, in turn, may be constructed based on other elements. In such a case, their branches will also be preceded by the box with the plus inside. The base (independent) elements in construction hierarchy are on plain branches, not prefixed with the box. The pane on the right-hand side of the dialog box displays information about the selected element. Besides that, the selected element is highlighted in the drawing or in the 3D window.



Besides viewing relations of variables with drawing elements, you can additionally perform certain manipulations with 2D or 3D elements selected in the list, using the following buttons:

[Parents] Upon clicking this button, the dialog window displays the tree of parent elements for the element selected in the list, instead of the list of all variables. This button is unavailable for the elements without parents.

[Children] Works similar to the previous button, but instead of the parent element tree displays the children's tree for the selected element.

[Delete] Closes the command window and calls the command to delete the selected element.

[Edit] Closes the command window and calls the command to edit the selected element.

[Show] Closes the command window. The current drawing (model) working window is adjusted so as to fully display the selected element on the screen.

[Select] Closes the command window. The selected elements stay selected for further manipulations.

[Measure…]. Calls the command **PM: Measure Element or relation between two Elements** for the selected element.

Properties…] Calls the parameters dialog for the selected element. After finishing working with the dialog, the **SR: Show Variable links** command window is resumed.

[Close]. Exits the command.

[<<] [>>] Open and close an additional console in the **Information** dialog window, containing the following fields:

Information. This field displays a brief information about the selected object.

Attributes. This field displays information about the attributes assigned to the selected element.

Attachment I. Rules for Writing Expressions. Functions for Working with Variables

Expression

Expressions determining the values of the variables can contain operands (real constants and variables, text constants and variables) and operations (a set of actions upon these variables). Expressions can also include functions.

Members of Expressions

Numerical constants.

Numerical variables.

String variables.

Numerical constants may not contain spaces.

Examples of correctly defined constants:

2; 3.344; -2.34; 1.234e+5; 1.2344E-32; 0.0034;

Examples of incorrectly defined constants:

2,34 – the comma is not allowed as the decimal symbol.

1.234 e+5 – inadmissible "space" symbol is used in a constant.

Note for local language users: in the exponential number representation, use only the US ASCII "E" or "e".

String constants.

A string constant is an arbitrary string entered in quotes:

"This is a string constant!"

Should a string constant include the quote symbol (“), it must be preceded by the backslash symbol (\).

“This is another \”string\” constant!”

The above is the way to enter a string constant, whose value is to read,

This is another ”string” constant!

To have the backslash symbol a part of a string constant, it must be duplicated.

Example:

“This \\is\\ t\w\o\!”

The value reads,

This \is\ two!

Note that a single backslash is ignored throughout.

Instructions (Operations) Used in Expressions

The string members can only be subject to the operation

concatenate, or, simply, addition of two strings (+)

“T-FLEX”+” CAD” = T-FLEX CAD

The numerical members are subject to common arithmetic operations, as

addition (+)

subtraction (-)

multiplication (*)

division (/)

unary negation (minus).

Examples of correctly defined expressions (followed by the result after the “=” sign):

2 + 3 = 5

5 - 9 = -4

Do not divide by zero. This will result in an error.

Use of “unary negation” operation is illustrated by the following example. Suppose, **VAR_1** is equal to **5**, then the following expression yields:

- VAR_1 = -5

An arbitrary number of spaces are allowed in expressions, for example,

5 * 3 + 2 = 17

Spaces make expressions more readable.

An important issue is the order of operations (precedence).

Thus, the resulting value of the following expression,

2 + 3 * 4

will be **14**, rather than **20**, because multiplication operation has higher precedence than addition. To change the order of operations, use parentheses. The previous expression can be modified in the following way in order to yield **20**:

(2 + 3) * 4

Proper use of parentheses helps avoiding unexpected results.

Power of (** or ^)

Example:

2 ** 3 = 8

-3 ** 3 = -27

The following examples demonstrate specifics of this operation:

0 ** 17 = 0 (zero to any power is zero).

23 ** 0 = 1 (any value to the power of zero is one).

Errors may occur on evaluating this operation. The following message is output in such a case: "`illegal power function in line 1`".

Errors occur in the following cases:

-2 ** 3.4 (an attempt to raise a negative value to a fractional power).

23 ** 234344 (overflow error due to too large resulting value).

Modulo division (%)

Example:

23 % 5 = 3

23.7 % 5.5 = 1.7

-23 % -5 = -3

23 % -5 = 3

-23 % 5 = -3

The result of the operation **member1 % member2** is the remainder of dividing **member1** by **member2**.

The value of **member2** may not be zero. In the case **member2** = 0, the error occurs, "`Zero divide in line 1`".

Besides the above-mentioned algebraic operations, logical (comparison) operations can be used in expressions. The result of a logical operation is the numerical value 1, if the relation defined by the operation is true, and 0 otherwise.

Logical Operations

Greater than (>)

Less than (<)

Greater than or equal (>=)

Less than or equal (<=)

Inequality (!=)

Equality (==)

Logical AND (&&)

Logical OR (> >)

Logical NOT (!)

Examples:

23 > 45 && 56 < 34

This example expresses the question: Is the number **23** greater than the number **45** and the number **56** less than the number **34**? Obviously, the answer will be - **no**, therefore the value of this expression is zero.

The expression **!VAR_1** is the same as **VAR_1 == 0**

Logical operations are usually used for comparing the value of a variable against a constant or a value of another variable. A shortcoming here is that only two values are possible as the result of evaluating a logical expression - **0** or **1**.

Another form of using logical operations is a conditional statement.

A conditional statement has the following structure:

condition ? value1 : value2

Example:

VAR_1 > 100 ? 1 : -1

If the value of **VAR_1** is greater than **100**, then the statement will yield **1**, otherwise it yields **-1**.

One can use arbitrary expressions for the **condition**, **value1** and **value2**.

VAR_1 ? 1 : -1

or, just the same thing,

VAR_1 != 0 ? 1 : -1

(VAR_1 != 0 && VAR_2 == 0) ? (VAR_3 + 1) : (VAR_4 -1)

Standard Mathematical Functions

ABS	Return absolute value of	*abs (-20) = 20*
ACOS	Calculate arccosine	*acos (0.5) = 60*
ASIN	Calculate arcsine	*asin (0.5) = 30*
ATAN	Calculate arctangent	*atan (1) = 45*
CEIL	Find integer ceiling	*ceil (3.98) = 4*

COS	Calculate cosine	*cos (60) = 0.5*
FLOOR	Find largest integer less than or equal to argument	*floor (3.13) = 3*
HYPOT	Calculate hypotenuse of right triangle	*hypot (3, 4) = 5*
INT	Round to nearest integer	*int (3.13) = 3*
LOG	Calculate natural logarithm	*log (1) = 0*
LOG10	Calculate base-10 logarithm	*log10 (10) = 1*
RACOS	Calculate arccosine, in radians	*racos (0.5) = 1.0472*
RASIN	Calculate arcsine, in radians	*rasin (1) = 1.5708*
RATAN	Calculate arctangent, in radians	*ratan (2) = 1.10715*
RCOS	Calculate cosine, angle input in radians	*rcos (1) = 0.540302*
ROUND(ARG1, ARG2)	Round the value ARG1 with accuracy ARG2.	*Round (2.357, 0.25) = 2.25* *Round (2.357, 0.1) = 2.4*
RSIN	Calculate sine, angle input in radians	*rsin (1) = 0.841741*
RTAN	Calculate tangent, angle input in radians	*rtan (1) = 1.55741*
SIN	Calculate sine	*sin (30) = 0.5*
SQRT	Find square root	*sqrt (16) = 4*
TAN	Calculate tangent	*tan (45) = 1*

All functions except **hypot** and **ROUND** have one numerical argument. Function arguments can be substituted by any expression, including other function calls that result in real numbers.

SIN (10 + 10 + 10) = 0.5

SIN (SQRT (900)) = 0.5

The functions **hypot** and **ROUND** have two numerical arguments separated by a comma:

HYPOT (1 + 1 + 1, 1 + 1 + 1 + 1) = 5

The angle arguments of trigonometric functions are input in degrees, except for the functions whose names begin with “R”.

T-FLEX CAD Functions

ATOF(“10.5”)	Convert the string *“10.5”* to the real number *10.5*
ATOT(1.5,0.01,1,0)LTOT(1.5,0.01,1,0) SATOT(1.5) SLTOT(1.5)	Convert the real number *1.5* to a string per the format specified by the rest three arguments
FTOT(“variable name”)	The function of conversion a numeric value to text. It converts a numeric variable or expression to the text using a comma instead of a dot.
CHECK(“file name”, type)	Find a file in the specified folders.
DISTANCE (“NAME1”, “NAME2”)	Get the distance between the entities specified by their names or Ids.
ERROR(“STRING”)	Display a user-defined message *“STRING”* on the screen
FTOA(10.5)	Convert the real number *10.5* to the string *“10.5”*.
FIXNODENAME(n)	Get the name of the fragment’s node used for inserting the current document as a fragment by fixing points. This function is helpful for creating the libraries of logical and algorithmic schemes. It helps to orient direction of connecting arrow between elements. Input parameter: the number of the fragment’s fixing point.
GET(“STR”,“P”)	Get the real value of the parameter *P* of the system element named *STR*. Instead of element’s name it is possible to specify its identifier (ID). The entire list of parameters whose values can be obtained with the help of the function get, can be found in the Attachment II of this chapter.
GETG/TGETG("NAME”,N)	Get the value of a numerical/string global variable named *NAME*.

GETV("NAME",N) or *GETV("NAME_Page",N)*	Get the value of the service parameter of the document named *NAME*. *N* – is a value that will be returned by the function if it does not find the indicated parameter. Certain parameters are defined separately for each 2D page of the document. In this case, the ending "_Page" is added to the name of the parameter, where "Page" – is a name of desired page of the current document. If the page name is not specified, the parameter value will be returned for the first page. The entire list of parameters whose values can be obtained with the help of the function getv can be found in the Attachment II of this chapter.
GRAPH("Graph_name",X)	Get the value of the function F(x), corresponding to the X argument value, for the graph named *"Graph_name"*.
ISFRAGMENT()	Find the assembly hierarchy level of a fragment. For the current drawing, returns zero.
MAX(N1,...,NN)	Find the maximum value among the input set.
MEASURE("NAME1", "NAME2", "RELATION")	Get the value of requested relation *RELATION* between specified elements *NAME1* and *NAME2* (elements' names or IDs can be used as input values).
MIN(N1,...,NN)	Find the minimum value among the input set.
SETG/TSETG("NAME",N)	Set the value N of a numerical/string global variable named *NAME*.
SETV("NAME_Page",N) or *SETV("NAME",N)*	Set the value *N* of the global parameter named *NAME* for the page named *"Page"* of the current document. If the page name is not specified, the parameter value will be set for the first page of the document.
STRLEN("STR")	Find the number of characters in the string STR
TFIND("string1", "string2")	Searches for the substring "string2" in the string "string1". Returns an integer value equal to the position number of the first substring occurrence, counted from 1. In the case of an error (the substring is not found) returns 0.

TGET()	This function allows the user to obtain textual properties of the elements: the name of the material of 3D operations, the values of text variables of the fragment. $text = tget("0xD000001","$textvar") – returns the string value from variable $textvar. $mater = tget("Extrusion_0","Material") – defines material for the operation "Extrusion _0".
TGETV ("system variable")	Get the string value of a system variable or characteristics of the current drawing. The entire list of parameters whose values can be obtained with the help of the function tgetv, is given in the Attachment II of this chapter.
TMGETV ("system variable")	Get the string value of a system variable of the assembly if the current drawing is inserted there as a fragment. This function works similar to the function TGETV.
TPART("string",N,N)	Get a substring.
TREPLACE("string1", "string2", "string3")	Replaces the substring "string2" by the substring "string3" in the string "string1". Returns a character string, which is the input string with the entry replaced.
TWORD("string", N)	Get a word from a string. $NAME=TWORD("William Henry Gates", 2) – results in assigning the value Henry to the variable $NAME.
WARN("STRING")or *WARN("STRING","element name")*	Display a user-defined message *"STRING"* in the diagnostics window. Include the element name in the message.

Database Management Functions

DBF(arg1, arg2, arg3)	A dBASE database query. *arg1* – database name. The database name can be defined by a string constant, variable or expression. *arg2* – the name of the field to copy from. The field name can be defined by a string constant, variable or expression. *arg3* – the condition for copying. The condition can be defined by a string constant, variable or expression.

DBFWIN(arg1, arg2, arg3)	A dBASE database query. Also converts text from DOS to WINDOWS format. Is used for correct handling of local language text. The parameters are analogous to those of the function dbf.
FIND(database_field, condition_1, condition_2, ...)	Get a value from the internal database The function returns the value of the specified field *database_field* of a record satisfying the conditions condition_**1**, condition_**2**. If no suitable record exists, the function outputs the error message “Wrong record number”.
MDB(arg1, arg2, arg3, arg4)	An Access database request. *arg1* - database name. The database name can be defined by a string constant, variable or expression. *arg2* -the table name in the database. Can be defined by a string constant, variable or expression. *arg3* - the name of the field to copy from. The field name can be defined by a string constant, variable or expression. *arg4* - the condition for copying. The condition can be defined by a string constant, variable or expression. For example: mdb ("c:\\T-FLEX_USER.mdb", "USER", "FULLNAME", "Code ={kod}") This means: select the value from the table “USER” of the database “T-FLEX_USER” from the field “FULLNAME” under condition that the value of the field “Code” is equal to the value of the variable kod. It is worth noting that the last operand of the function, specifying condition for record selection, can be written in the form of SQL query and then it must correspond to the WHERE clause of the SELECT command. If, upon writing the condition, the text variables are used, the expression will take the following form: mdb ("c:\\T-FLEX_USER.mdb", "USER", "FULLNAME", "Post=\"{$Dol}\"").
REC(condition)	Get the Id of a record in the internal database. *condition* – a logical expression, taking values True or False. The expression may contain members – queries to the fields of the database.

FREC(arg1, arg2, arg3, arg4)	Get the Id of a record in the internal database or in the database by reference, where the value in the specified column matches most closely the specified value. arg1 – a database column, in which the search is performed. Must be of real or integer type; arg2 –sought value; arg3 –search option. Possible values: 0 –find nearest value; -1 –find nearest lesser value; 1 –find nearest greater value. arg4 –defines the column type to perform the search (the order in which the values occur in this column): 0 -the values are not in order; the search is performed over all database records; 1 –the values in the column are ordered ascending or descending. Once the difference between the value sought and the value in the current column of the database is greater than in the previous one, the search completes. The parameters arg3 and arg4 are optional. If these are skipped, the default values are used: arg3 = 0; search for nearest value; arg4 = 0; column not ordered;
FTOT	Executes the transformation of a real variable or expression into the text with the use of comma instead of a period.
VAL(record_number, database_field)	Get the value from the internal database by the record Id. *record_number* -an arbitrary arithmetical expression, that yields an integer. *database_field* -a field query.
#.<name>	Get the number of records in the specified internal database

The detailed description of T-FLEX CAD functions follows below. The database managing functions description are also available in the chapter "Databases".

Examples of function uses:

sin (30) = 0.5

min (5, 67, 34, 28, 0.67) = 0.67

SQrt (16) = 4

As evident in the last example, the function naming is case-insensitive.

Attachment II. Examples of Using Some Functions

Using Functions ATOT (), LTOT (), SATOT (), SLTOT ()

These four functions are intended for converting numerical values to character strings in a specified format. The conversion format is similar to that used for converting the nominal dimension values.

The function *SATOT ()* converts a real number of an angular value into a string of text. The conversion parameters are defined by the settings of the **ST: Set Document Parameters** command on the **Dimensions** tab.

The function *SATOT ()* specifically uses the definitions from the **Angular Dimensions** group. Suppose, the following parameters are defined in the **ST: Set Document Parameters** command.

In this case, calling *SATOT (120.34567)* leads to the following result:

12%%d30'

Angular dimensions	
Units	10° 30' 15"
Minimum Digits	0
Accuracy	0.5

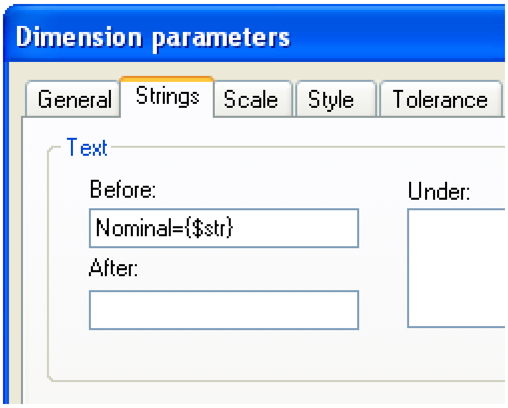
The character combination *%%d* stands for the degree symbol (°).

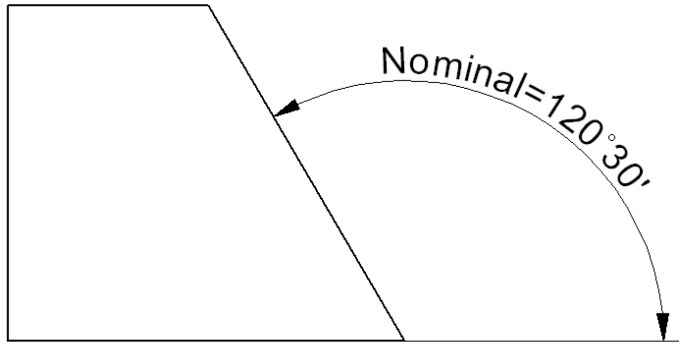
In order to realize this character combination as the degree symbol, the variable must be used as a parameter of some detailing element, such as a text (string text only), dimension, leader note, etc.

Let's clarify the above on the following example. Suppose, the following value is assigned to the variable $str by the function call:

$str = SATOT (120.34567) = 120%%30'

We will then create a dimension with the "Before" string containing the variable in braces instead of the default. The following result will then be displayed:





The variables that are assigned return values of such functions can also be entered (in braces) in string text. Such text will then be correctly displayed as well.

Modifying the **Units** parameter in the command **ST: Set Document Parameters** as shown leads to the following result:

SATOT (120.34567) returns → 120.5%%d

Angular dimensions	
Units	10.125°
Minimum Digits	0
Accuracy	0.5

The “Accuracy” parameter allows controlling the accuracy of the returned result:

SATOT (120.34567) returns → 120%%d21'

Angular dimensions	
Units	10° 30' 15"
Minimum Digits	0
Accuracy	0.05

The “Minimum digits” parameter is used only for decimal representation:

SATOT (120.34567) returns → 120.500%%d

Angular dimensions	
Units	10° 30' 15"
Minimum Digits	3
Accuracy	0.05

The function *SLTOT ()* differs from the previous one only in that the value to be converted is considered a linear value. Therefore, the conversion formats change accordingly. There are four standard formats for this function:

1 – decimal format. Example: 1.123;

2 – in inches;

3 – in inches and fractions;

4 – in feet, inches and fractions.

The following settings showing in the pictures result in:

SLTOT (120.34567) returns → 120.35

- 

The functions *ATOT ()* and *LTOT ()* are, respectively, the variations of the functions *SATOT ()* and *SLTOT (),* with the conversion parameters defined explicitly.

The calling sequence of the function LTOT is as follows,

LTOT (value, accuracy, standard, digits), where

value – a real number to be converted;

accuracy – the accuracy to be used in the conversion;

standard – the measurement system; takes one of the following values:

1 – decimal format. Example: 1.123;

2 – in inches;

3 – in inches and fractions;

4 – in feet, inches and fractions;

digits – the number of significant decimal digits in the decimal representation.

The calling sequence of the function ATOT is as follows,

ATOT (value, accuracy, standard, digits), where

value – a real number to be converted;

accuracy – the accuracy to be used in the conversion;

standard – the angular units; takes one of the following values:

1 – decimal format. Example: 1.123;

2 – degrees, minutes and seconds. Example: 1°2’30”.

digits – the number of significant decimal digits in the decimal representation.

Example:

LTOT (120.34567, 0.001, 1, 5) returns → 120.34600

Using Function *GET ()*

This function returns the value of the requested property of a 2D or 3D element, as well as the current drawing page. Sets of accessible properties vary depending on the queried element.

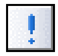
> The function will be automatically substituted as expression for a variable if such variable is created when measuring properties of 2D or 3D element in **PM: Measure Element or relation between two Elements** command.

The calling sequence:

GET ("string1", "string2"), where

string1 – the queried element name or Id,

string2 – the parameter name.

All 3D elements and operations are assigned Ids (unique identification numbers). They are also assigned a "Name" parameter that is automatically initialized by a system default. The name can be changed by the user if necessary. The 2D elements by default only get an Id. Names can be assigned to some 2D elements, particularly, to 2D nodes and graphic lines. The names are assigned in the editing commands, such as "EN: Edit Node" or "EG: Edit Graphic Line", using the  option.

The elements whose parameters can be got by the described function, are:

- the drawing's 2D pages;

- construction lines;

- nodes;

- graphic lines;

- text;

- fragments;

- hatches;

- 3D elements;

- 3D operations;

- faces, edges, **loops**.

The following reserved names are used for the available parameters:

for 2D pages of a drawing:

"ZONES_STEP_X" – the size of one zone along X-axis (the step size along X-axis);

"ZONES_STEP_Y" – the size of one zone along X-axis (the step size along Y-axis_;

"ZONES_OFFSET_X" – the offset of the area being divided into the zones from the point (0,0) - along X-axis;

"ZONES_OFFSET_Y" – the offset of the area being divided into the zones from the point (0,0) - along Y-axis;

"ZONES_COUNT_X" – the number of zones along X-axis;

"ZONES_COUNT_Y" – the number of zones along Y-axis.

> The parameters of dividing a drawing page into zones are defined in the command **ST: Set Document Parameters**, on the tab **Paper > Zones**.

for nodes:

"X" – X-coordinate of the node;

"Y" – Y-coordinate of the node;

for construction lines:

lines:

"X", *"Y"* – the coordinates of the first node the straight construction line is passing through;

"P1", *"P2"* – the coordinates of the second node the straight construction line is passing through;

circles and ellipses:

"LENGTH" – the length of the circumferential;

"P1" – the radius of the circle (this parameter is specific to circles);

"X", *"Y"* – the coordinates of the center of the circle;

splines, 2D paths, functions and offsets:

"LENGTH" – the entity length;

for graphic lines:

segments:

"LENGTH" – the segment length;

"START_X", *"START_Y"* – the coordinates of the start point of the segment;

"END_X", *"END_Y"* – the coordinates of the end point of the segment;

based on circles:

"LENGTH" – the length of the circle arc;

"CENTER_X", *"CENTER_Y"* – the coordinates of the circle center;

"ANGLE" – the angular arc length of the graphic line;

"RADIUS" – the radius of the circle;

based on ellipses, splines, **2D** paths**,** offsets and functions:

"LENGTH" – the entity length;

for graphic lines constructed as a circular or elliptical arc, portion of spline, 2D path, offset or function, there are following additional parameters:

"START_X", *"START_Y"* – coordinates of the arc start point;

"END_X", *"END_Y"* – coordinates of the arc end point;

for hatches:

"AREA" – the hatch area;

"PERIMETER" – the hatch contour perimeter;

"XMASS" – X-coordinate of the center of gravity;

"YMASS" – Y-coordinate of the center of gravity;

"XAREAMOMENT" – 1x component of inertia moment;

"YAREAMOMENT" – 1y component of inertia moment;

"PRODUCTAREAMOMENT" – Centrifugal inertia moment of area;

"XINERTIARADIUSVALUE" – Radius of inertia X;

"YINERTIARADIUSVALUE" - Radius of inertia Y;

"XAREAMOMENTMAINVALUE" – 1x component of principal moment of inertia relative to mass center;

"YAREAMOMENTMAINVALUE" – 1y component of principal moment of inertia relative to mass center;

"MAINAXESROTATIONVALUE" – Rotation angle of principal axes.

for text:

"WIDTH" – the text width;

"HEIGHT" – the text height;

"X", *"Y"* – X and Y coordinates of text fixing point;

"TEXT" – text content;

for fragments: "string1" represents the fragment name or Id, "string2" – the name of the fragment variable. The function returns the value of the variable.

In addition, the following parameters are available for 2D fragments:

"BoundBoxLeft" – left coordinate of bound box (X-coordinate);

"BoundBoxRight" – right coordinate of bound box (X-coordinate);

"BoundBoxTop" – top coordinate of bound box (Y-coordinate);

"BoundBoxBottom" – bottom coordinate of bound box (Y-coordinate);

"BoundBoxCenterX" –X-coordinate of bound box center;

"BoundBoxCenterY" –Y-coordinate of bound box center.

In the three-dimensional version of the system, the following 3D element parameters are accessible:

for operations:

"Area" – the surface area;

"Mass" – the mass of the body (material density accounted);

"Xmass", *"Ymass"*, *"Zmass"* – X, Y, Z-coordinates of the center of gravity;

"IX", *"IY"*, *"IZ"* – the moments of inertia about the axes X, Y, Z respectively;

"IXY", *"IYZ"*, *"IZX"* – the products of inertia with respect to the planes XY, YZ, ZX;

"Volume" – volume;

for Apply Material operation the following additional parameter is used:

"MaterialArea" – total area of faces with the attached material;

for all 3D Arrays (except arrays of faces) the following additional parameter is used:

"CopyCount" – actual number of copies in the array (with account of limitations and exclusions);

for 3D nodes:

"POINTX", *"POINTY"*, *"POINTZ"* – X, Y, Z-coordinates of the 3D node;

for a 3D profile:

"Area" – the area of the surface surrounded by the 3D profile;

"Perimeter" – perimeter of a closed 3D profile or length of an open one;

for a 3D path:

"Perimeter" – the length of the 3D path;

for a face:

"Area" – the area of the face surface;

"Perimeter" – the perimeter of the face border;

special for a flat face:

"LocationX", *"LocationY"*, *"LocationZ"* – X, Y, Z-coordinates of the face's underlying plane origin;

"NormalX", *"NormalY"*, *"NormalZ"* – X, Y, Z-coordinates of the normal to the face's underlying plane;

"RefDirectionX", *"RefDirectionY"*, *"RefDirectionZ"* – X, Y, Z-coordinates of the reference vector on the plane (the plane vector defines the direction of the X-axis of a flat plane);

special for a cylindrical face:

"Radius" – the radius of the cylinder;

"Diameter" – cylinder diameter;

"CenterX", *"CenterY"*, *"CenterZ"* – X, Y, Z-coordinates of the center;

"AXISX", *"AXISY"*, *"AXISZ"* – X, Y, Z-coordinates of the axis;

for a toroidal face the following additional parameters are used:

"MaxRadius", *"MinRadius"* - major and minor torus radii;

"CenterX", *"CenterY"*, *"CenterZ"* – X, Y, Z-coordinates of the torus center;

"AxisX", *"AxisY"*, *"AxisZ"* – X, Y, Z-coordinates of the torus axis;

for a **spherical face** additional characteristics are used:

"Radius" – sphere radius;

"Diameter" – sphere diameter;

"CenterX", *"CenterY"*, *"CenterZ"* – X, Y, Z-coordinates of sphere center;

for an edge:

"Perimeter" – the edge length;

"StartX", *"StartY"*, *"StartZ"* – X, Y, Z-coordinates of the start point;

"EndX", *"EndY"*, *"EndZ"* – X, Y, Z-coordinates of the end point;

special for a straight edge:

"VECTORX", *"VECTORY"*, *"VECTORZ"* – X, Y, Z-coordinates of the edge direction;

for a circular edge or along circular arc:

"Radius" – circle radius;

"Diameter" – circle diameter;

"CenterX", *"CenterY"*, *"CenterZ"* – X, Y, Z-coordinate of the circle center;

"AxisX", *"AxisY"*, *"AxisZ"* – X, Y, Z-coordinates of the circle plane normal;

"StartAngle" – arc start angle;

"EndAngle" – arc end angle;

special for a elliptical edge:

"MajorRadius" – ellipse major radius;

"MinorRadius" – ellipse minor radius;

"CenterX", *"CenterY"*, *"CenterZ"* – X, Y, Z-coordinate of the ellipse center;

"AxisX", *"AxisY"*, *"AxisZ"* – X, Y, Z-coordinates of the normal to the ellipse plane;

"MaxDirectionX", *"MaxDirectionY"*, *"MaxDirectionZ"* – X, Y, Z-coordinates of the ellipse major axis;

"MinDirectionX", *"MinDirectionY"*, *"MinDirectionZ"* – X, Y, Z-coordinates of the ellipse minor axis;

"StartAngle" – the start angle;

"EndAngle" – the end angle;

for loop:

"Perimeter" – loop perimeter.

The parameter names are case-insensitive. If an incorrect element name or parameter name is input, then the function returns **0**.

Examples:

Get the X-coordinate of a node named **NODE_1** by calling the function GET () as follows,

get ("NODE_1", "X")

Get the length of a graphic line named *IMAGE_1* as follows,

get ("IMAGE_1", "length")

Get the perimeter of the contour of the hatch Id 0x9000001 as follows,

get ("0x9000001", "PERIMETER")

Get the volume of the 3D body of a 3D fragment named 3D fragment_11 as follows,

get ("3D fragment_11", "volume")

To compute the total length of a chain of connected graphic lines, one could call the function GET() for each line and sum up the results. However, a more efficient way is to construct a 2D path that follows along the chain of the graphic lines, and create a single graphic line from this path. In this case, the 2D path is considered a spline. Then it is easy to get the length of this single graphic line.

Using Functions DISTANCE() and MEASURE()

Function distance() returns the value of distance between two 2D or 3D elements.

> The function will be automatically substituted as expression for a variable if such variable is created when measuring distance between two 2D or 3D elements in *"PM: Measure Element or relation between two Elements"* command.

The calling sequence:

distance ("name1", "name2"), where

name1 - name or ID of the first element,

name2 - name or ID of the second element.

Distance can be calculated for the following pairs of 2D or 3D objects:

2D elements – is the distance between a 2D node and another 2D node, construction line, graphic line or hatch (the order of selecting the measurable entities is not significant);

3D objects – is the distance between two arbitrary 3D objects that are 3D construction entities (except for LCS), operations or such topological objects as an edge, loop, face, vertex.

Function distance() also measures distances for such geometrical 3D objects as vertices, edges, loops, faces. This is possible only for already named objects. Naming (e.g. "Vertex_1 or "Edge_2") occurs automatically in *"PM: Measure Element or relation between two Elements»* or in commands where such objects were specified as source data. These names should be used as input parameters.

Function measure() calculates various relations between two 2D or 3D objects.

> The function will be automatically substituted as expression for a variable if such variable is created when measuring any relation (except distance) between two 2D or 3D elements in *"PM: Measure Element or relation between two Elements"* command.

The calling sequence:

Measure ("name1", "name2", "relation"), where

name1 – name or ID of the first element,

name2 – name or ID of the second element.

relation – type of relation.

Function calculates the following types of relations:

for 2D elements:

"Angle" – is the angle between two lines, segments or a line and a segment.

for 3D objects:

"Angle" – is the angle between directional vectors of two 3D objects. Listed below are 3D objects, for which a direction can be defined (and, therefore, this relation can be calculated). Additionally specified is what will be selected as the direction vector for each object:

for a 3D path or open 3D profile lying on a straight line – the line direction;

for a 3D path or open 3D profile lying on an ellipse (circle) – the vector directed from the center of the ellipse (circle) normal to the plane of the ellipse (circle);

for a flat 3D profile; workplane; an operation body consisting of one face lying in a plane – the normal to the plane;

for a cylindrical work surface; 3D profile lying on a cylinder; an operation body consisting of one face lying on a cylinder – the axis of the cylinder;

for a 3D profile, lying on a cone; an operation body consisting of one face lying on a cone – the axis of the cone;

for a toroidal work surface; 3D profile or face lying on a torus; an operation body consisting of one face lying on a torus – the axis of the torus.

"AxisDistance" – is the distance between the axis of two 3D objects. The same 3D objects can be selected as the objects of the measurement as in the previous case (when identifying *"Angle"*), except for workplanes. In the latter case, the axes of the selected objects coincide with the directional vectors of the planes;

"DX" – Shift of two 3D nodes or 3D points with respect to each other along the X-axis;

"DY" – Shift of two 3D nodes or 3D points with respect to each other along the Y-axis;

"DY" – Shift of two 3D nodes or 3D points with respect to each other along the Z-axis.

Analogous to distance(), measure() can calculate relations between various topological 3D objects - vertices, edges, loops, and faces, that were already named earlier in *"PM: Measure Element or relation between two Elements"* or other commands.

One more additional relation can be measured for vertices, edges and faces, as well as for a pair "geometrical object – 3D node":

"GeomDistance" – is the distance between 3D points, 3D curves or surfaces corresponding to two respective 3D objects of the types: 3D node, vertex, edge, face.

Input parameters for distance() and measure() functions can be specified using either small or capital letters analogous to get() function. If an incorrect element name or relation name is input, then the function returns *0*.

Examples:

Distance between two 3D nodes “3D Node_0” and “3D Node_1” can be measured with the following call of distance() function:

distance (“3D Node_0”, “3D Node_1”)

Distance between image line with ID 0x3000014 and 2D node named as “Node 1” will be returned by the following call:

distance (“0x3000014”, “Node 1”)

Axial distance between cylinder axis resulted from extrusion operation named “*Extrusion*_1” and torus axis from “*Rotation_2*” operation can be calculated with the following call to measure() function:

measure ("Extrusion_1", "Rotation_2", "AxisDistance")

> *“PM: Measure Element or relation between two Elements”* command is recommended for using distance() and measure() functions. Read more details in chapter “Measure Elements and Relations between Them”.

Using Function *CHECK ()*

This function searches files by name in certain directories. Specifically, the files are searched in the current directory, and in the directories specified in the command **SO: Set System Options** on the **Folders** tab.

CHECK (“FILE NAME”, TYPE)

The file name must be specified with the extension. The file type is defined as follows,

0 – undefined;

1 – T-FLEX CAD document;

2 – font;

3 – database;

4 – pattern;

5 – BOM.

Using Functions SETV () and GETV ()

Functions setv() and getv() allow the user to obtain and specify the values of real service characteristics of the T-FLEX CAD documents, such as the number of pages in the document, scale and font size, assigned for each page, coordinates of corners of the title block, volume of all bodies in the 3D scene (if the document contains a 3D model), etc. The values of these characteristics can be used, for example, for transferring data between the assembly drawing and the fragments.

Syntax of Functions

setv("NAME",N)

This function sets the numeric value N for the parameter of the current document NAME.

getv ("NAME",Err)

This function returns the value of the parameter NAME of the current drawing. Err – the value which will be returned by this function if the indicated parameter is not found.

In multipage document certain characteristics (scale, font size, etc.) are set independently for each page of the document. By default, the functions get() and set() deal with the characteristics of the first page of the document.

To address characteristics of other pages it is required to add "_Page" to the end of the parameter's name, where "Page" – the name of the desired page.

List of Characteristics of the T-FLEX CAD Document Processed by Function GETV()

Characteristics of 2D pages of Document:

PAGES – total number of pages in current document;

DPAGE – the number of the current drawing's page;

DPAGES – total number of drawings' pages in the document;;

SCALE – scale of current drawing;

XL – coordinate X of the left border of the drawing;

XH – coordinate X of the right border of the drawing;

YL – coordinate Y of the lower border of the drawing;

YH – coordinate Y of the upper border of the drawing;

FSIZE – font size of the drawing.

LTHICK – thickness of main lines in the drawing.

TLTHICK - thickness of thin lines in the document.

Characteristics of 3D scene of Document:

MASS – mass of all bodies in the 3D scene of the current document;

VOLUME – volume of all bodies in 3D scene of the current document;

AREA – surface area of all bodies in 3D scene of the current document;

EXPLODE – state of exploded view mode for the fragments of 3D model of the current document (0 – exploded view mode is turned off, 1 – turned on).

XSIZE, YSIZE, ZSIZE – measure overall 3D model sizes by X, Y and Z axes of the global coordinate system.

Characteristics of fragments calculated from assembly:

_XL – coordinate X of the left border of the assembly drawing when using the current document as a fragment;

_XH – coordinate X of the right border of the assembly drawing when using the current drawing as a fragment;

_YL – coordinate Y of the lower border of the assembly drawing when using the current drawing as a fragment;

_YH – coordinate Y of the upper border of the assembly drawing when using the current drawing as a fragment;

APAGES - returns the total number of pages in the document containing a given fragment;

APAGE - returns the number of the page which contains a given fragment;

BOMPAGES – returns the total number of pages in the BOM of the document containing a given fragment;

BOMPAGE – returns the number of the page of the BOM which contains a given fragment.

> The values of these characteristics are calculated for the document-fragment inside the assembly. For example, if in the document of the fragment the user created the variable the value of which is specified by the expression (“apage”, -1), then inside the document of the fragment the value of this variable will be equal to -1, and inside an assembly document – to the number of the page of the document which contains a given fragment.

List of Characteristics of T-FLEX CAD Document Processed by Function SETV()

Characteristics of 2D pages of document:

SCALE – scale of current drawing;

XL – coordinate X of left border of drawing;

XH – coordinate X of right border of drawing;

YL – coordinate Y of lower border of drawing;

YH – coordinate Y of upper border of drawing;

FSIZE – font size of the drawing.

LTHICK – thickness of main lines in the drawing.

TLTHICK – thickness of main lines in the drawing.

Characteristics of 3D scene of document:

EXPLODE – state of exploded view mode for the fragments of 3D model of the current document (0 – exploded view mode is turned off, 1 – turned on).

Examples of Using Functions

getv("SCALE",0) – returns the value of the scale specified for the first page of the current document;

getv("SCALE_Page 2",-1) – returns the value of the scale specified for the page of the current document with the name “Page 2”. If this page is absent in the document, the function returns the value “-1”;

getv("mass",0) – returns the value of mass of all bodies in the 3D scene of the current document. If the 3D scene is empty, the function returns the value "0";

setv("SCALE_Front_0",2) – this functions sets the value of the scale equal to 2 for the page "Front_0" (page of workplane "Front view");

setv("explode",1) – this function activates the exploded view mode for the 3D model in the current document.

Using Function TGETV ()

Function tgetv() allows the user to obtain the values of system's text characteristics of the current T-FLEX CAD document.

Syntax of functions:

tgetv ("NAME"), where NAME – the name of text parameter of the system or the current document.

Any of the following characteristics can be used as parameter of this function:

USERNAME – Name of current user of the system;

YEAR – Current year;

MONTH – Current month (number);

DAYOFWEEK – Current day of week (number);

DAY – Current day of month (number);

HOUR – Current hour;

MINUTE – Current minute;

SECOND – Current second;

DATE – Current date (the name) in accordance with system's current settings;

TIME – Current time in accordance with system's current settings;

REGNAME – User name on whom the system is registered;

REGCOMPANY – Name of company on which the system is registered;

TITLE – Title of current drawing;

SUBJECT – Subject of current drawing;

AUTHOR – Name of author of current drawing;

KEYWORDS – Keywords of current drawing;

COMMENTS – Comment of current drawing;

TEMPLATE – Template of OLE document;

LASTAUTHOR – Name of author who last saved the drawing by;

REVNUM – Number of revision of current drawing;

EDITTIME – Total editing time of current drawing;

PRINTDATE – Date of the last print of current drawing;

CREATEDATE – Date of creation of current drawing;

SAVEDATE – Date of last save of current drawing;

FILENAME – Name of current file (the returned line contains full path to current document and the file name with extension);

SHORTFILENAME – Name of current file (the returned line contains the file name only);

FORMAT – Title block of current drawing;

_FORMAT - Format name for the drawing of the page on which the current fragment is inserted;

FORMAT_BOM – Returns format of first "BOM" type page. If there is no such a page, it returns first page format.

SCALE – Scale of current drawing;

_SCALE - Scale for the drawing of the page on which the current fragment is inserted;

SCALEVALUE – Scale of the current drawing (the returned string contains the value of the scale without the letter M);

_SCALEVALUE – Scale of the drawing's page onto which the current fragment is inserted (the returned string contains the value of the scale without the letter M).

PAGENAME – Name of the page on which the current fragment is inserted;

PAGETYPE – Type of the page on which the current fragment is inserted.

Example of using the function tgetv:

$TIME = tgetv ("TIME")

MEASURE ELEMENTS AND RELATIONS BETWEEN THEM

The command **PM: Measure Element or relation between two Elements** serves for measuring various geometrical characteristics (coordinates, length, perimeter, area, volume, etc.) of a 2D or 3D object, as well as relations (distance, angle, etc.) between two objects. A new variable can be automatically created based on a measured parameter, or the value of an existing variable can be modified.

CONDUCTING MEASUREMENTS

The command **PM: Measure Element or relation between two Elements** can be called as follows:

Icon	Ribbon
	Measure → Measure → Measure
Keyboard	Textual Menu
<PM>	Parameters > Measure

After calling the command, the following actions become available:

	<Ctrl+Enter>	Finish input
	<Esc>	Exit command
	<1>	Measure one element parameter
	<2>	Measure relation between two elements
	<3>	Measure Multiple Elements
	<4>	Select LCS
	<C>	Reset target LCS
	<5>	Body measurement mode

At the first step of the command you need to select what will be measured:

- a single object parameters

- a relation between two objects (the option)

- the sum of parameters of several objects.

After activating the selected mode, pick the object or objects to be measured in the 2D or 3D window. The state of element selection filters in the system toolbar affects the selection. The set of filters depends on which type of window is currently active in the system (2D or 3D).

Measuring Parameters of Single Element

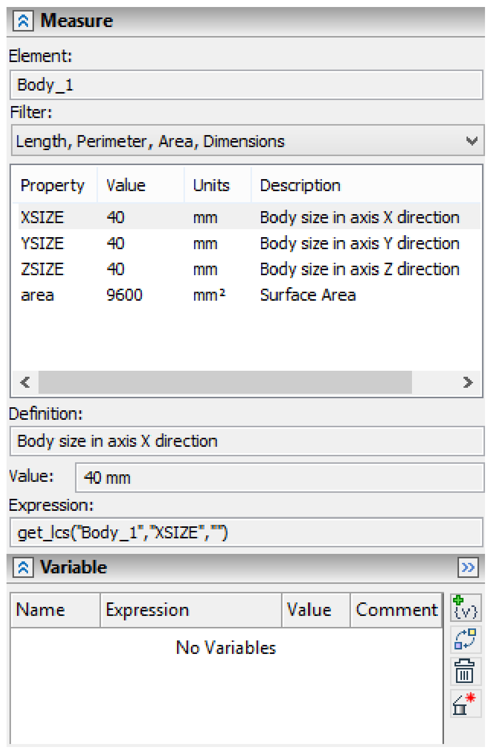
Upon activating this mode and picking the object to be measured, the name of the selected object will be displayed in the **Measure** section of the properties window within the **Element** field, along with its available set of parameters (the table **Property**).

Upon selecting the desired parameter in this list, you can view the following in the additional fields below the parameters list:

- description of the given parameter (for example, “Element length” or “Circle radius”);
- its value in the model units (model units are defined in the command **ST: Set Document Parameters** on the tab **3D**);
- an expression that is used to evaluate the given parameter (for example, get(“0x3000011”,“LENGTH”)).

To create a variable based on the selected parameter, you need:

Select the desired parameter in the list;



In the “Variable” section of the properties window set the radio switch “Create/Replace” in the “Create” state (the default setting);

Enter the name of the variable being created;

Enter a comment for the variable being created in the field “Comment” comment (optional);

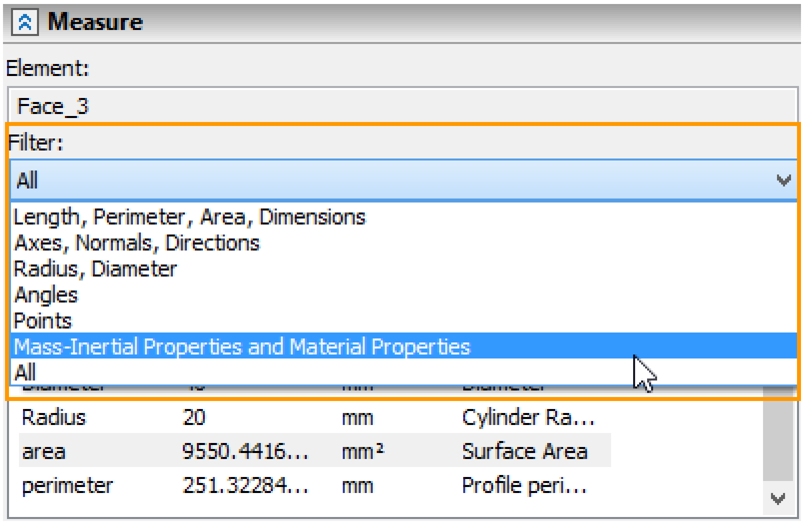
Click the button [Apply].

After that, you can complete the command or create another variable by selecting its respective parameter in the list.

In the case when, instead of creating a new variable, a new expression needs to be defined for an already existing variable, the order of steps is similar to the described, with one exception: the “Create/Replace” switch shall be set in the “Replace” state. The name of the variable being edited is selected from the combo box at the right of the radio switch. The list displace all variables present in the given document (except the hidden ones). Upon clicking the [Apply] button, the old expression of the specified variable will be replaced by the new expression corresponding to the selected parameter.

> All created and edited variables can be viewed in the variable editor (the command “**V: Edit Variables**”).

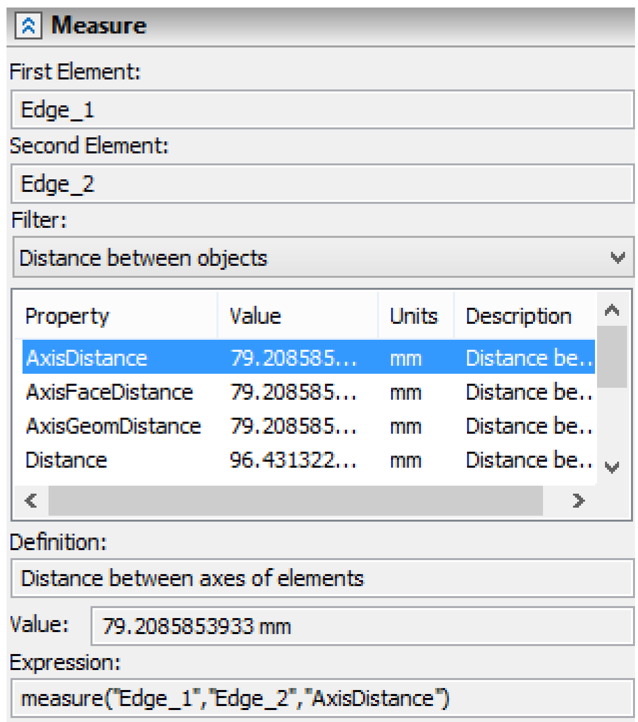
Filter of the measured values allows to display values that satisfy selected filter in the properties window.



Measuring Relations between Two Elements

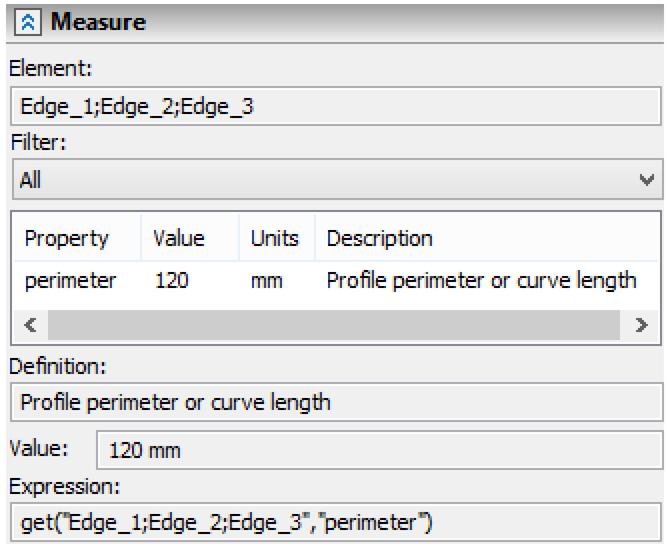
To measure relations between two 2D or 3D objects, activate the mode  and subsequently select two measured objects. The selected objects will be highlighted and entered in the fields **First element** and **Second element** of the properties window. The relations that can be measured on the selected elements appear in the **Relation** table.

In the rest, working in this mode is analogous to the steps when measuring parameters of a single element.



Measure Multiple Elements

To get a summary value of the identically-named characteristics of several elements, it is necessary to select the option.



After this mode is enabled, it is required to select the required objects. In the command's properties window, in the **Element** field the names of the selected objects will be shown. In the **Property** table those summary characteristics which can be calculated for the selected set (for example, total length or mass) will be listed.

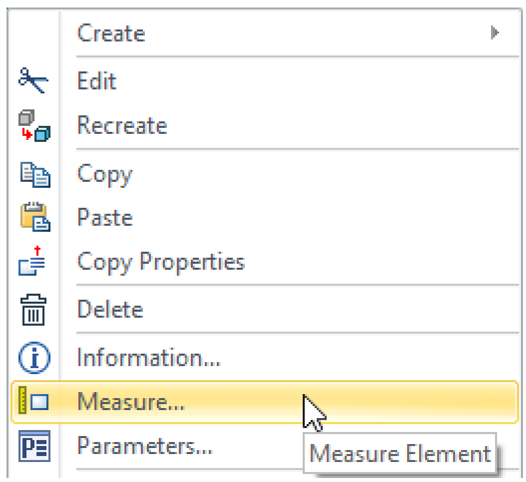
In this way it is possible to measure only the following parameters:

- Mass (total mass of the elements);
- Perimeter (total perimeter of the elements);
- Area (total area of the elements);
- Length (total length of the elements).

Additional Methods of Calling Command

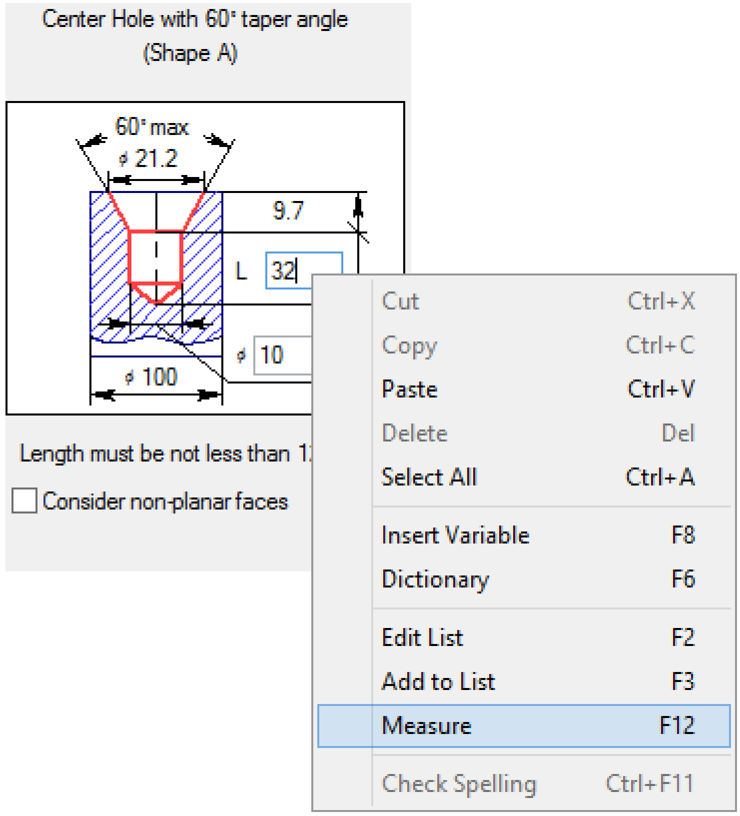
Calling Command from Context Menu

This method is convenient when you need to measure parameters of a single object. For this, select an object in the 2D or 3D window, whose parameters are to be measured, and right-click. In the coming up context menu select the item **Measure…**. This will result in launching the command **Parameters** > **Measure** in the mode of measuring parameters of a single object. The 2D or 3D object over which the context menu was accessed gets automatically selected for conducting measurements. Further steps within the command are fully identical with the described above.



Calling Command in Transparent Mode when Defining Parameters of 2D or 3D Elements

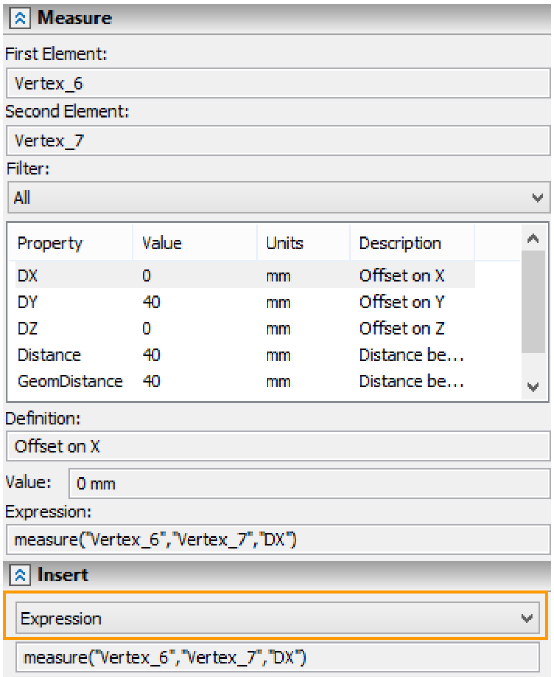
When defining numerical parameters of any T-FLEX CAD element (construction or graphic line, 3D node or operation, etc.) one could need to relate one of the parameters with a certain geometric characteristics of the given model. This could be, for example, the distance between two of the model elements or the length (angle, perimeter, area etc.) of another element.



To define such relation, simply set the course are in the input field of the respective parameter in the properties window or the parameters dialog and press <F12>. Alternatively, you can access the context menu at this moment (by right clicking) and select the **Measure** item in it. The properties window (or parameters dialog) of the element being edited temporarily disappears, while the command **PM: Measure Element or relation between two Elements** is launched.

Next, you need to select the command mode (measuring one or two objects), the object or objects to measure and the desired measurement parameter.

Using the **Expression, Value, Variable** drop-down list, specify in what form the value of the selected parameter will be returned: as an expression (using the function get(), distance() etc.), variable or a constant. Clicking completes the command. After that, the original parameters dialog reappears on the screen, that launched the command. It will already have the geometrical parameter value or expression that was used to evaluate it, in the parameter input field.



ADDITIONAL FEATURES OF THE MEASURE COMMAND

If you call the command when designing a 3D model additional options become available.

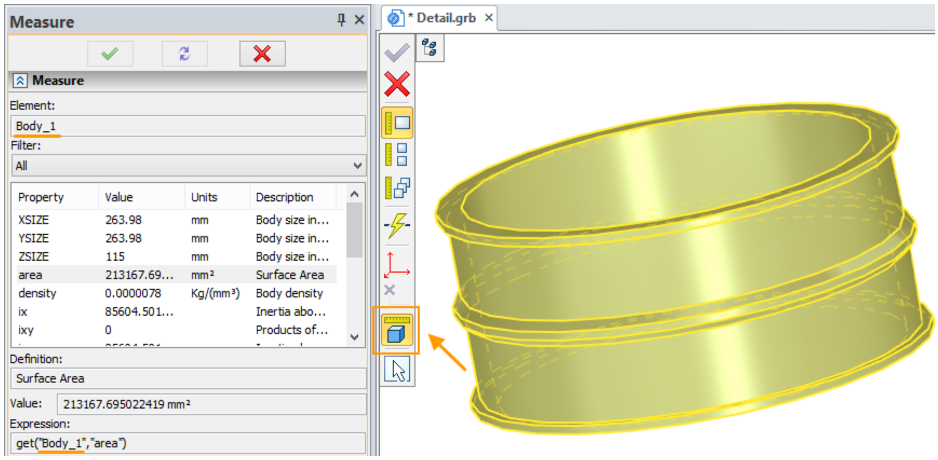
	<N>	Select objects with rectangle

Body Measurement Mode

Body measurement mode allows to consider possible changes in the once measured body.

It is available for single body measurement mode . You need to select option and select a body in the scene.

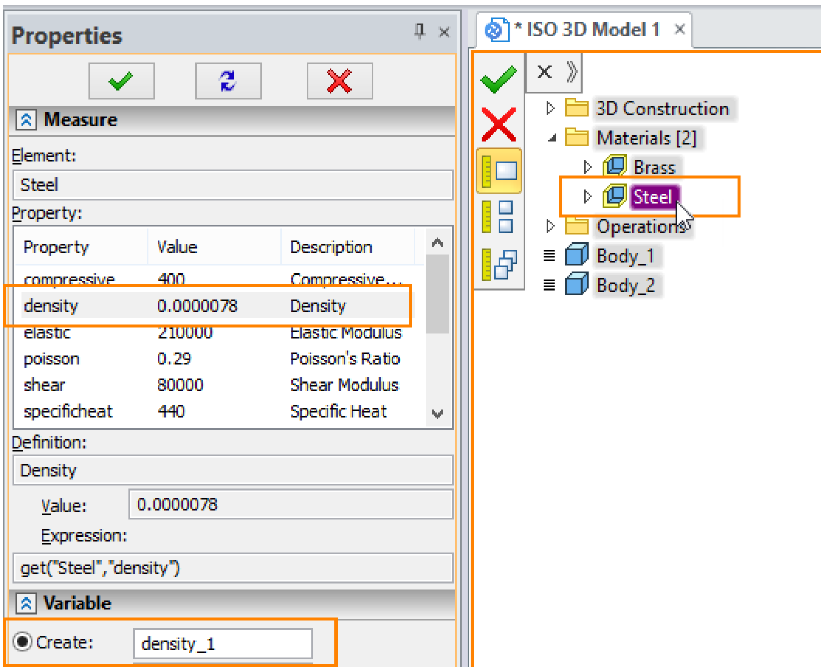
	<5>	Body measurement mode



For example, you measure the area of a body and assign variable for it. Then you apply Boolean operation to the body and the area changes. In this case the corresponding variable will change as well.

Material Properties

The **Measure** command is available for materials of 3D models. For measurement of material properties, it is necessary to activate the command and to choose material in the 3D model tree.



The measured values of materials physical properties can be saved as variables. You can use them in calculations so that there is no need to enter them manually.

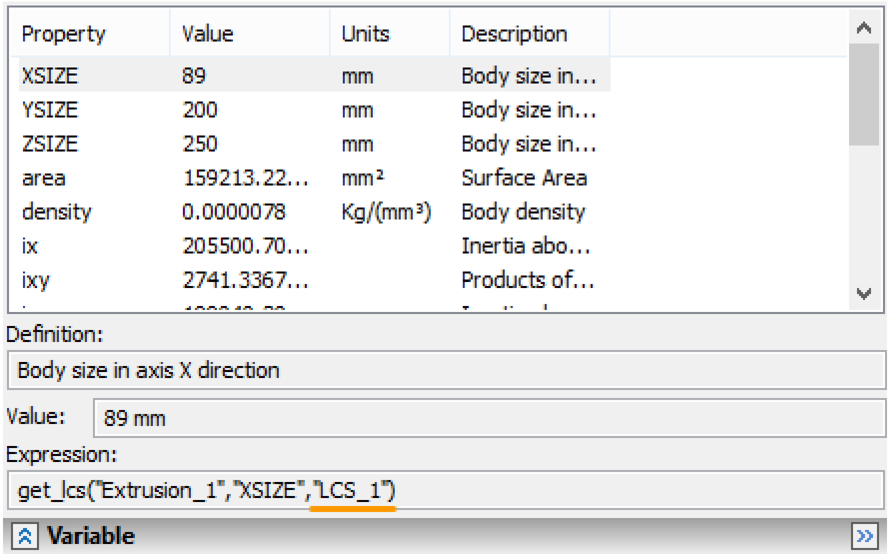
Measurements Relative to LCS

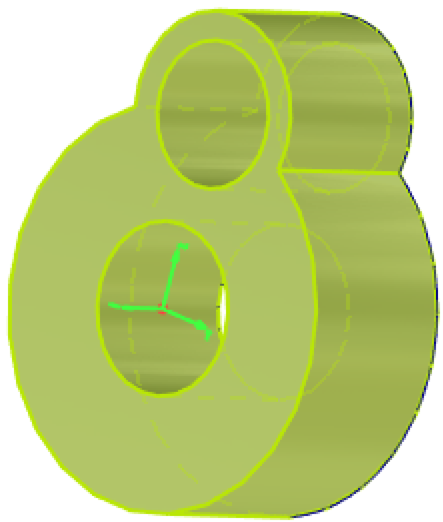
You can measure bodies according to the selected LCS using option:

	<4>	Select LCS

If a single body was selected, the body will be measured relative to the selected LCS.

For example, we have an exported body. If you want to receive overall dimensions of the body, you need to create an LCS and measure dimensions relative to the LCS.





If several bodies are selected, you can measure distance between them relative to the “X” axis of the coordinate system.

For example, you need to measure distance between body vertexes relative to the “X” axis.

First, you need to create a LCS. The measurement will be performed with respect to its X-axis.

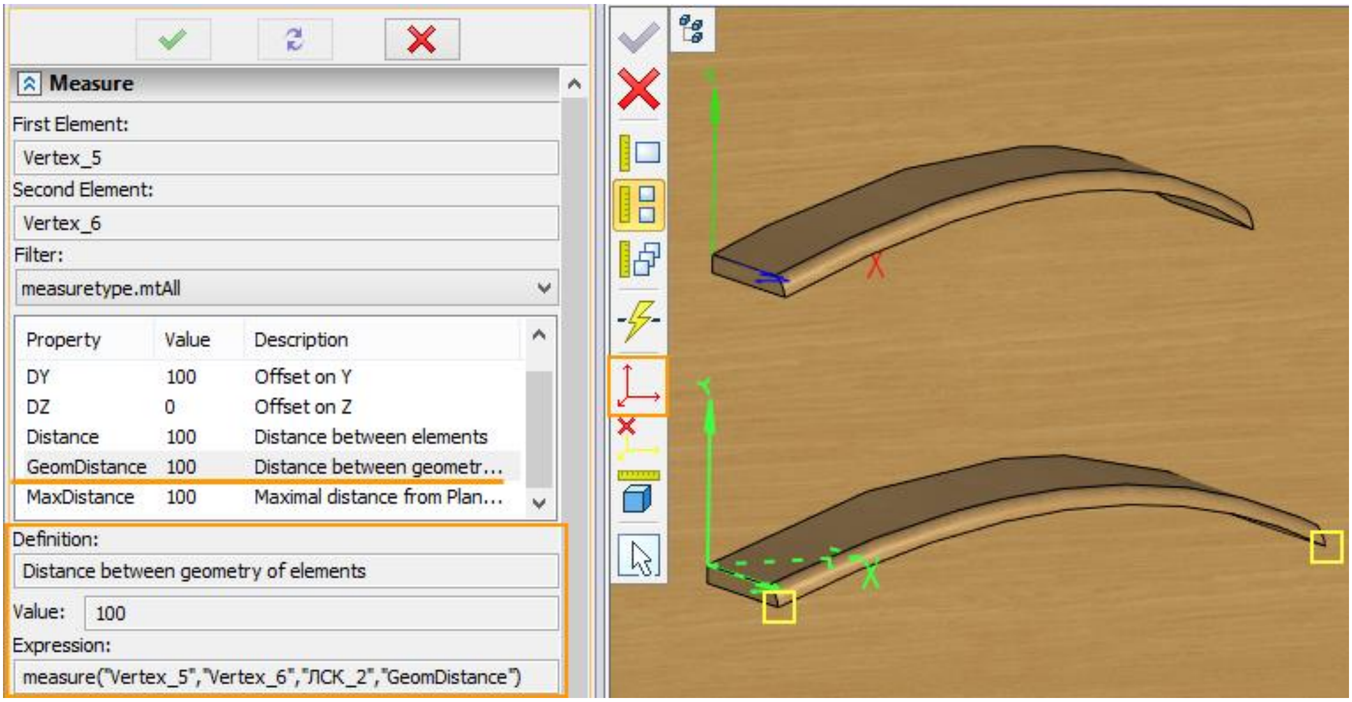
After that select **Measure relations between two elements** option and vertexes of the body.

Then activate **Select LCS** option and select the already created LCS.

The selected elements are projected on the X-axis of the specified local coordinate system, and then the distance between them is measured.

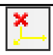
In the current example, when measuring the distance between two vertices (Vertex_5 and Vertex _6), these vertices are projected on the X-axis of the specified coordinate system (LCS_2).

This possibility is helpful, For example, when you want to measure objects of irregular shape.



You can specify variable for the measured value.

To reset target LCS use option:

	<C>	Reset target LCS

Measurable Parameters and Relations

Measurable Parameters

The set of parameters that can be measured by the command **PM: Measure Element or relation between two Elements** depends on the selected object of measurement. Follows below is the list of parameters for all 2D and 3D objects.

> The command uses the function get() for measuring parameters. Syntax description and examples of standalone use of this function are provided in the chapter "Variables".

2D elements

2D nodes:

"X" – X-coordinate of the node;

"Y" – Y-coordinate of the node;

Construction entities:

Straight lines:

"X", *"Y"* - coordinates of the first point through which the construction line passes;

"P1", *"P2"* - coordinates of the second point through which the construction line passes;

Circles and ellipses:

"LENGTH" – length of the circle or ellipse;

"P1" – circle radius (this parameter is specific to circles);

"X", *"Y"* – coordinates of the circle or ellipse center;

Splines, 2D paths, functions and offsets:

"LENGTH" – element length;

Graphic entities:

Straight segments:

"LENGTH" - segment length;

"START_X", *"START_Y"* – coordinates of the segment start point;

"END_X", *"END_Y"* – coordinates of the segment end point;

Circles:

"LENGTH" - arc length;

"CENTER_X", *"CENTER_Y"* – circle center coordinates;

"ANGLE" - angular length of a graphic entity;

"RADIUS" – arc or circle radius;

Ellipses; splines; curves constructed by 2D path, offsets and functions:

"LENGTH" – element length;

for *graphic lines constructed as a circular or elliptical arc, portion of spline, 2D path, offset or function*, there are following additional parameters:

"START_X", *"START_Y"* – coordinates of the arc start point;

"END_X", *"END_Y"* – coordinates of the arc end point;

Hatches:

"AREA" – hatch area;

"PERIMETER" - hatch perimeter;

"XMASS" - X-coordinate of center of mass;

"YMASS" – Y-coordinate of center of mass;

"XAREAMOMENT" – Ix component of inertia moment;

"YAREAMOMENT" - Iy component of inertia moment;

"PRODUCTAREAMOMENT" -Centrifugal inertia moment of area;

"XINERTIARADIUSVALUE" - Radius of inertia X;

"YINERTIARADIUSVALUE" - Radius of inertia Y;

"XAREAMOMENTMAINVALUE" - Ix component of principal moment of inertia relative to mass center;

"YAREAMOMENTMAINVALUE" - Iy component of principal moment of inertia relative to mass center;

"MAINAXESROTATIONVALUE" - Rotation angle of principal axes.

Text:

"HEIGHT" – text height;

"WIDTH" – text width;

"X", *"Y"* – X and Y coordinates of text fixing point;

"TEXT" – contents of the text;

2D fragments: fragment parameters are defined by the values of the fragment's real (numerical) variables. In addition, the following parameters are available for 2D fragments:

"BoundBoxLeft" – left coordinate of bound box (X-coordinate);

"BoundBoxRight" – right coordinate of bound box (X-coordinate);

"BoundBoxTop" – top coordinate of bound box (Y-coordinate);

"BoundBoxBottom" – bottom coordinate of bound box (Y-coordinate);

"BoundBoxCenterX" –X-coordinate of bound box center;

"BoundBoxCenterY" –Y-coordinate of bound box center.

Dimensions:

"FIT" – fitting dimension;

"LOWER_DEVIATION" – lower value of tolerance;

"UPPER_DEVIATION" – upper value of tolerance;

"TEXT_BEFORE" – text before (the dimension's value);

"TEXT_AFTER" – text after (the dimension's value);

"TEXT_UNDER" – text under (the dimension's value);

"TOLERANCE" – dimension's tolerance;

"VALUE" – dimension's value;

Leader Notes:

"INSCR_TEXT" – leader note text;

"INSCR_TEXT_UNDER" – text under leader of the leader note;

"TEXT_ON_LEADER" – text on the arrow;

"TEXT_UNDER_LEADER" – text under the arrow;

"INSCR_ZONE" – place;

2D connectors – *connector's values are returned as connector's characteristics*.

3D objects

Operations:

"Area" – surface area;

"Mass" – body mass (according to the material density);

"Xmass", "Ymass", "Zmass" - X,Y,Z-coordinates of the center of mass;

"IX", "IY", "IZ" – Moments of inertia with respect to the axes X, Y, Z;

"IXY", "IYZ", "IZX" – Inertia value with respect to the planes XY, YZ, ZX;

"Volume" – volume;

for *"Apply Material"* operation the following additional parameter is used:

"MaterialArea" – total area of faces with the attached material;

for all *3D Arrays (except arrays of faces)* the following additional parameter is used:

"CopyCount" – actual number of copies in the array (with account of limitations and exclusions);

3D nodes:

"POINTX", "POINTY", "POINTZ" - X, Y, Z-coordinates of a 3D node;

3D profiles:

"Area" - surface area of a 3D profile;

"Perimeter" - *perimeter* of a closed 3D profile or length of an open one;

3D paths:

"Perimeter" – length of a 3D path;

Faces:

"Area" - surface area;

"Perimeter" - face perimeter;

for a *flat face* the following additional parameters are used:

"LocationX", "LocationY", "LocationZ" – X, Y, Z-coordinates of the plane origin;

"NormalX", "NormalY", "NormalZ" – X, Y, Z-coordinates of a flat face normal;

"RefDirectionX", "RefDirectionY", "RefDirectionZ" – X, Y, Z-coordinates of a plane vector (the plane vector defines the direction of the X-axis of a flat plane);

for a *cylindrical face* the following additional parameters are used:

"Radius" – cylinder radius;

"Diameter" – cylinder diameter;

"CenterX", "CenterY", "CenterZ" – X, Y, Z-coordinates of the center;

"AXISX", "AXISY", "AXISZ" – X, Y, Z-coordinates of the cylinder axis;

for a *toroidal face* the following additional parameters are used:

"*MaxRadius*", "*MinRadius*" – major and minor torus radii;

"*CenterX*", "*CenterY*", "*CenterZ*" – X, Y, Z-coordinates of the torus center;

"*AxisX*", "*AxisY*", "*AxisZ*" – X, Y, Z-coordinates of the torus axis;

for a *spherical face* additional characteristics are used:

"*Radius*" – sphere radius;

"*Diameter*" – sphere diameter;

"*CenterX*", "*CenterY*", "*CenterZ*" – X, Y, Z-coordinates of sphere center;

Edges:

"*Perimeter*" – edge length;

"*StartX*", "*StartY*", "*StartZ*" – X, Y, Z-coordinates of the start point;

"*EndX*", "*EndY*", "*EndZ*" – X, Y, Z-coordinates of the end point;

for a *straight edge* the following additional parameters are used:

"*VECTORX*", "*VECTORY*", "*VECTORZ*" – X, Y, Z-coordinates of the segment direction;

for a *circular edge* or *along circular arc*:

"*Radius*" – circle radius;

"*Diameter*" – circle diameter;

"*CenterX*", "*CenterY*", "*CenterZ*" – X, Y, Z-coordinate of the circle center;

"*AxisX*", "*AxisY*", "*AxisZ*" – X, Y, Z-coordinates of the circle plane normal;

"*StartAngle*" – arc start angle;

"*EndAngle*" – arc end angle;

additional parameters for the *elliptical edge* are:

"*MajorRadius*" – ellipse major radius;

"*MinorRadius*" – ellipse minor radius;

"*CenterX*", "*CenterY*", "*CenterZ*" – X, Y, Z-coordinate of the ellipse center;

"*AxisX*", "*AxisY*", "*AxisZ*" – X, Y, Z-coordinates of the normal to the ellipse plane;

"*MaxDirectionX*", "*MaxDirectionY*", "*MaxDirectionZ*" – X, Y, Z-coordinates of the ellipse major axis;

"*MinDirectionX*", "*MinDirectionY*", "*MinDirectionZ*" – X, Y, Z-coordinates of the ellipse minor axis;

"*StartAngle*" – start angle of the elliptical arc;

"*EndAngle*" – end angle of the elliptical arc;

Loops:

"*Perimeter*" – loop perimeter.

3D connectors – connector's values are returned as connector's characteristics.

Measured Relations

The list of relations that can be measured by the command **PM: Measure Element or relation between two Elements** depends also on the selected objects of the measurement. Follows below is the list of relations, with specified pairs of 2D and 3D objects, for which such relations can be defined.

> The functions distance() and measure() are used in this command for measuring parameters. The syntax of these functions is described in the chapter "Variables".

2D elements

"Distance" – is the distance between a 2D node and another 2D node, construction line, graphic line or hatch (the order of selecting the measurable entities is not significant);

"Angle" – is the angle between two lines, segments or a line and a segment.

"DX" – Offset on X;

"DY" – Offset on Y;

3D objects

"Distance" – is the distance between two arbitrary 3D objects that are 3D construction entities (except for LCS), operations or such topological objects as an edge, loop, face, vertex.

"GeomDistance" – is the distance between 3D points, 3D curves or surfaces corresponding to two respective 3D objects of the types: 3D node, vertex, edge, face.

"Angle" – is the angle between directional vectors of two 3D objects. Listed below are 3D objects, for which a direction can be defined (and, therefore, this relation can be calculated). Additionally specified is what will be selected as the direction vector for each object:

for a 3D path, edge or open 3D profile lying on a straight line – the line direction;

for a 3D path, edge or open 3D profile lying on an ellipse (circle) – the vector directed from the center of the ellipse (circle) normal to the plane of the ellipse (circle);

for a flat 3D profile; workplane; flat face; an operation body consisting of one face lying in a plane – the normal to the plane;

for a cylindrical work surface; cylindrical face; 3D profile lying on a cylinder; an operation body consisting of one face lying on a cylinder – the axis of the cylinder;

for a 3D profile or face, lying on a cone; an operation body consisting of one face lying on a cone – the axis of the cone;

for a toroidal work surface; 3D profile or face lying on a torus; an operation body consisting of one face lying on a torus – the axis of the torus.

"AxisDistance" – is the distance between the axis of two 3D objects. The same 3D objects can be selected as the objects of the measurement as in the previous case (when identifying *"Angle"*),

except for workplanes. In the latter case, the axes of the selected objects coincide with the directional vectors of the planes.

"*DX*" – Offset on X;

"*DY*" – Offset on Y;

"*DY*" – Offset on Z.

"MaxDistance" – Maximum distance between two 3D objects (between two points the most distant from each other).

"MaxGeomDistance" – Maximum distance between two geometric *elements (*3D points, 3D curves or surfaces).

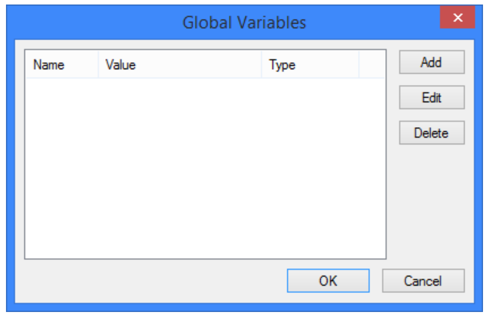
Global Variables

A global variable is a named value of one of the two types: text (string) or real. Global variables are accessible for editing in the variable editor or via the special functions across all currently open documents. The list of global variables and their values are saved automatically upon exiting the application (in the registry) and are restored upon launching the application.

Global variables are created by the command **SG: Create/Edit Global Variables**. The command is called by one of the following means:

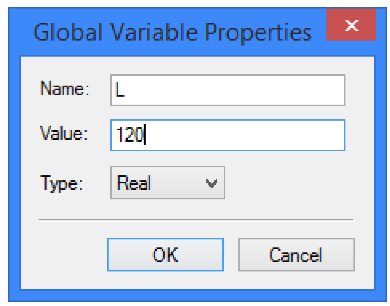
Icon	Ribbon
	Parameters → GUI Control → Global Variables
Keyboard	Textual Menu
<SG>	Parameters > Global Variables

Upon calling the command, the dialog box is displayed on the screen where you can create new global variables or delete and edit the existing ones. To create a new variable, press the graphic button **[New]**. In the coming up dialog box defining the name, value and type of the variable being created.



Upon the confirmation, the entered data will be displayed in the command dialog box.

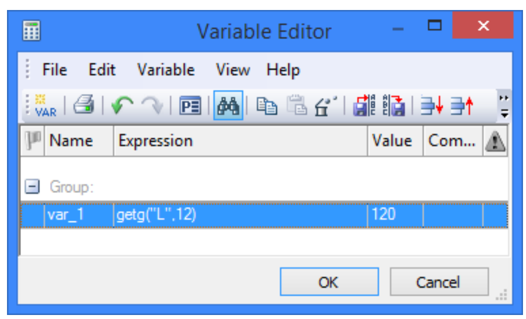
Created global variables will be stored until you delete those or update your system installation (the global variables are stored in the system registry). A global variable can be accessed from any document managed by one user.



You can retrieve the value of a global variable in any dialog that allows use of variables, by the following special functions:

getg ("Name",N) – gets the value of a real global variable;

tgetg ("Name",N) – gets the value of a text global variable.



The first argument defines the name of the created global variable. The expression after the comma defines the value to be used in the case if the global variable is not found.

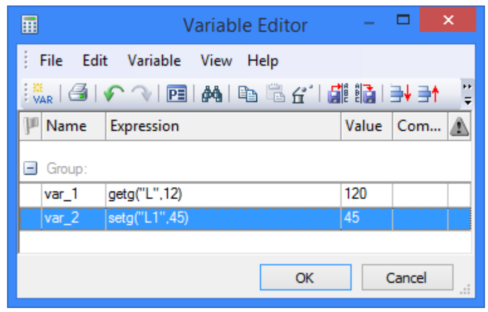
Exercise: create a new variable of a certain type (real or text) in the variable editor within a drawing, and then make a function call for accessing a global variable of this type. If the global variable with the specified name (“L”) exists in your user registry, then its value will be displayed in the “Value” column. If the specified global variable is not found then the second function argument will be displayed in the “Value” column.

Besides that, a global variable can be created, or its value modified, in the common variable editor or in any dialog that allows use of variables. The following functions are provided for this purpose:

setg (“Name”,N) – sets the value of a real global variable;

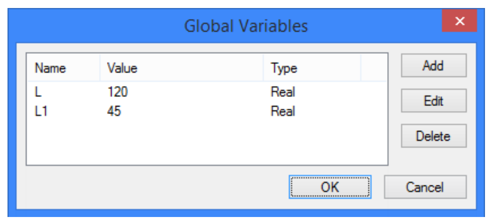
tsetg (“Name”,N) – sets the value of a text global variable.

In this case, the first argument is the name of the global variable, and the expression in the second argument defines the global variable value.



Exercise: create a new variable (*“var_2”*) in the variable editor, and assign it to the function setting the value of a global variable.

If a global variable with the specified name already exists, then its value will be modified. If the variable with the specified name does not exist, then it will be created and displayed in the main pane of the Global Variables dialog box. If a global variable is defined in the drawing by one of the functions, setg or tsetg, then opening this drawing in a different user account (on a different computer) creates the global variable on that account/computer automatically.

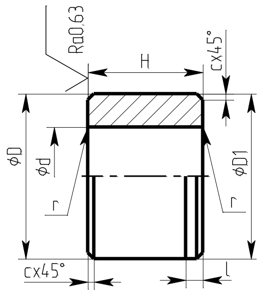


Note that global variables can be used, for example, for automatic creation of drawing documentation. Suppose, the user creates a global variable - the developer's name. The value of this variable can be entered in the title block of the drawing by using the described functions. Thus, a portion of the title block template will be filled in automatically as you insert the title block.

DATABASES

T-FLEX CAD has a capability of creating databases. The usage of databases in T-FLEX CAD allows us to realize the whole catalogs of products in one single drawing. It is possible to create elements of structures by specifying their parameters from the databases.

The database of parameters of the bushing can serve as a good example. In T-FLEX CAD there is no need to create several separate drawings for bushings of different diameter. It is sufficient to construct parametric model of the bushing and obtain different modifications of the bushing by specifying the corresponding values from the database for the parameters of construction elements. The process of creation of the database with bushing's parameters is described below.



GOST 12215-66

Bushing GOST code 12215-66	d	H	D	D1	l	R = c	Weight, kg per GOST 12215-66
7030-0172	4	6	8	8	1,2	0,2	0,002
7030-0173	6	8	10	10	1,5	0,6	0,003
7030-0174	8	10	12	12	1,5	0,6	0,005

In T-FLEX there are two ways of storing the data. The first method – store data in the external file of one of the standard formats (for example, MDB format). Such files can be created with the help of T-FLEX CAD system as well as any other programs designed for this purpose. The second method – store data inside a specific drawing. Databases that are stored together with the drawing are called **internal databases**, all other databases are called **external databases**.

There are several variations of how to work with **external databases**. The way of working with the external database upon which, on the basis of external data file, the database-copy is created in T-FLEX CAD document, which retains connection with the external source-file, is called **database by reference**. The contents of the database by reference can be updated from the source-file, automatically or upon user's request. In addition, if the external file is absent the parametric model continues to work using the copy of the database inside the document. When the source-file again becomes available, connection with it is automatically updated. The given way of working with external data files allows different parametric models to refer to the same database.

Another way of working with the external databases is possible upon which the database is not directly downloaded into the T-FLEX CAD document. In this case to get an access to the database content the functions are used.

GENERAL INFORMATION

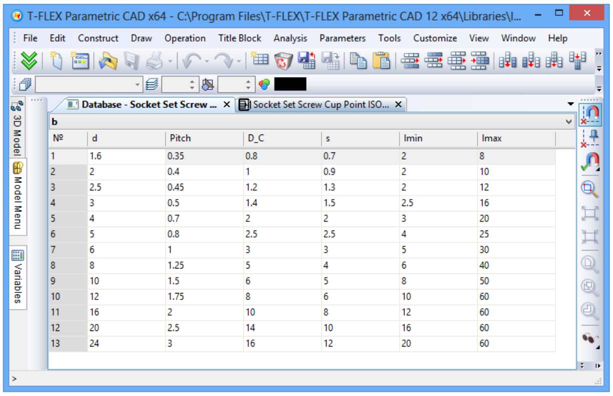
For creation and editing internal databases and databases by reference we use the command **ID: Edit database**:

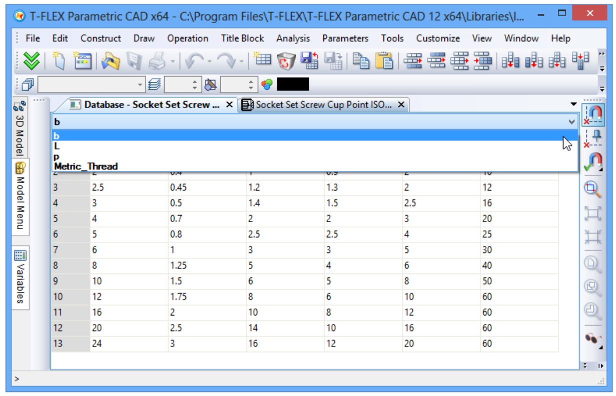
Icon	Ribbon
	Parameters → Tools → Database
Keyboard	Textual Menu
<ID>	Parameters > Database

The command offers the following **capabilities**:

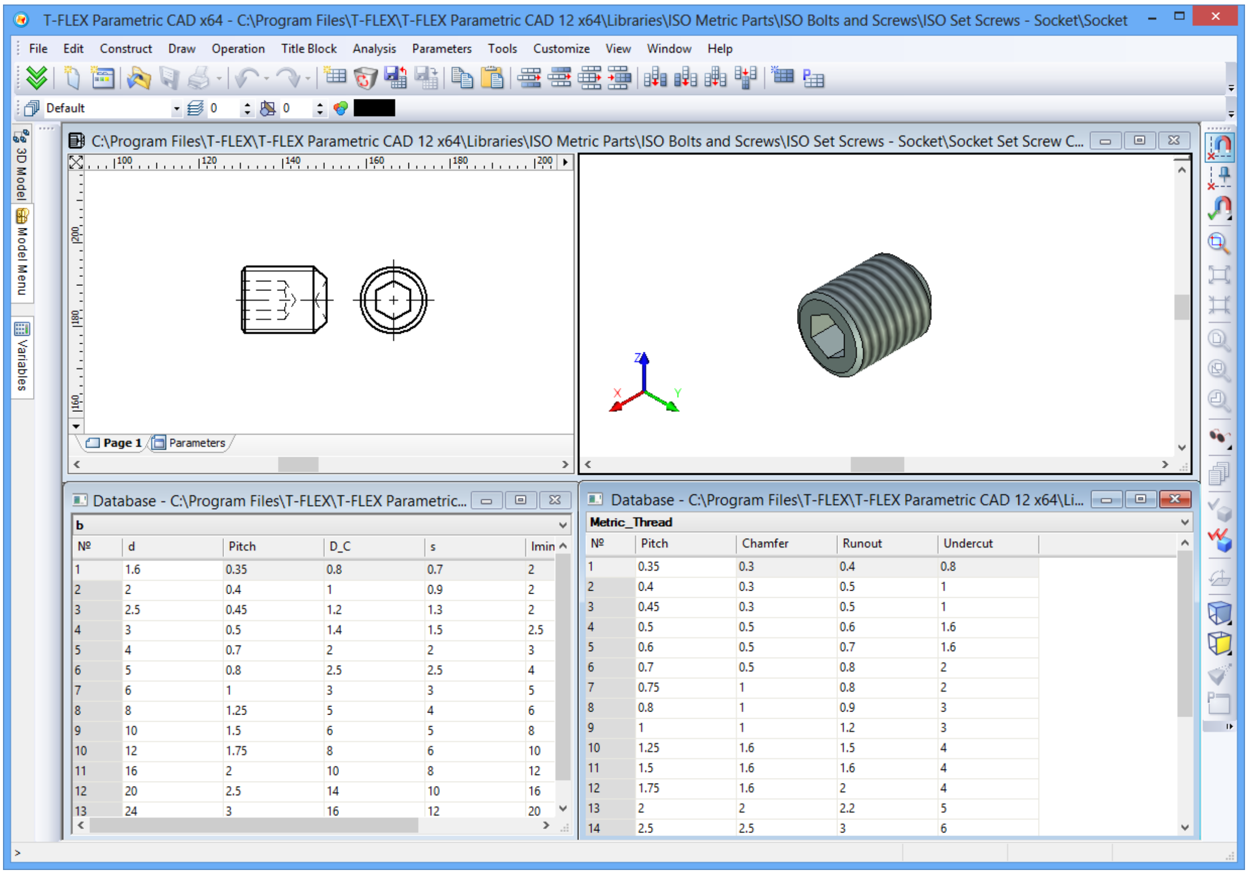
- create new internal database;
- create database by reference to external file;
- edit the contents and the header of the existing database;
- save the internal database into the external file of the format dBase or Access;
- transform the database by reference to the internal database;
- delete the database.

When invoking the **Parameters** > **Database** command, additional window of the current document is created in which the database editor is displayed. The window contains a table of the database (if the document does not have a database, the table will be empty) and the list of databases of the current document with the help of which it is possible to switch from one database to another.

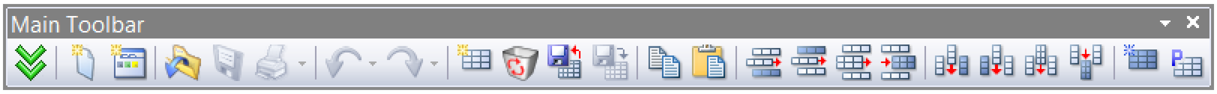




Owing to the fact that the database editor dialog opens in a separate window, it is possible to simultaneously work in the main window of T-FLEX CAD document and in the editor's window, and also simultaneously edit several databases of a single document.



All main commands for working with the database are included into a special group of the main toolbar – **Database** (it becomes available when working with the database editor):



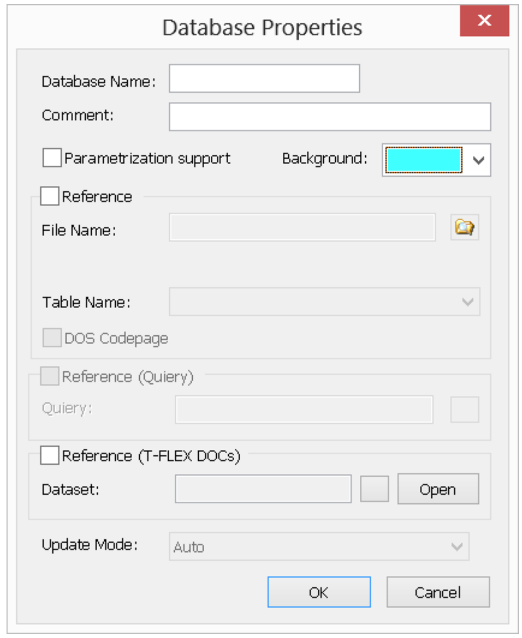
		Create new database
		Remove current database
		Save database into a file
		Update contents of database (only for databases by reference)
		Copy to buffer
		Copy database to buffer
		Insert from buffer
		Insert a row before
		Insert a row after
		Insert rows
		Remove rows
		Insert a column to the left
		Insert a column to the right
		Insert columns
		Remove columns
		Move downwards
		Move upwards
		Sorting
		Select table
		Table's properties

If the current drawing does not have other databases, then only the "Create new database" command is available. With the help of this command, it is possible to create new internal database or a database by reference.

Creating Internal Database

Creating New Database

To create a new database, invoke the option on the main toolbar. In the "Database properties" window that appears, specify the database properties and comments (if necessary).

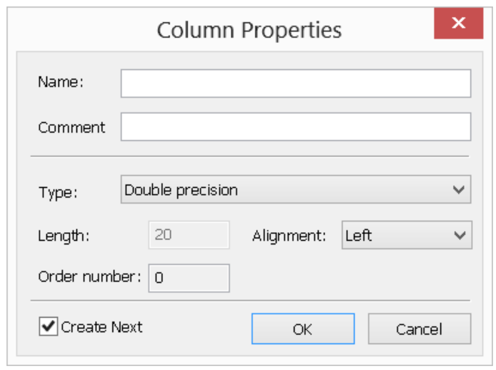


Each database has its own unique name. The name of a database – a string of symbols (no more than 10). It is important to pay attention to the fact that the upper-case and lower-case letters when entering the name lead to creation of different names. The name BASE_1 and base_1 – different names. The same is true for the names of columns of the database.

It is also possible to specify the background color for the cells of database's table with the help of the parameter "Background". When creating new database the background color is selected in a special field located in the upper part of the "Database properties" window.

Different databases can have different background color. When opening another database, the color of the background will change.

After pressing [OK] the "Database properties" window will close. The system automatically transfers to the database table creation mode. The dialog window for specifying parameters of the first column of the created database will appear on the screen:



Name. Defines the name, by which the access to the values of a database will be carried out. A field is identified by its name. The length of the field name should not be greater than 10 characters. The field name can be an arbitrary string of letters, numbers and the underline sign (_). The string should begin with a letter. All field names within one database should be unique.

Comments. Explanatory text for the column of length of no more than 80 symbols.

Type. Defines the information representation format for the given column. Columns can be of one of the following types:

Integer. In this column you can input only integers. The range is from −32768 to 32767 (16-digit sign integer);

Long integer. Integers in the range between −2147483648 and 2147483647 (32-digit sign integer);

Float. In this column you can input only real numbers. Admissible range is from -3.4×10^{38} to $+3.4 \times 10^{38}$ (7 digits);

Double float. Real number in the range from $\pm 5.0 \times 10^{-324}$ to $\pm 1.7 \times 10^{308}$ (15–16 digits);

Text. In this column it is possible input any text information.

Length. Defines the maximum admissible number of symbols when entering and editing the value in a column of the *textual* type.

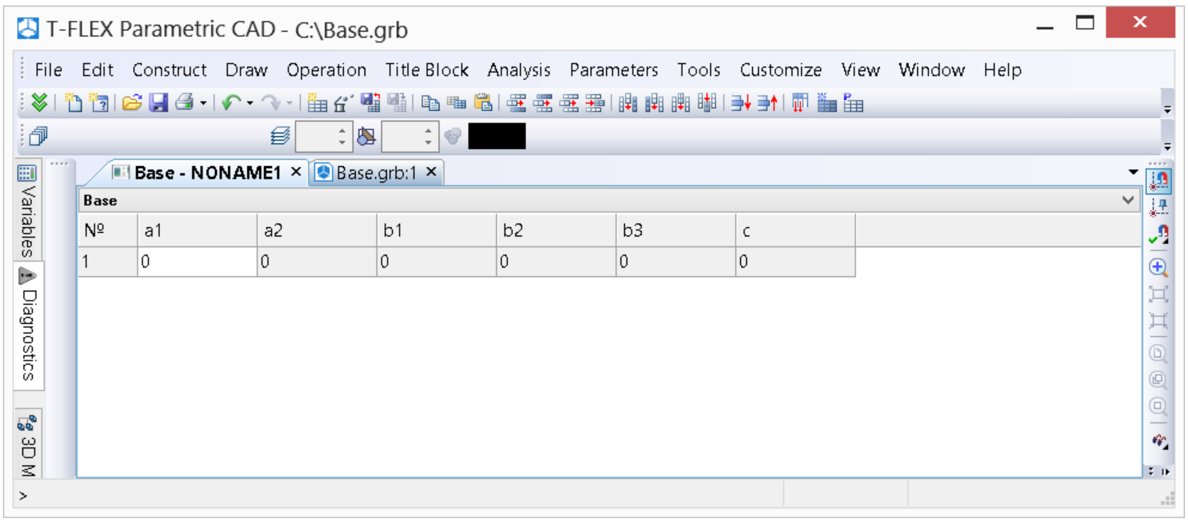
Alignment. This parameter defines the way of displaying the values in a column: with alignment by the left margin of the column, right margin or the center.

Order number. Shows order number of chosen column in database.

If the **Create next column** flag is selected, after entering the values and pressing **[OK]**, the column's properties dialog will appear again which will allow us to specify parameters of the next column, etc. If the flag is disabled, after closing the dialog window the system will transfer to the mode of filling in the database.

Filling in and Editing the Database

After specifying parameters of the database and its columns, on the screen will appear an empty table of the database that contains one line and those columns which were specified by the user.



> Note that in the database table there is always an additional, service column with the name "№". This column contains line numbers of the table.

To create additional lines, it is possible to use the options of the main toolbar:

		Insert a line before
		Insert a line after

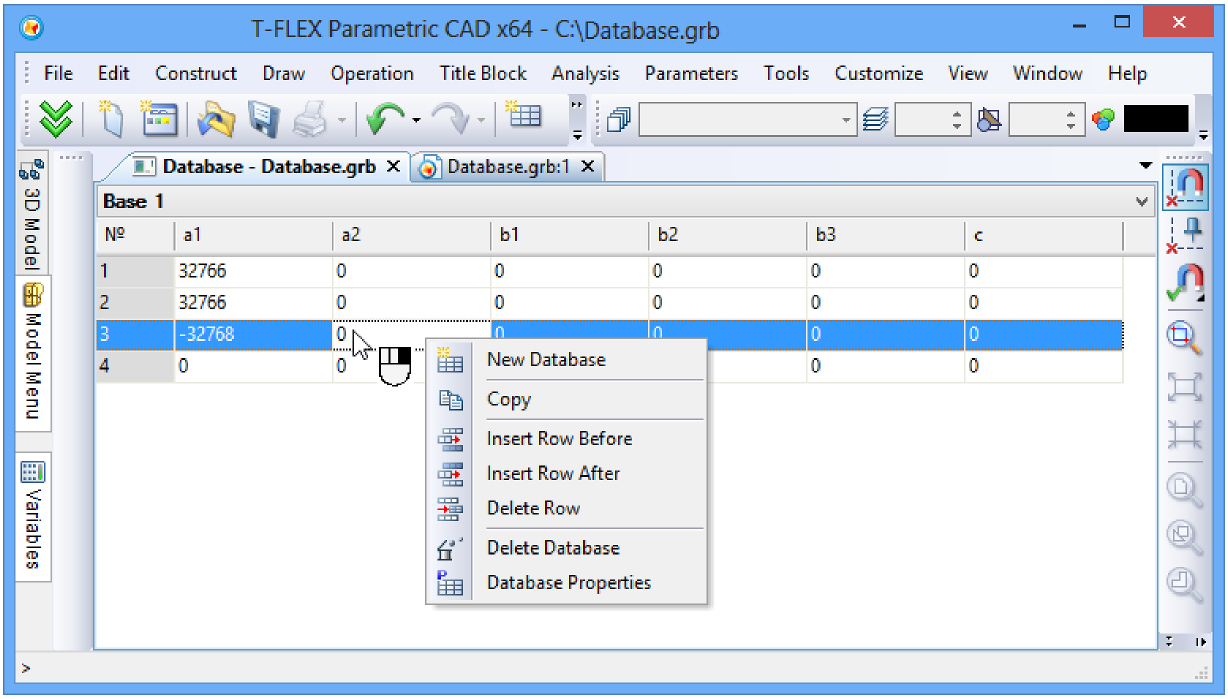
		Insert lines

Also, you can create an empty line if you press <Enter>, when the cursor is located at the last line of the table.

To remove the lines, use the option:

		Delete lines

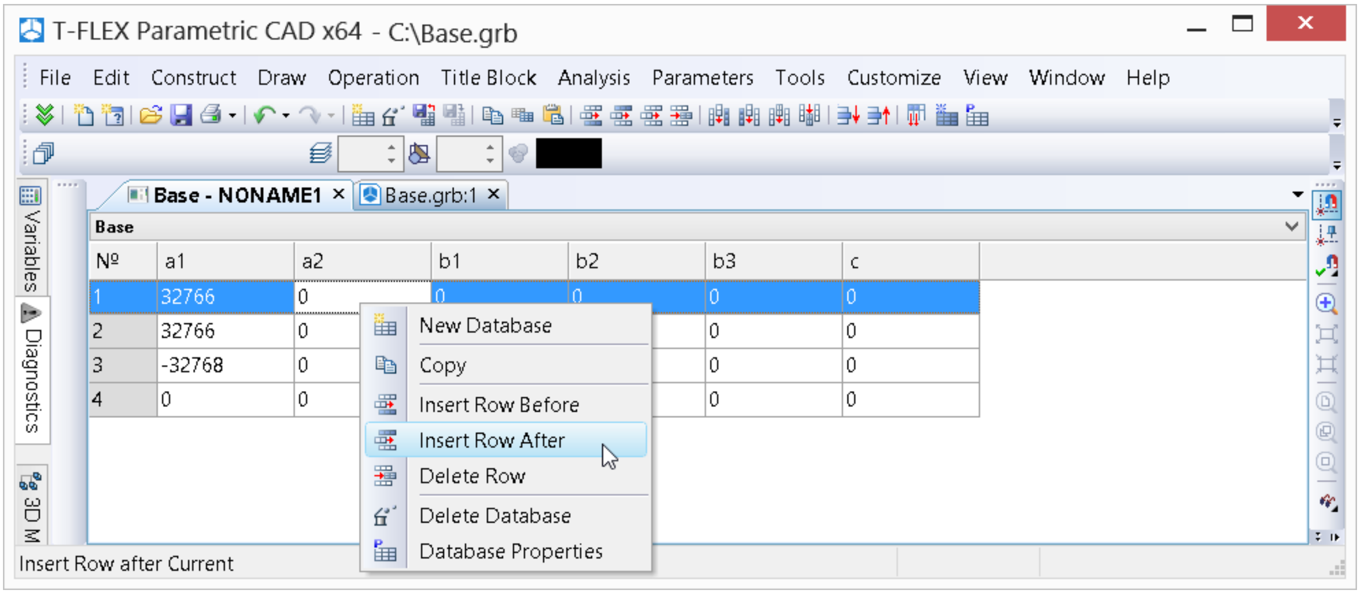
All options are repeated in the context menu that can be invoked with for any string of the table (out of the mode of editing the contents of a specific cell of the table).



You can add and remove the columns to/from the database table by using the following options of the main toolbar:

		Insert column to the left
		Insert column to the right
		Insert columns
		Remove columns

The same options are accessible from the context menu in the area of columns' headers of the database table:



In addition, in the context menu the following commands for a column are available:

– **Sort in the ascending order**. As a result of applying this command, the lines of the table are displaced in such a way that the contents of the cells of the current column are sorted in the ascending order;

– **Sort in the descending order**. As a result of applying this command, the lines of the table are displaced in such a way that the contents of the cells of the current column are sorted in the descending order;

– **Select width**. Selects the width of the current column according to the contents of the column's cells;

– **Column's properties**. Invokes the dialog of column's parameters (the same as used upon creation of a database). In the dialog you can change the name and the type of already existing and completed column. It should be taken into account that upon changing the column type its contents can be lost. In addition, the change in the column's name to which the links were already created in the variables' editor, will lead to the errors in the variables, which can be corrected only by the user.

The column's parameters dialog can also be invoked without using the context menu, i.e., simply by clicking on the column's header.

Moving across the lines when editing the contents of the database is carried out in the following ways:

– With the help of pointing at the required string with the cursor and pressing ;

– With the help of pressing the keys <Up> or <Down>. In this case, the cursor moves one line up or down, respectively;

- With the help of the key <Enter> in any line, except the last one. As a result, the cursor moves to the next line;
- With the help of pressing the keys <PageUp> or <PageDown>. In this case, the cursor moves one page up or down, respectively, i.e., to the first/last string in the current window;
- With the help of pressing the keys <End>/<Home>. In this case, transition to the first/last lines of the database table is carried out.

The user can move rows and the data contained in them inside the database table with the help of the following options:

		Move upwards
		Move downwards

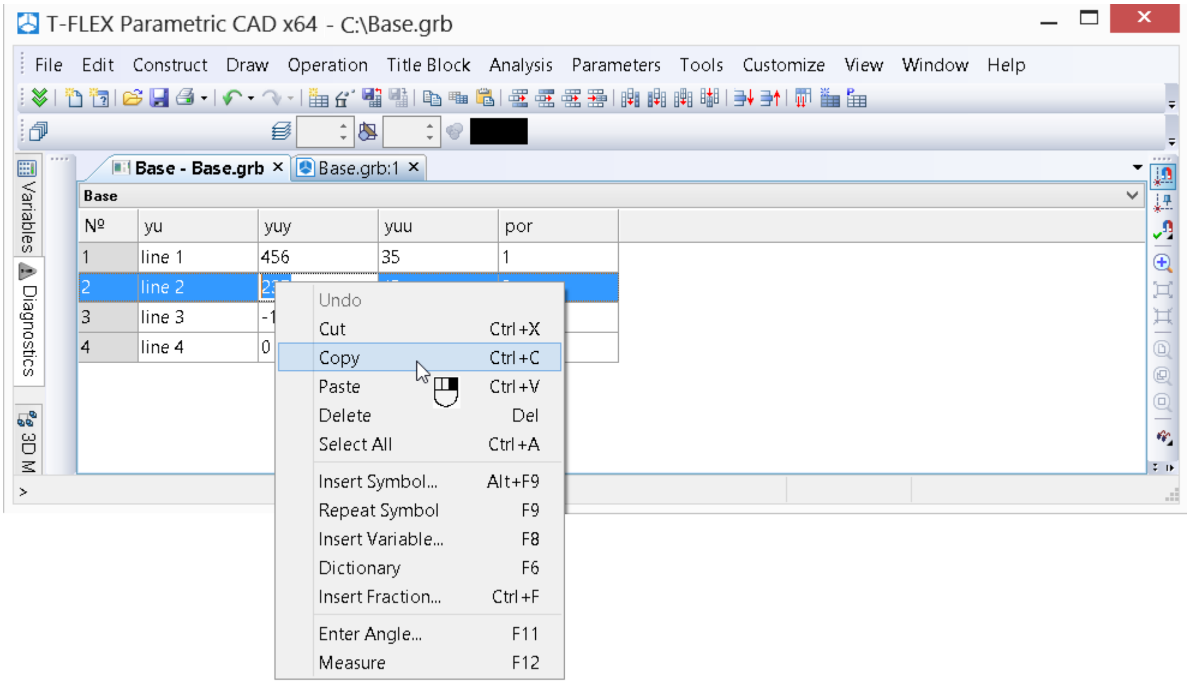
The same actions can be performed by pressing the key combination:

- <Ctrl>+<Down> - *to move the row downwards*
- <Ctrl>+<Up> - *to move the row upwards*

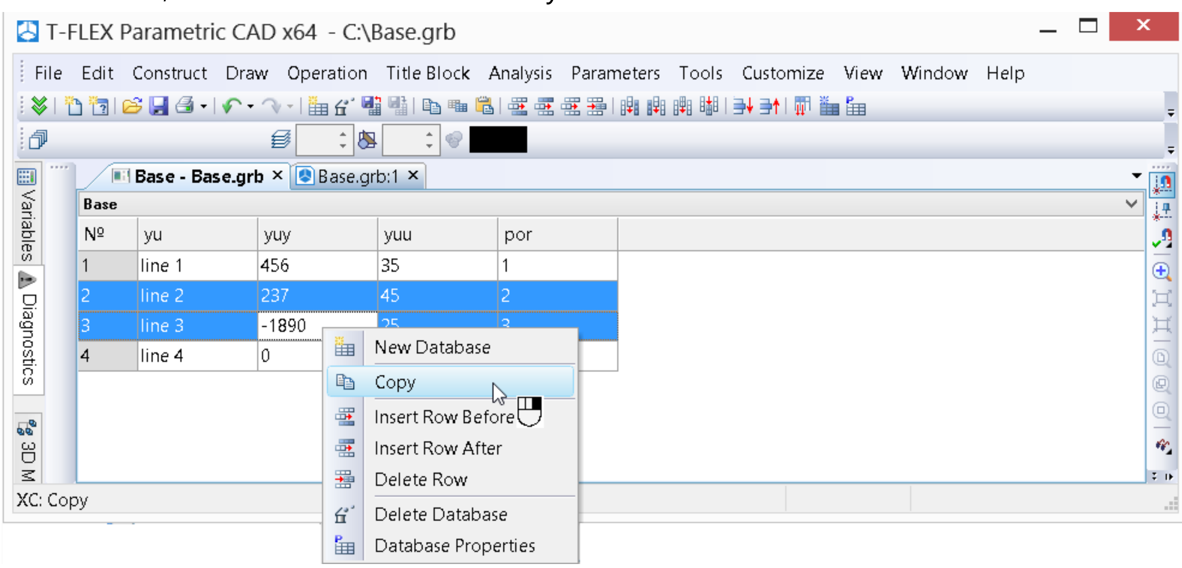
To specify the contents of cells of the data's table, it is required to enter into the mode of editing the required cell with the help of . The selected cell is highlighted with an inverted rectangle (the background color becomes the text's color and vice versa) and the blinking cursor appears in the cell.

To speed up the work of filling in the database table, it is possible to use the options of copying/insertion. It is possible to copy both the contents of individual cells and the entire strings of the database table. It is allowed to copy both inside a single database and from one database to another, even if the second database is inside another document of T-FLEX CAD.

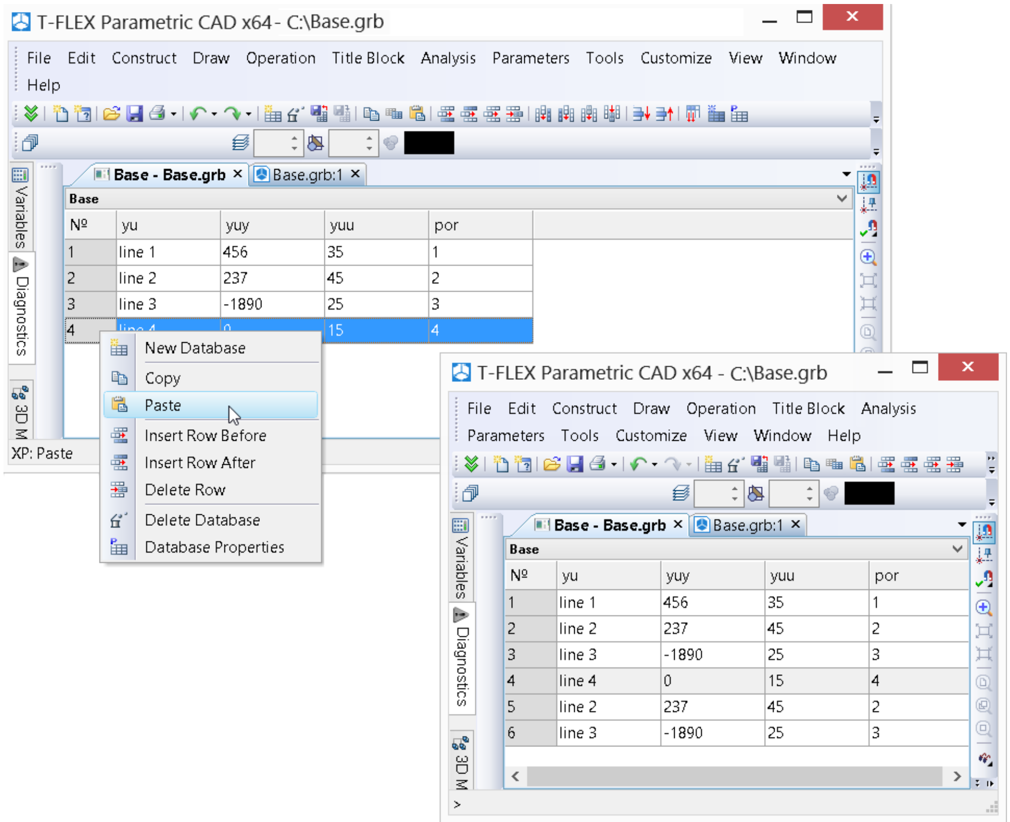
To copy the contents of a specific cell, it is required first to select the contents and choose the required command from the context menu. Insertion of the copied data into another cell is carried out in a similar way. The commands of insertion of symbols, variables, fractions, etc. are also available in the context menu for the cell's contents of the data's table. A detailed description of these commands is given in the **"Texts"** chapter of this user's manual.



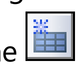
Copying the entire string or several strings simultaneously is carried out in a similar way. To select a string, it is possible to use two options. The first option – indicate the desired string with the help of , which right away also invokes the context menu with the commands of copying/insertion of strings. The second option – select a string with the help of , by indicating the very first, service column of the database table, with the name "Nº". To select several strings, you can use the selection with the help of <Ctrl> and <Shift>, as is done elsewhere in the system.



For insertion of the copied string/strings, it is sufficient to choose the desired place (string) in the table and invoke the "Insert from clipboard" command from the context menu. Insertion will be carried out after the line at which the cursor was at the moment of invoking the insertion command.



To copy/insert the entire strings, it is also possible to use the commands of the main toolbar  and . It is important to note that these commands work only with the entire strings.

The entire contents of the table can be selected for copying with the help of the  command found on the main toolbar (the "Database" group).

For the contents of the cells of the database's table there is a capability of simultaneous sorting by several columns. Such sorting can be used in order to group the data with equal values in one single column, and then carry out sorting of another column in these groups with equal values.

To sort by several columns there is an option on the main toolbar:

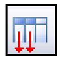
		Sorting

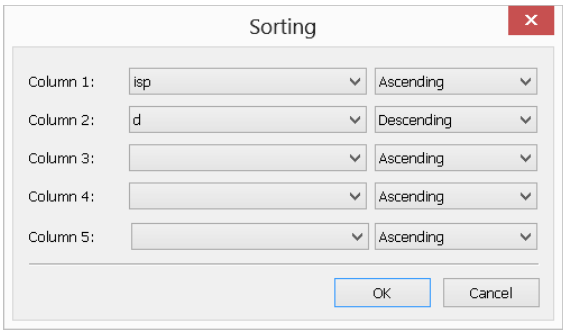
After calling this option the dialog window **Sorting** will appear in which it will be possible to indicate the sorting options for several columns.

For example, suppose there is a database of parameters of flange couplings.

Parameters (Flange couplings)						
№	d	isp	D	l	L	mass
1	11	1	80	30	64	0.68
2	11	2	80	25	54	0.68
3	12	1	80	30	64	0.72
4	12	2	80	25	54	0.72
5	14	1	80	30	64	0.72
6	14	2	80	25	54	0.72
7	16	1	80	40	84	0.78
8	16	2	80	28	60	0.72
9	18	1	80	40	84	0.82
10	18	2	80	28	60	0.76

It is required to group the data that has equal values in the column **isp**, then execute sorting of the column **d** in these groups with equal values.

To do so, on the main toolbar invoke the  option. In the **Sorting** window that appears in the fields of the columns 1 and 2 specify conditions for sorting the values of parameters **isp** and **d**, respectively.

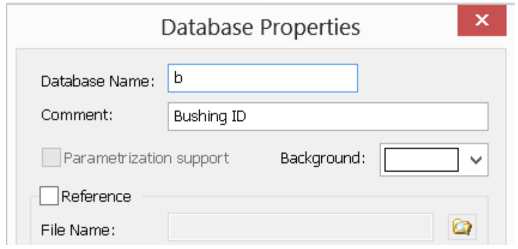


After pressing **[OK]**, the data of the table will be grouped according to the specified conditions.

Parameters (Flange couplings)						
№	d	isp	D	l	L	mass
1	11	1	80	30	64	0.68
2	12	1	80	30	64	0.72
3	14	1	80	30	64	0.72
4	16	1	80	40	84	0.78
5	18	1	80	40	84	0.82
6	11	2	80	25	54	0.68
7	12	2	80	25	54	0.72
8	14	2	80	25	54	0.72
9	16	2	80	28	60	0.72
10	18	2	80	28	60	0.76

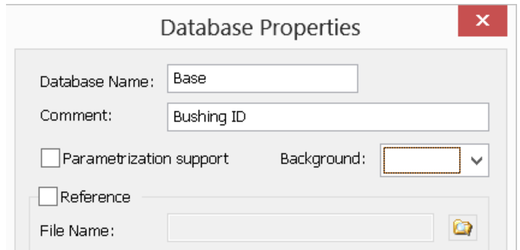
Example of Internal Database Creation

The database creation process will be explained on the example of a bushing. First, let us create parametric drawing of the bushing with the number of variables equal to the number of columns in the table that is shown on the first page of this chapter. After that it is required to create a database and select the desired values from it.



To create a new database, invoke the  option on the main toolbar. In the "Database properties" window that appears, specify the database name.

After pressing [OK] the "Database properties" window will close. The system will automatically transfer to the mode of creation of a database table. The dialog window for specifying parameters of the first column of the database being created will appear on the screen. Specify for it the name "BUSH_ID" and the type "textual".



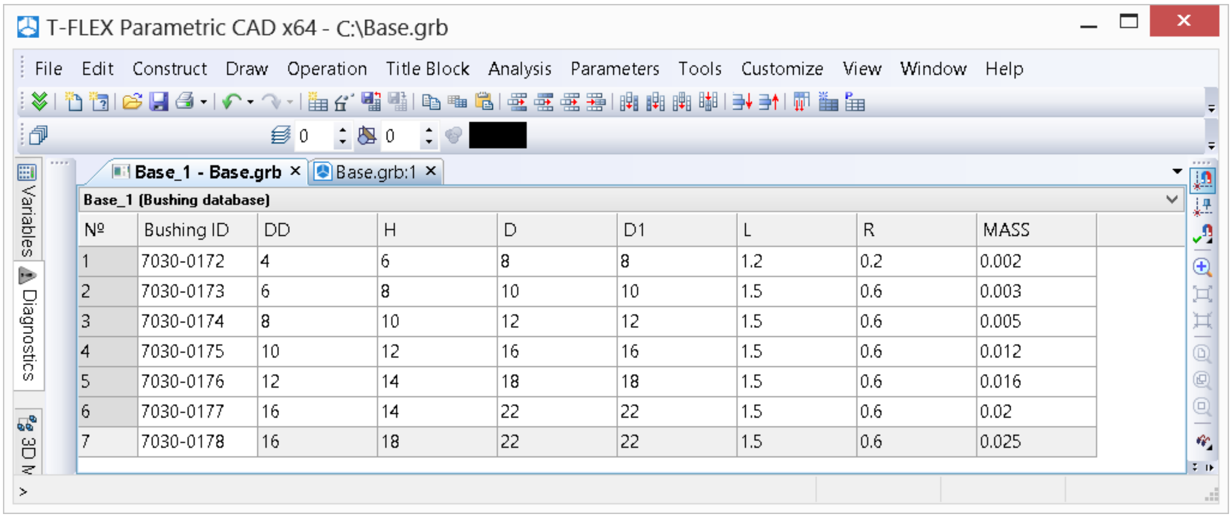
The type of a column is selected based on its expected contents. For example, it is clear that the column «Bushing ID» must contain information of the textual type, the column «length» – integer type, and the column «Mass» – float type.

Since the **Create next column** flag is enabled in the dialog by default, after the input of data and pressing **[OK]** this window will appear again. Specify parameters of the next column in it and so on.

When specifying parameters of the last column, the "Create next column" flag can be disabled. In this case the system will automatically transfer to the mode of filling in the database. If the flag was not disabled, then it is possible just to refuse creation of the new column by pressing **[Cancel]** in the column's parameters window that will appear again.

Now we need to fill in the cells. After filling in the first string according to the GOST data, we need to press the <Enter> **key** – the new empty string will be created and the same actions can be repeated up to the end of the table. If the data coincides for several fields, then it will be more efficient not to retype the data each time from the beginning but instead carry out the operation of copying of rows. Then the contents can be edited.

As a result of your work, the following table should be obtained:



№	Bushing ID	DD	H	D	D1	L	R	MASS
1	7030-0172	4	6	8	8	1.2	0.2	0.002
2	7030-0173	6	8	10	10	1.5	0.6	0.003
3	7030-0174	8	10	12	12	1.5	0.6	0.005
4	7030-0175	10	12	16	16	1.5	0.6	0.012
5	7030-0176	12	14	18	18	1.5	0.6	0.016
6	7030-0177	16	14	22	22	1.5	0.6	0.02
7	7030-0178	16	18	22	22	1.5	0.6	0.025

Thus we created the database inside the drawing which is an analog of the table from the database. Now in the variables' editor it is possible to carry out selection of the required values from this table depending on the controlling parameter (in our case, internal diameter) for recalculation of the model and obtain a drawing of the bushing of the desired size.

Functions for Getting Values from Internal Databases

The syntax of referencing a database field is as follows:

<database name>.<field name>

An entry *BASE.WEIGHT* refers to the field *WEIGHT* in the database *BASE*.

There are four functions for accessing values in the internal databases:

- **REC** - gets the number of the record satisfying the given condition;
- **FREC** – gets the number of the record whose contents matches best the specified value;
- **VAL** - gets the value of the field from the record with the specified number;
- **FIND** - gets the value of the field from the record satisfying the given condition.

Function REC

rec (condition), where

condition is a Boolean expression assuming the values true or false. The expression can contain members that are themselves calls to the fields of the database.

Example:

rec (BASE.DD == 4)

This call means: find the record number in the internal database *BASE*, satisfying the following condition: the value of the field *DD* in this record should be equal to *4*.

Function FREC

frec (`argument_1, argument_2, argument_3, argument_4` *)*, where

argument_1 – is the column in the database that is subject to the search. Must have Real or Integer type;

argument_2 – the sought value;

argument_3 – the search option. Possible options are:

0 – find the nearest value;

-1 – find the nearest floor value;

1 – find the nearest ceiling value.

argument_4 – the parameter indicating the structure of the column subject to the search. It indicates how the values are ordered in the column. The value 0 means the entries are not ordered, forcing the search over all records in the database. The value 1 means the column is ordered ascending or descending. The search completes once the difference between the sought value and the value in the current column of the database is greater than in the previous one.

The parameters argument_3 and argument_4 are optional. If those are not specified, the default values are used:

argument_3 = 0; search for the nearest value;

argument_4 = 0; the column is not ordered;

Function VAL

val (record_number, database_field), where

record_number - is an arbitrary numerical expression yielding an integer.

database_field - the reference to the field.

Example:

val (4, BASE.H)

This call means: get the value from the record number *4* of the field *H* in the database *BASE*.

Function FIND

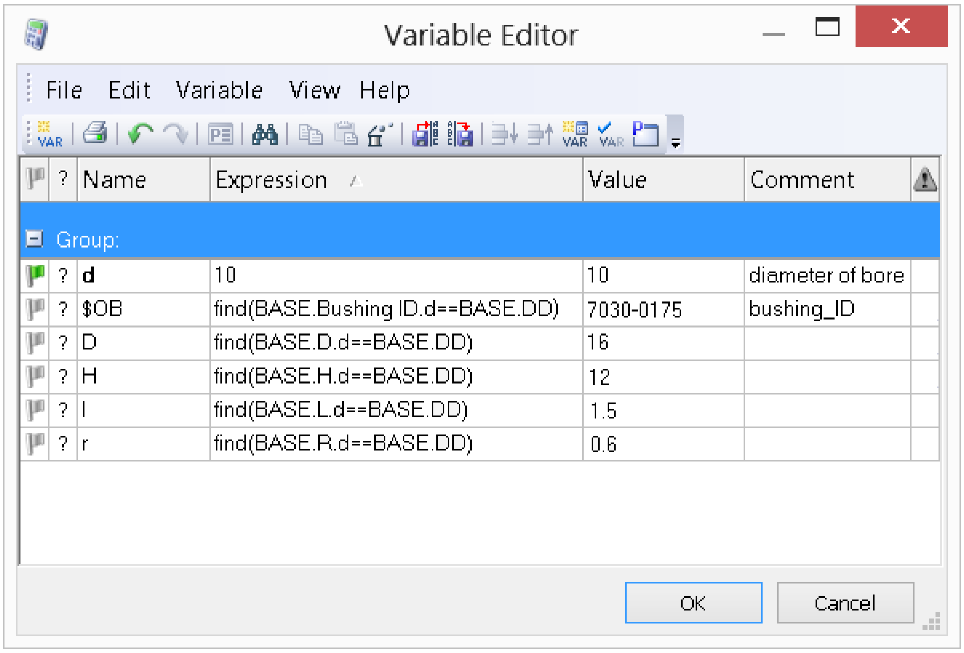
find (database_field, condition_1, condition_2, ...)

This function combines the previous two.

The function returns the value of the specified field *database_field* from the record satisfying the conditions *condition_1*, *condition_2*.

If such record is not found, the function signals the error “Incorrect record number”.To get the parameter values for the drawing of the bushing from the internal DB, use the function FIND (). To do this, enter the variable editor by using the command **V: Edit Variables** and define the expressions for the variables as shown on the right hand side diagram.

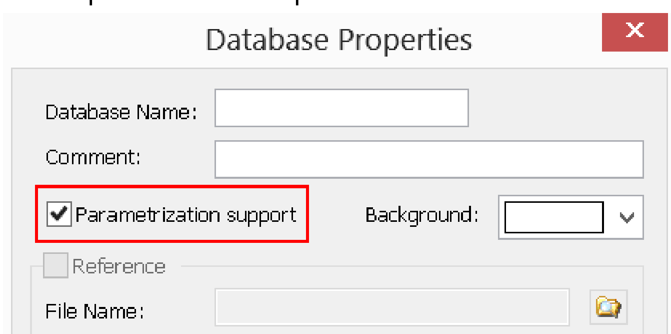
Thus, we have created a parametric drawing of a bushing. The values of the drawing elements will be selected from the internal database *BASE* depending on the value of the inner bushing diameter *d*. In this way, you create a whole family of bushings for latches and mounting rods per GOST 12215-66 by creating a parametric drawing of the bushing and connecting it to the DB.



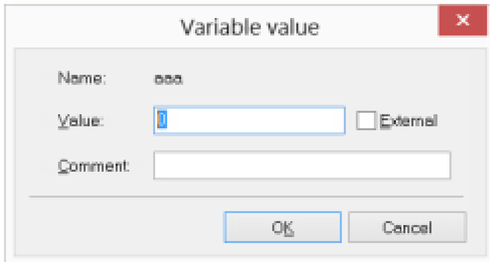
By using this example we have reviewed the main steps of creating databases in T-FLEX CAD. Now, let's review the full set of the database editor commands.

Parameterization of Databases

The values of cells in databases can be specified as variables and expressions with the help of the **Support parameterization** parameter. This parameter can be enabled when creating the database.



b		
№	aa	bb
1	1	10
2	2	20
3	3	30
4	4	40
5	5	50
6	aaa	0



The value of such a cell is recalculated when recalculating parametric model. When being displayed the value of the cell in the table, specified either as a variable or expression, is highlighted with bold font.

b		
№	aa	bb
1	1	10
2	2	20
3	3	30
4	4	40
5	5	50
6	**1000**	0

When clicking the cell the expression appears instead of the value and it can be edited.

b		
№	aa	bb
1	1	10
2	2	20
3	3	30
4	4	40
5	5	50
6	**aaa**	0

Along with the functions for working with databases, the value in a cell can depend on other cells.

In the given example the 6th row contains the sum of all five previous rows:

b

№	aa	bb
1	1	10
2	2	20
3	3	30
4	4	40
5	5	50
6	db_sum(b.aa,1,5)	0

b

№	aa	bb
1	1	10
2	2	20
3	3	30
4	4	40
5	5	50
6	**15**	0

Recursive specification of values is not allowed in this case. In case of appearance of the recursion the cell will be highlighted with red color. In this example this event can occur if the range of rows 1, 5 is removed:

b

№	aa	bb
1	1	10
2	2	20
3	3	30
4	4	40
5	5	50
6	**30**	0

The message about recursion is also sent to the diagnostics window.Cross-like specification of cell's values from one table to another is also possible, but also without recursion.

Functions for Working with Ranges of Cells in Databases

The functions for working with the ranges of cells should be applied when upon working with databases we need the functionality similar to the work functionality of electronic cells, for example, in Excel.

In the variables' editor there are the following functions for working with the ranges of cells of databases:

db_sum – evaluation of sum of the cells in an interval

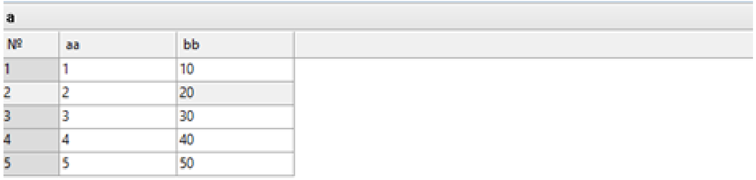
db_mid – evaluation of the average value in an interval

db_max – evaluation of the maximum value in an interval

db_min – evaluation of the minimum value in an interval

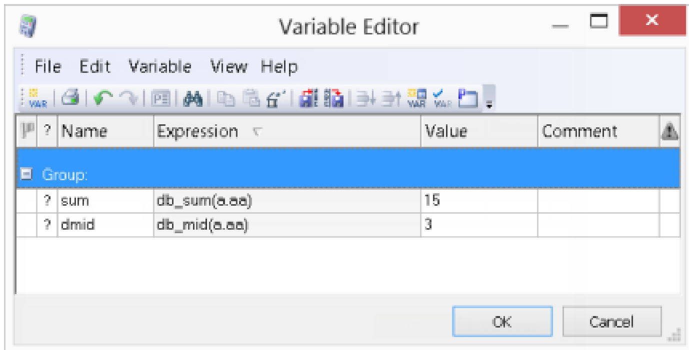
Interval can be specified in the following way:

1. **Specifying the name of a column in DB**. In this example we evaluate the sum and the average value by the column.

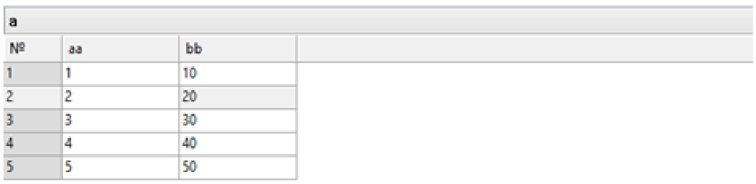


a

№	aa	bb
1	1	10
2	2	20
3	3	30
4	4	40
5	5	50

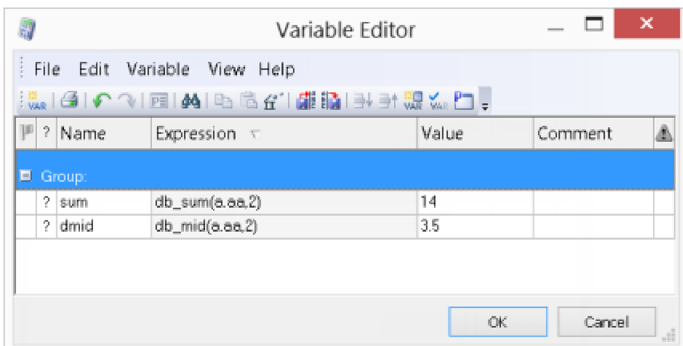


2. **Specifying the column and initial row number**. In this example we evaluate the sum and the average value for the column **aa** starting from the second row.

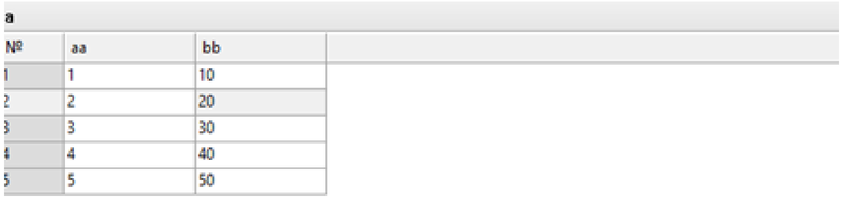


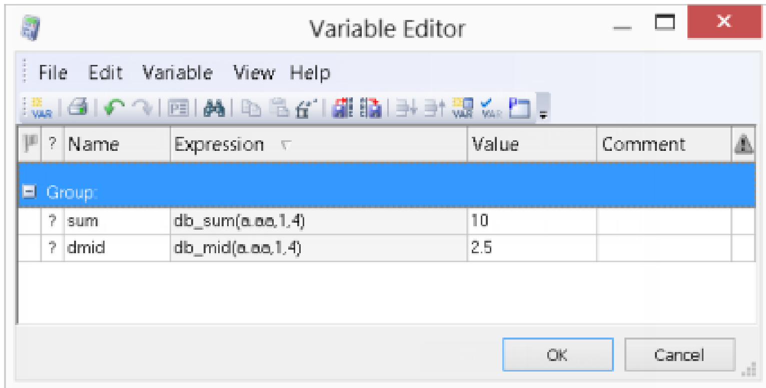
a

№	aa	bb
1	1	10
2	2	20
3	3	30
4	4	40
5	5	50

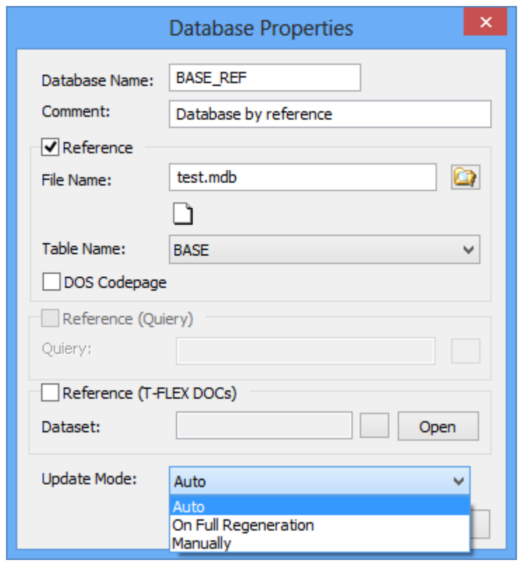


3. **Specifying initial and final rows**. In this example we evaluate the sum and average value from the 1st to 4th rows.





Databases by Reference



As with internal database creation, to create a database by reference you need to invoke the command . In the **Database properties** window that appears it is required to specify the name of the future database, comments (if necessary), data source file and mode of updating the database.

For specifying the source file it is required preliminary to enable the **By reference (file)** flag. After that the **File name** field becomes accessible. To select a file, use the button. The standard window of the file selection dialog will open, in which selection of the required file format and its location is carried out.

To create a database by reference, it is possible to select the files of the following formats:

- ✓ Microsoft Access Files (*.mdb);
- ✓ Files of the format dBase (*.dbf);
- ✓ Microsoft Excel Files (*.xls);
- ✓ FoxPro Files (*.dbf);
- ✓ Paradox Files (*.db);

- ✓ Text Files (*.txt, *.csv).

When creating the database on the basis of the file of ".xls" format***,** it is required to specify additionally that part of the information from the Excel book which will be used as the database's contents. Selection is carried out with the help of the drop down list of the **Table's Name** parameter. By default, the system prompts the user to select one of the pages of the Excel document. In this case, as the contents of the created database, the entire page of the Excel book will be used. The columns' headers will be created on the basis of the first string of the selected page.

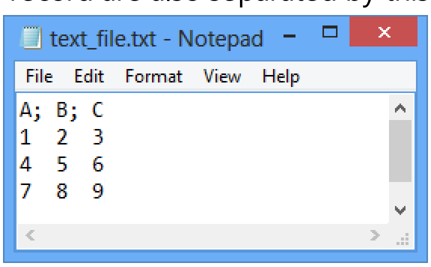
When it is required to use only a part of the page's contents, it is possible to preliminary create the named data domain in the Excel document (select the desired cells in Excel and perform "Insert\Name\Assign"). In this case, when creating the database by reference, this name will also appear in the list of the **Table Name** parameter. The first line of the selected information will also be treated by the system as the string with the columns' headers of the database table.

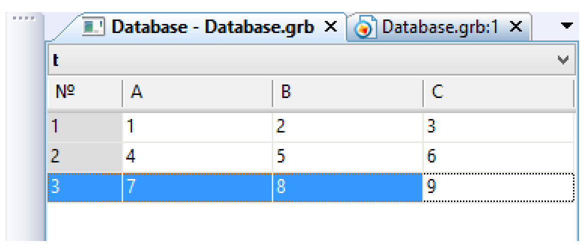
When creating the database on the basis of the file of ".mdb"* format, it is required, in addition to the file's name, specify the table's name from this file. The contents of the selected table will be transferred to the database being created. The list of all tables in the selected file will automatically be placed in the drop down list of the **Table Name** parameter.

When creating the database on the basis of the file of ".dbf" and "*.db" formats*, you need to specify only the file's name (the **Table Name** parameter is not available). In addition, the **DOS Coding** flag is available. When this flag is disabled (default state), the system assumes that the format of the external file is Dbase-Windows. When this flag is enabled, the external file is opened only as a file of the format Dbase-DOS. This allows us to correctly open the databases that contain Russian names for the columns and the contents of the text cells.

When creating the database by reference on the basis of the file of textual format, this file must have the following structure:

- the first line must contain the column's names that are delimited by a symbol. The symbol of delimitation is specified in register in JET parameters (HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\ Microsoft\Jet \x.x\ Engines\ Text \Format - for W7, HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\Jet\x.x\Engines\Text\Format – for Windows XP, HKEY_LOCAL_MACHINE\ SOFTWARE\ Wow6432Node\ Microsoft \Jet \x.x\Engines\Text\Format\Delimited(,) - for Windows 8 the symbol of delimitation is specified in parentheses of Delimited parameter);
- the following lines contain the records of the database. The contents of the fields in each record are also separated by this symbol.





> In the case when the T-FLEX DOCs system is installed on your computer, one more option of creation of the database by reference is available – the database by reference to the T-FLEX DOCs dataset. This is described in more detail in the T-FLEX DOCs user's manual.

The mode of updating the created database can be selected from the drop down list of the **Mode of update** parameter:

Automatically. Updating the data is carried out when opening the file of the model.

During total recalculation. Updating the data is carried out when performing the total recalculation of the model with the update of references.

Manually. Updating the data is carried out only manually by using the special option **Update contents of database**.

By pressing **[Ok]** you complete creation of a database by reference. The "Database properties" dialog is closed, and the table of the new database appears on the screen. It can be viewed but editing the contents is not allowed. The access to the data from a database by reference is carried out similar to the access to internal databases, i.e., by using the name of the database by reference similar to the name of the internal database. In this case the syntax of functions remains the same.

Additional Commands of Database's Editor

Let us review still not mentioned commands of the main toolbar that are designed for working with databases.

The **Delete current database** command allows us to delete the internal database or the database by reference.

The **Save database into file** command allows us to copy the internal database into the external file of the format "*.mdb" or "*.dbf". The database itself does not change, and it remains internal. This option can also be used for the database by reference, in this case the current, internal version of the database will be copied into the external file.

The **Update database's contents** command is used for databases by reference and allows us to compellingly update (from the file) the contents of the database.

The **Table's properties** command allows us to edit the database's attributes. For internal databases you can edit the name and the comments. It is also possible to transform it to the database by reference by enabling the corresponding flag and indicating the name of the external source file. The contents of the database that already exists will be lost in this case. For a database by reference by invoking the command it is possible to change the name, comments and path (including the file's name) and the table's name.

Functions for Working with External Databases

In T-FLEX CAD it is also possible to use the option of working with external databases, when the database is not downloaded directly to the T-FLEX CAD document. In this case to get an access to the database's contents other functions are used.

Functions for Getting Values from External Databases: *DBF () and DBFWIN ()*

The functions DBF*()* and DBFWIN*()*are provided for getting data from external databases in DBF format. The difference between the two functions is in the way of handling the ASCII extension fonts. The user decides what function to use depending on the textual data encoding. In the following description, anything said about the function DBF*()* is also true for the function DBFWIN*()*.

Function syntax:

dbf (arg1, arg2, arg3), where

arg1 - the name of the database. The name of the database can be defined by a string constant, a variable or an expression.

arg2 - the name of the field to get data from. The name of the data field can be defined by a string constant, a variable or an expression.

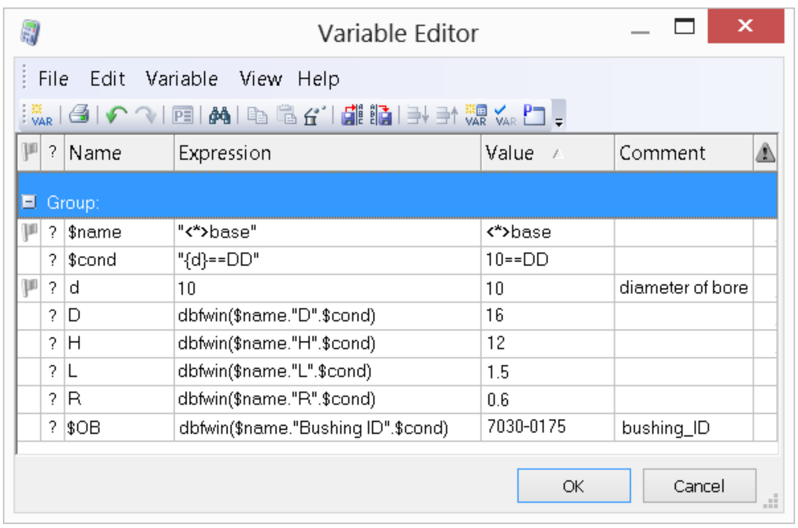
arg3 - the access condition. You can use string constants, variables and expressions for defining the condition.

Example:

dbf("DBF_NAME", "COL1", "COL2 = 30")

The above means: get the value of the field *COL1* in the database *DBF_NAME* under the condition that the value of the field *COL2* is equal to *30*.

If we were to use an external DB for defining the bushing parameters, then we would have to define the following expressions:



Function for Getting Values from External Databases: *MDB()*

The function MDB*()* is provided for getting data from the external databases in XLS (Access) format.

Function syntax:

mdb (arg1, arg2, arg3, arg4), where

arg1 - the name of the database. The name of the database can be defined by a string constant, a variable or an expression.

arg2 - the name of the table in the database. Can be defined by a string constant, a variable or an expression.

arg3 - the name of the field to get data from. The name of the data field can be defined by a string constant, a variable or an expression.

arg4 - the access condition. You can use string constants, variables and expressions for defining the condition.

Example:

mdb ("c:\\T-FLEX_USER.mdb", "USER", "NAME", "Code={code}")

The above means: get the value of the field *NAME* from the table *USER* in the *T-FLEX_USER* database under the condition that the value of the field *Code* is equal to the value of the variable "code" *(*equal to 15 in this case*)*.

It should be noted that the last argument of the function that specifies record selection criterion can be written in the form of SQL query and must correspond to the statement “WHERE” of the “SELECT” command. If text variables are used when writing the condition, then the expression will look like the following: mdb ("c:\\T-FLEX_USER.mdb", "USER", "NAME", "Job title=\"{$Job}\"").

> Because of the slower process of receiving information from the external database the *DBF ()* and *MDB ()* functions should be used only in cases where it is impossible to use the database by reference on the basis of formats*"*.xls"* and *"*.txt"*.

Control Elements. Creating User Defined Dialog Boxes

When working with parametric models and while building an assembly, the user often needs to edit the values of external variables of the models being designed or fragments being a part of an assembly. The T-FLEX CAD general variable editor can be used for handling external variables. However, it is much easier and simpler to work with specially created custom dialogs containing Windows-native tools (input boxes, drop-down lists, "Yes/No" switches, etc.).

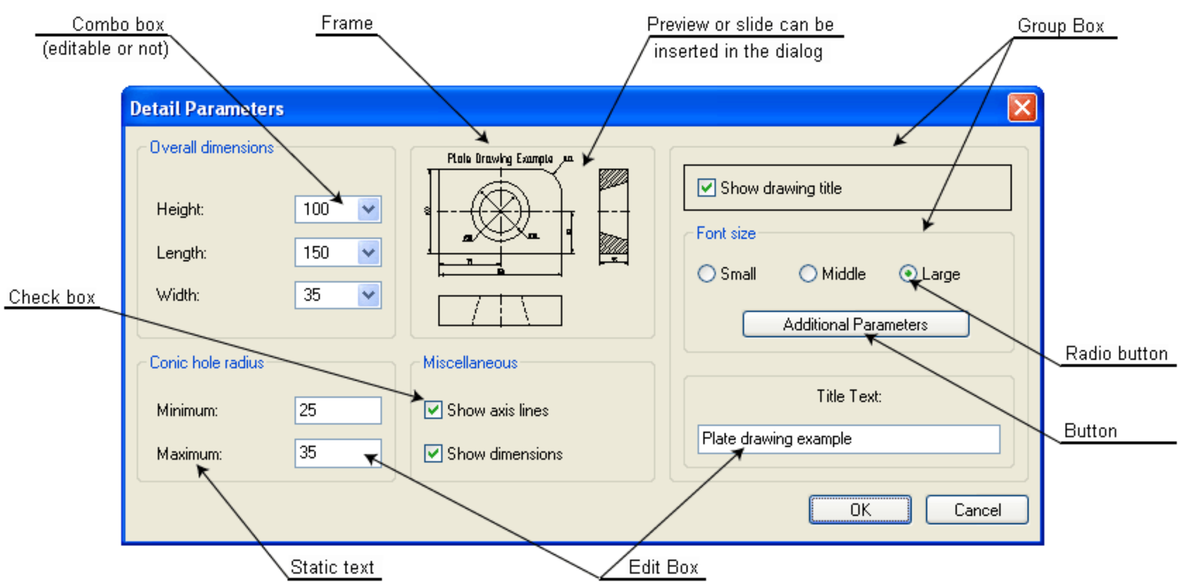
A custom user-defined dialog provides a quite convenient and easy way of editing variables in a parametric model. It is developed by the user when creating a model or fragment. Special T-FLEX CAD system elements, namely, control elements, are used for creating a custom dialog. When creating a dialog, the developer can make it clear and easy to use by adding necessary explanations and comments in the dialog and by structuring the input of the model parameters in a most comprehensible way.

In the future, a custom dialog will be called instead of the general variable editor whenever necessary to edit external variables of a given model or fragment. A custom dialog will be coming up upon calling the command for modifying external variables, **Parameters** > **Model**, in the current document and in the parameters window when inserting the current document as a fragment.

General Information

Types of control elements

T-FLEX CAD system supports the following types of control elements for use in the custom dialogs:



The control elements “Static text”, “Frame” and “Group Box” are not related to any variables and are used for making a clear dialog layout and providing hints.

Static Text is a text string positioned in the specified area of the dialog box.

Frame is a rectangular frame or rectangular area of the specified color. By default, this is a black frame or a rectangle of the background color.

Group Box is a frame with a text aligned with the top border of the box.

The two last elements are used for enhancing and structuring the visual appearance of control elements in the dialog box. Other visual means can also be used for this purpose.

Other elements – “Edit Box”, “Button”, “Combo Box”, “Check Box”, “Radio Button”, “Preview” - are related to variables or the model pages and is intended for manipulating those in certain ways.

Edit Box is a rectangular field for editing the value of the variables associated with this box. Edit box is used for variables without lists of predefined values. It is possible to work in a mode when the displayed value cannot be edited (in this case, in the Edit Box field the value of the connected to this field variable will be shown but this value will not be possible to modify).

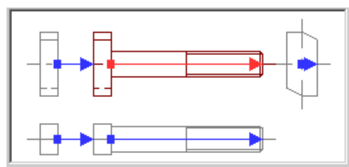
Button. A button defines the sequence of actions performed upon pressing it. Such actions can be:

- Activate Page (show contents of the specified page in the current dialog box);
- Open Dialog Box (displays the dialog box contained on the specified page, in a new window);
- Set Variable Value (assigns the specified value to the selected variable);
- Regenerate 3D Model;
- Run Macro (a macro is a program written in some programming language using T-FLEX CAD API functions. See the “Macros” chapter for details or creating and working with macros).

Combo Box is a rectangular field with a pulldown list button on the right. A combo box is used for modifying a variable value when there is a predefined list of values. The element parameters define whether it is possible to edit the input value.

Check Box switches between the two values of the selected variable. The particular value depends on the check box state.

Radio Button switches between the variable values defined in the radio group parameters. Radio buttons can be conveniently used for variables with several fixed choices of values. (In the case of just two choices, “Check Box” can be used instead.) In this case, a radio group of switches is created for the variable, each representing one of its values.



The custom dialogs allow creation of **Preview** controls. Such an element allows displaying a preview pane for a page per the specified fixing factor in the dialog box for editing the fragment external variables. Preview is not available in the dialogs called by the command **Parameters** > **Model**.

When creating a dialog box, besides the control elements proper, you can use any drawing elements: nodes, construction lines, images, pictures, etc. Construction lines and nodes are not displayed as part of the dialog box window, however, those can be used for precise positioning of control elements when creating complicated dialogs.

Graphic elements and pictures can be used as additional detailing elements in the dialog box along with the standard control elements, such as “Frame”, “Group Box”, “Static Text”. Those elements can be used, for example, for creating a simplified parametric drawing on the page of the dialog box. In this way, the user can evaluate the changes made to the model parameters by this drawing.

Dialog pages

All control elements of a dialog should be located on one page. Dialog elements can be placed on a drawing page or on an additional manually created page. However, we recommend placing control elements on a separate page of “Controls” type, created automatically. Such page is assigned “Custom” format with the paper height and width recommended for dialog boxes, as well as optimum font size and grid step for dialogs.

The dialog box size is defined by the value of the parameter "Paper size" in the command **ST: Set Document Parameters** (the tab **Paper**), defined for the dialog page. The size of the dialog box can be modified via the parameters of the mentioned command or by using the command **Customize** > **Page Size**.

For automatically created “Controls”-type pages, grid display is turned on for easy positioning of control elements. To turn off the grid or to change its step, use the command **Customize** > **Grid**. If necessary, you can turn on snapping to existing 2D nodes or use the absolute coordinates.

The dialog box can be used in the future for editing the model external variables. Therefore, the automatically created dialog page is added for convenience to the list of pages on the tab **External Variables** of the command command **ST: Set Document Parameters** . You need to switch the parameter **External variable editor** to the option **Control**, and check mark the created dialog page in the list of pages.

The page name will be used as the dialog box title. The page name can be modified using the command **Customize** > **Pages…** or with the help of the command **Rename** found in the context menu for the tab of the given page.

Multipage dialogs

A T-FLEX CAD document may contain an arbitrary number of dialog pages. You can build separate dialogs for various groups of parameters and specify different scenarios of the dialogs interaction. The two main scenarios of the dialogs interaction are as follows:

Complex multi-tab dialog box. Separate dialog boxes are joined into a complex one with tabs, each tab corresponding to a particular contributing dialog box;

Main-subordinate dialog scheme. In this scheme, one of the dialogs is rendered main and appears upon calling the command **Parameters** > **Model**, while the rest of the dialog boxes are called, if necessary, via the controls of the main dialog.

The combination of the two schemes is also possible.

When creating a dialog box with tabs, keep in mind that the pages of the dialogs being joined must have same size. Otherwise, all pages and their respective elements will be forced to scaling to the size of the first one.

The second scheme that relies on using several dialog boxes allows different page sizes.

General principles of creating control elements

To create a dialog box use the command **Create Control**. It can be called by one of the following means:

Icon	Ribbon
	Parameters → GUI Control → Control
Keyboard	Textual Menu
<TR>	Draw > Control

Upon calling the command, the following options become available in the automenu:

	<S>	Create Dialog Page
	<F	Frame
	<T>	Static Text
	<E>	Edit Box
	<G>	Group Box
	<B>	Button
	<C>	Check Box
	<R>	Radio Button
	<O>	Combo Box
	<V>	Preview
	<P>	Edit Control Parameters
	<A>	Set absolute coordinates
	<N>	Set relation with Node
	<F4>	Edit Control
	<Esc>	Exit command

Creation of All control elements can be fit in the following scheme:

1. **Defining the size and position of the element being created**. This is done by specifying two diagonal points of the box defining the element extents on the dialog page.

2. **Defining the element parameters** in the "Control Parameters" dialog box. This dialog is launched automatically.
3. **Defining additional parameters** of the variable bound to the control, or of the sequence of actions performed upon activating the control.

Depending on the type of the control, some of the steps in this scheme can be skipped. For example, when creating the "Frame" and "Preview" controls, all you need to specify is the position. Other parameters of these controls are defined by default. To modify those, you can use the control editing command described in the section "Modifying control elements". Additional parameters need to be defined only for the elements "Edit Box", "Combo Box" and "Button".

Parameters of control elements

Control elements, just like any other T-FLEX CAD elements, have a set of parameters. Some of the control element parameters belong to general system parameters: "Level", "Layer", as well as font parameters. The default value of the "Level" parameter is "0", the parameter "Layer" – "Main", the font parameters – "By default".

The general parameter "Priority" is assigned automatically in the process of control element creation, reflecting the order of creation: the first control is assigned priority "0", next – "1", etc.

Besides the general system parameters, control elements have a set of additional parameters:

Caption (the text displayed on the control);

Color of characters (in the cases of "Frame" and "Preview" – the color of the frame strokes or the box background);

Background Color of the control element (for the element "Group Box" – the background color under the group name);

Horizontal Alignment;

Vertical Alignment.

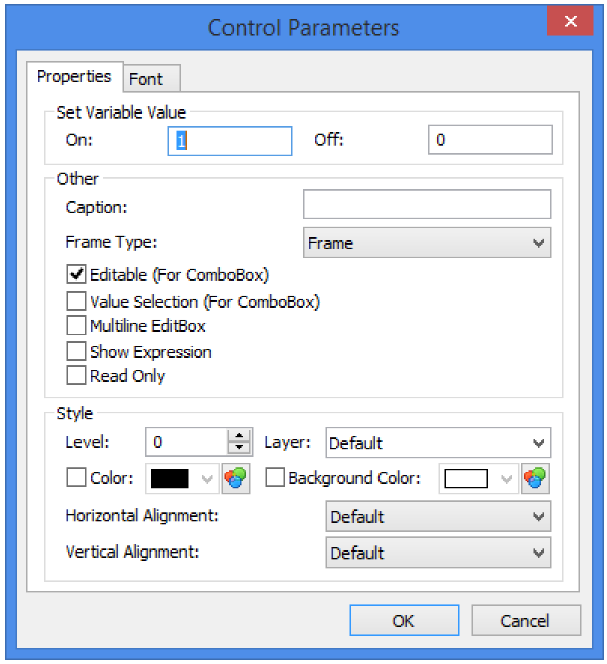
The two latter parameters define the caption text position with respect to the control element boundaries.

Some of the above-mentioned parameters are omitted in certain control elements. For example, the "Button" control omits the parameter "Background Color", while the "Group Box" does not require the parameter "Vertical Alignment", since the group name is always positioned along the top boundary. Note that, when defining parameters of a particular control element, only its relevant parameters will be accessible in the "Control Parameters" dialog box.

Some control elements also have their own specific parameters:

Frame:

Frame Type: frame or rectangle.



Edit Box:

Multiline EditBox – used for text variables only. Permits entering multiple lines.

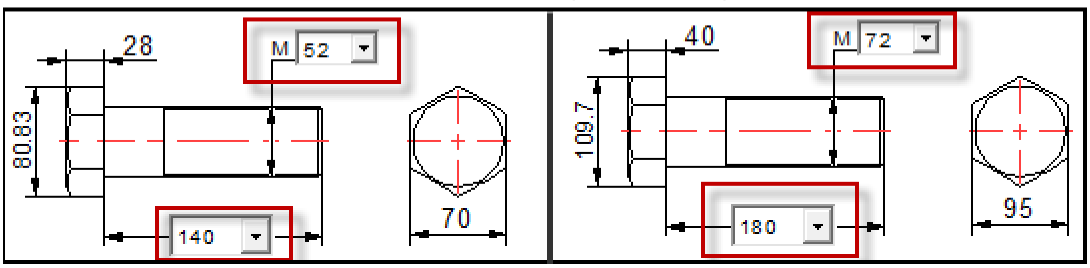
Show Expression. This parameter affects custom dialogs when using variables. Usually, names of the variables are not displayed in the dialog boxes, when such variables or expressions are used as parameters of the fragments being inserted into assemblies. This option outputs name of a variable in edit fields of the dialog boxes. At the same time variable's value or expression is displayed nearby.

Read Only. When this flag is turned on, the editor will work in a regime "read only", i.e., in the Edit Box field the value of the connected to this field variable will be shown but this value will not be possible to modify.

Combo Box:

Editable – allows direct input of variable values without selecting from the list.

Value Selection. Option is used for binding external variables between each other. For example to select the thread pitch from list, according to the thread diameter, automatically. If the option is not activated, the thread pitch will not change according to diameter.



Check Box:

On – defines the value assigned to the variable when the box is checked;

Off – the value assigned to the variable when the box is cleared.

Radio Button:

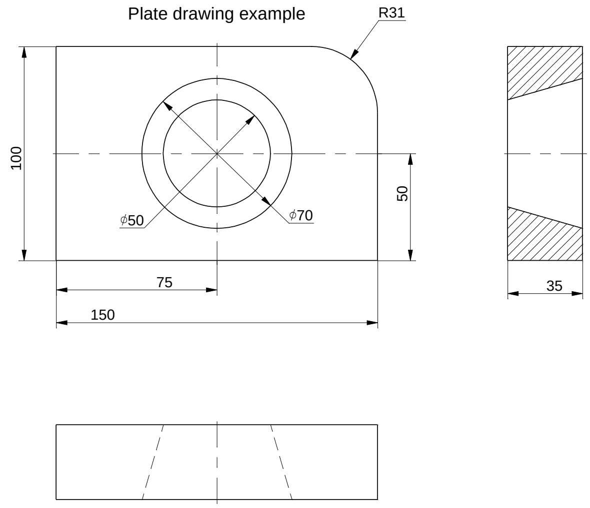
On - defines the value to assign to the variable when switched on.

In the further description of the controls, only the elements will be mentioned that are specific to this control element.

Dialog Box Creation

Let's review the process of creating a dialog box using the example from the chapter "Brief Introductory Course" (section "Creating Parametric Drawing").

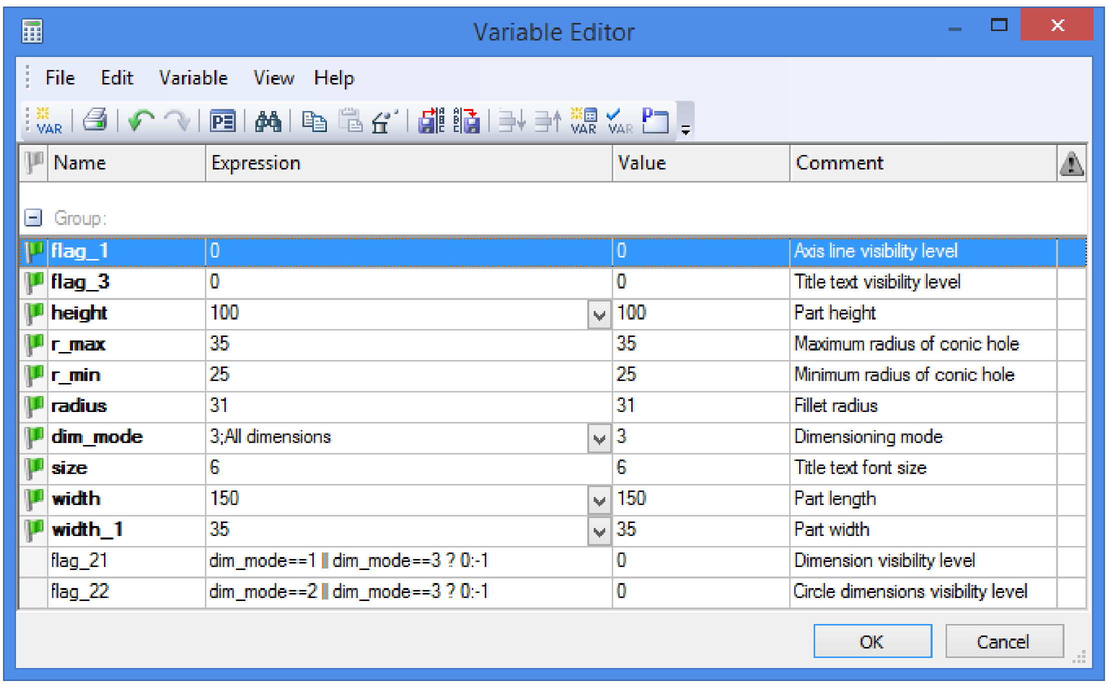
The files for the examples described in the present chapter are located in the library *"Examples"* in the folder *"\T-FLEX Parametric CAD 15 \Libraries\Examples\Parametrization\Control"*. The drawing being considered is stored in the file *"Sample plate drawing.GRB"*.



To modify parameters of the drawing, let's introduce a set of variables:

The variables **height**, **width** and **width_1** are used as parameters of the lines defining the part dimensions;

The variables **r_min** and **r_max** define the radii of the conic-shape hole; **flag_1** defines the visibility level of the axes;

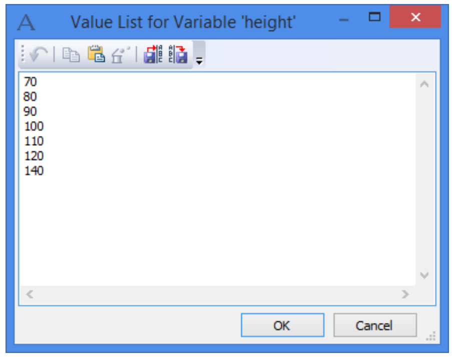


Name	Expression	Value	Comment
flag_1	0	0	Axis line visibility level
flag_3	0	0	Title text visibility level
height	100	100	Part height
r_max	35	35	Maximum radius of conic hole
r_min	25	25	Minimum radius of conic hole
radius	31	31	Fillet radius
dim_mode	3;All dimensions	3	Dimensioning mode
size	6	6	Title text font size
width	150	150	Part length
width_1	35	35	Part width
flag_21	dim_mode==1 || dim_mode==3 ? 0:-1	0	Dimension visibility level
flag_22	dim_mode==2 || dim_mode==3 ? 0:-1	0	Circle dimensions visibility level

flag_3, **size** and **$text** define, respectively, the visibility level, the font size and the drawing title text;

flag_21 and **flag_22** define, respectively, the visibility level of the linear dimensions and the circle-referencing dimensions. Their values are driven by the variable **dim_mode** that controls the type of the displayed dimensions (none, linear only, circle dimensions only or all dimensions).

All variables, except **flag_21** and **flag_22**, are external. Predefined lists of values are created for the variables **height**, **width** and **width_1**, and **dim_mode**. Note that the variable **dim_mode** uses a list of a special type: each numerical symbol is separated by ";" from the string bound to this numerical value. How to use such a list is described below. The introduced variables allow getting different versions of the original drawing: modifying the part size and the size of the hole, displaying axes and various dimensions. You can also modify the font size of the title, or hide it altogether.



Let's consider various choices of the dialogs for this example. Let's begin with creating a simple dialog for this model, as shown on the diagram. The respective library file is "`Sample single dialog.GRB`".

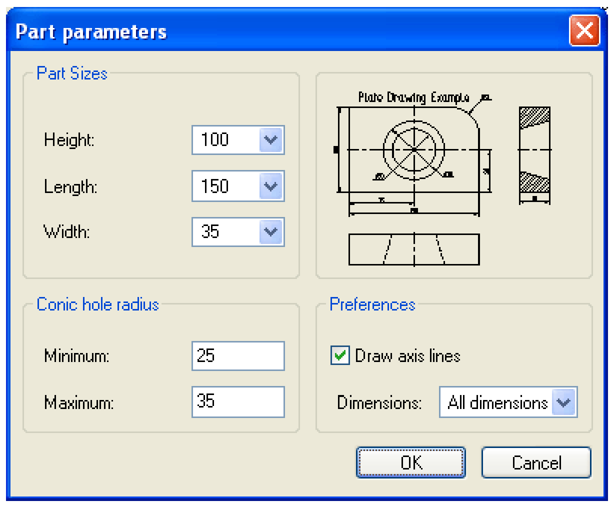
The dialog being created allows modifying the part size and the size of the conic hole and showing/hiding dimension and axes display.

Call the command **TR: Create Control**. Let's create the dialog page:

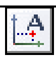
	<S>	Create Dialog Page

By default, the newly created page is assigned the type "Controls", with the recommended sizes set appropriately. This page is also automatically added and check marked in the list of pages displayed by the command **ST: Set Document Parameters** (the tab **External variables**), while the parameter **External variable editor** is switched into the mode **Controls**

Name the new page "Part parameters".



Grid snapping is turned on this page by default. You can also turn on snapping by the absolute coordinates:

	<A>	Set absolute coordinates

When using construction lines and nodes, you can use snapping to 2D nodes:

	<N>	Set relation with Node

If necessary, you can define default parameters for all newly created control elements:

	<P>	Edit Control Parameters

Calling the option <P> brings up the “Control Parameters” dialog box. The latter provides the full set of parameters for a control element. Thus, in our example, the parameter “Editable” can be turned off for “Combo Box”-type controls.

The variables of the model, whose values will be controlled by the dialog being created, can be divided into several groups:

Variables that drive the part sizes (height, length, width);

Variables that drive the size of the hole (minimum and maximum radii);

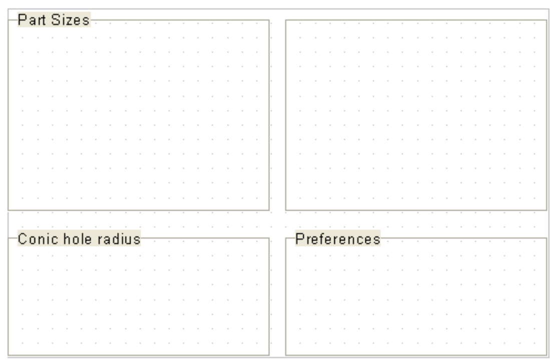
Variables that control axis and dimension display.

Let's divide the dialog box accordingly into several theme areas. In the first area, let's put the control elements for modifying the “Part Sizes”. The second area will hold the controls for defining the “Conic Hole Sizes”. The third area will contain the control elements for axis and dimension display “Preferences”. Finally, the fourth group can unite auxiliary elements: the preview pane (for selecting the active page when inserting the drawing as a fragment) and the picture of the drawing (to display when modifying the model parameters in the command **Parameters > Model**).

The “Group Box” control element will be used for enhanced visual representation/grouping of the control elements in the dialog box being built:

	<G>	Group Box

Create the first group on the dialog page. Set the size and position of the element as shown on the diagram.



In the “Control Parameters” dialog box, enter the group name as: “Part Sizes”.

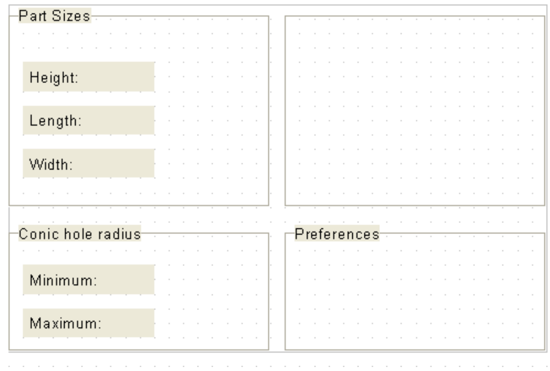
Thereafter, create the rest of the groups in the same way. Position the groups as shown on the diagram. Note that the group reserved for the preview pane is left unnamed. To have it this way, leave the “Caption” entry blank in the “Control Parameters” dialog. The element “Frame” can be used for similar purpose.

Thus, we have created group boxes for various model parameters. Now, we need to create control elements within each group, that will allow modifying the values of the respective variables.

Let's create explanation notes. To do this, select the automenu option:

	<T>	Static Text

To create a string of text, specify two bounding points of the text.



In the coming up “Control Parameters”, enter the string “Height:” in the “Caption” parameter input box.

Similarly, create the captions “Length:”, “Width:” and “Minimum:”, “Maximum:”, placing those as shown on the diagram.

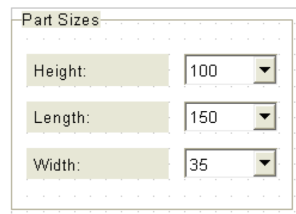
Next, we need to create control elements for editing certain model parameters. The control elements “Edit Box” and “Combo Box” can be used for this purpose.

Since the lists of predefined values exist in this example for the variables height, width and width_1, responsible for the height, length and width of the part, respectively, let's use the “Combo Box”-type controls for these variables.

Since the parameter **Editable** is turned off in the default settings, the direct input of the variable values is disallowed for all created “Combo Box” controls in this example. The variables can only be defined by selecting values from the list.

First, let's create the control element for defining the part height. Call the respective automenu option:

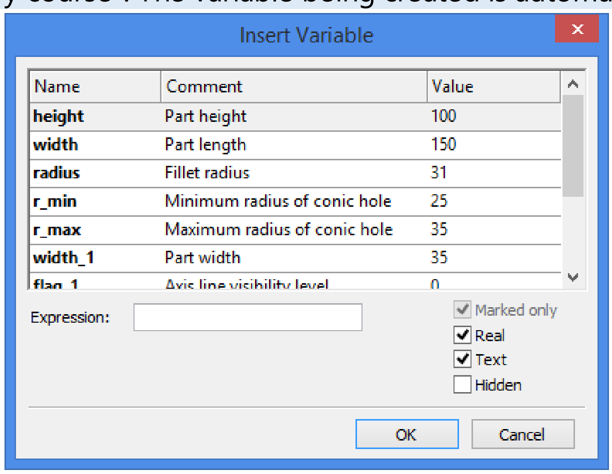
	<O>	Combo Box



Position the control to the right of the “Height” caption, as shown on the diagram.

Next, in the coming up “Insert Variable” dialog box, select the variable to bind to the control being created (the variable “height” in the case of the part height).

> If necessary, you can specify the name of any non-existent variable by manually entering it in the “Expression” field. This brings up the standard “Variable value” dialog box, described in the chapter “Brief introductory course”. The variable being created is automatically marked as external.



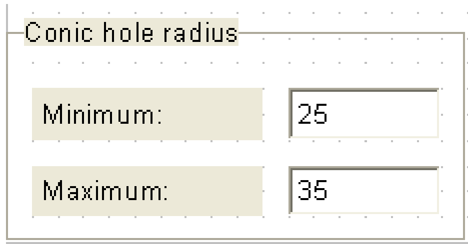
Create yourself the “Combo Box”-type controls for the variables “width” and “width_1”.

The next step is creation of control elements for modifying the variables r_min and r_max that drive the size of the conic hole. Since these variables do not have predefined lists of values, you can use “Edit Box” controls for them:

	<E>	Edit Box

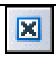
Let's create the “Edit Box” of the variable r_min. Activate the option and define the control position according to the diagram. In the “Insert Variable”, select the variable to bind the control to – r_min.

Similarly, create “Edit Box” for the variable r_max.



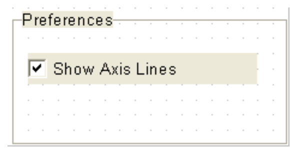
To make the selector for axis display, in this example one can use the “Check Box” control, placing it in the “Preferences” group.

Select the automenu option:

	<C>	Check Box

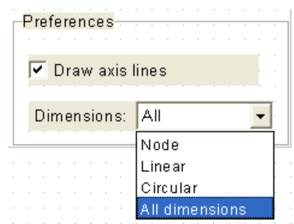
As in the case of any control elements, the first step of creating the switch is selecting two points for defining the switch position. Next, the “Insert Variable” dialog box appears. Select the variable flag_1 from the list. The “Control Parameters” dialog box then comes up automatically.

In it, you need to define the respective variable values for **On** and **Off** states. Since the variable flag_1 in our example defines the visibility level of the axes, the “On” state corresponds with “0”, while “Off” – with “-1”. In the **Caption** input box enter the string “Show Axis Lines”. We will use another “Combo Box” control for managing the dimension display options, bound to the variable **dim_mode**.



Place this control in the “Preferences” group box. Note that, with the flag “Editable” turned off, the described combo box displays in its value field and the pulldown list the textual string portions defined in the values list of the variable **dim_mode**, rather than the numerical values. Put the explanation text “Dimensions:” before the “Combo Box” control, using the “Static Text” element.

The remaining empty space on the dialog page can be used for placing a picture or preview pane.



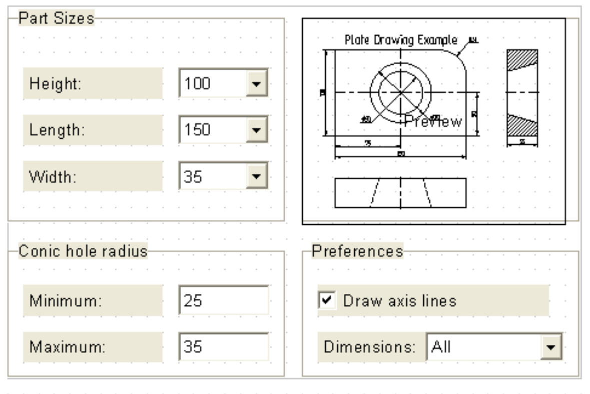
Let's insert a picture in the dialog, illustrating our model. Save your model as a picture, using the command **EX: Export**. After that, add the created image to the dialog page, using the command **IP: Insert Picture**, and place it in the empty group box. Use **File** > **Assembly** > **Links** command for breaking link between a picture and the corresponding file. Set “Embedded” flag for the respective item.

The “Preview” element can be placed on top of the picture. To create it, select the option:

	<V>	Preview

Specify two points defining the size and position of the preview pane.

In this example, align the preview pane boundaries with those of the group box hosting the picture. The “Preview” element should fully cover the picture.



From now on, when inserting this drawing as a fragment, there will be a preview pane in the dialog box.

In this way, both the picture and its hosting group will be screened by the preview. However, when calling the dialog by the command "Parameters| Model", the preview pane will not appear, exposing the underlying picture.

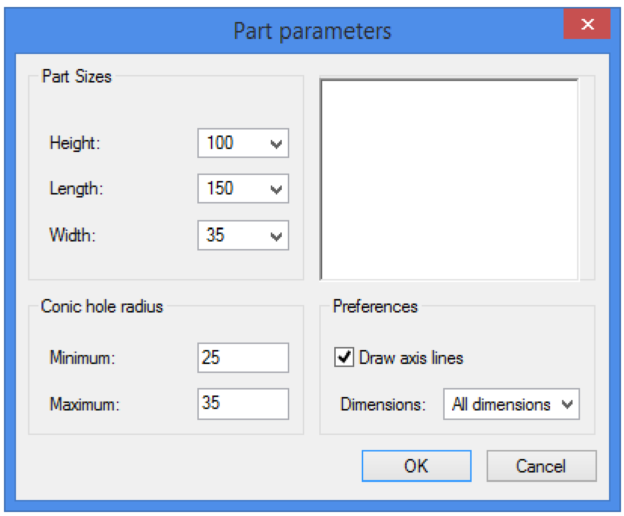
Now, all dialog controls are created. To insure that the transition from one to another control in the dialog follows the desired order (as, for instance, when using the <Tab> key), we need to define the activation sequence of the control elements while working with the dialog. This can be done using the control element editing command **EO: Edit Control** described in the section “Modifying Control Elements”.

Thus, the dialog creation is complete. In the course of creation, you learned how to create a dialog page and control elements for decorating the dialog box and modifying the values of the external variables.

USE OF THE DIALOG

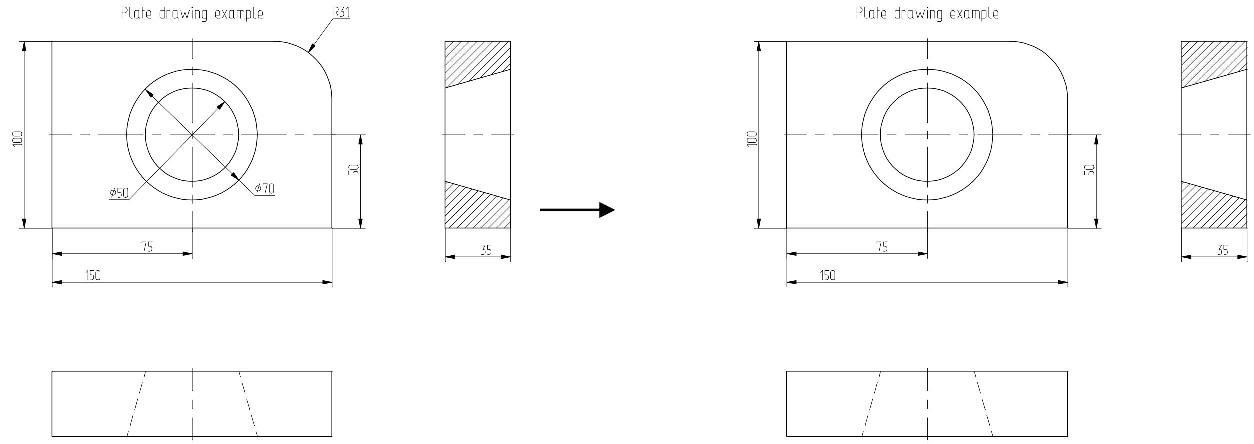
As was mentioned above, the created dialog box can be used for defining the model parameters and when using the model as a 2D or 3D fragment.

In the first case, the command **Parameters** > **Model** is used for launching the dialog box. Upon calling the command, the newly created dialog appears on the screen, allowing to modify the values of the model variables. Modifying a parameter in the dialog box instantly reflects on the part drawing. However, the image on the picture does not change since it is not parametric to our model.

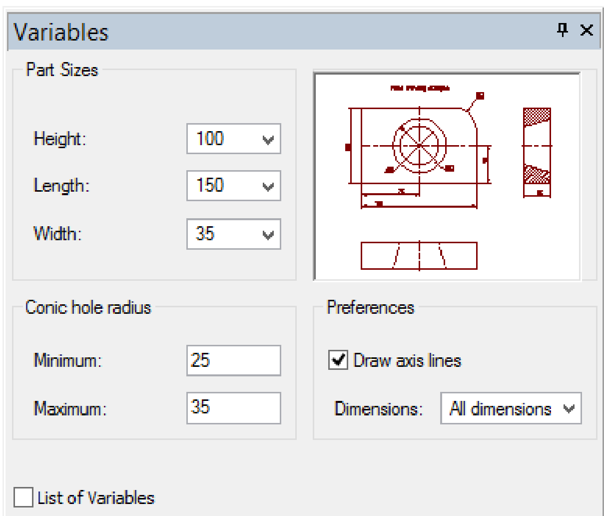


Try changing the dimension display mode: set the parameter “Dimensions:” to the value “Linear”. The drawing will adjust to this setting as shown on the diagram.

To finish working with the dialog box, you need to press *[Ok]*.



When inserting the model as a fragment, the created dialog box becomes part of the fragment parameters dialog box. To verify that, save the resulting model using the command **File** > **Save**. Then open a new document using the command **File** > **New 2D Model**. Call the fragment creation command **Draw** > **Fragment** and select the automenu option . In the coming up window find the file of your example. The fragment parameters dialog box will then display the created dialog. By default, the user-defined dialog of fragment parameters will be a part of the general dialog of parameters of fragment insertion command. By pressing  at the upper right corner of the parameters dialog, the dialog can be shown in a mode of viewing in a separate window.

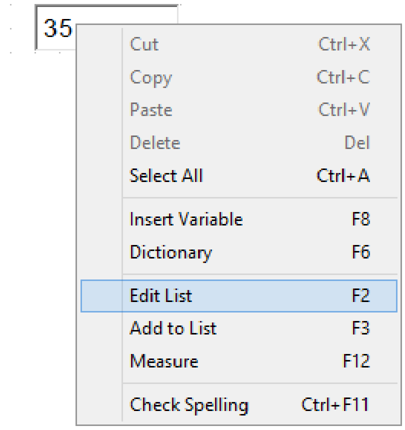


Note that the preview pane now appears in the dialog box, screening out the picture. All changes done to the model parameters using the dialog controls will be reflected in the preview pane. The preview pane also displays the selected drawing page and the fragment fixing vectors present in the model.

The control elements that make up the dialog can be used for modifying the model external variables directly in the 2D window, without calling the command **Parameters** > **Model** or **Draw** > **Fragment**. In this way, working with the dialog page is similar to working with the actual dialog box when the latter comes up on the screen.

Thus, upon clicking on the "Button" control, all functions defined for this control elements are activated from the dialog page. Upon clicking on the elements "Edit Box" and "Combo Box", one can manually define or select from the list a new variable value. Clicking changes the states of the "Radio Button" and "Check Box" control elements.

To finish inputting changes, you can press , <Enter> or over any empty area on the dialog page. The entered changes can be canceled by pressing , <Esc> or . In this case, a confirmation window will appear, double-checking about the cancellation.



A "Combo Box" control allows an additional manipulation. If it is bound to a variable whose predefined list is based on a file, then you can modify this list directly in the dialog page by turning on the "Editable" parameter. To do this, point the mouse to the input box of this control and access the context menu.

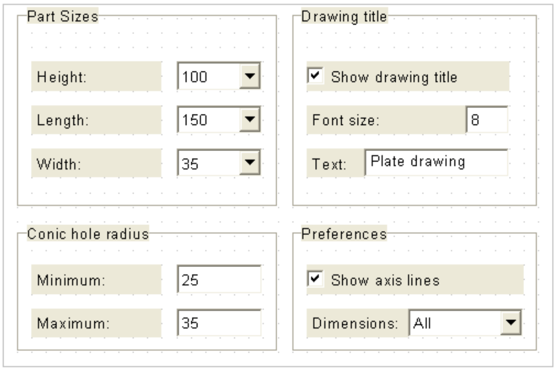
Commands will be provided in the context menu for modifying the list of predefined values of the variable. Upon selecting the command **Add to list**, the value entered in the input box is added to the list of values for this variable. Selecting the command **Edit List** brings up the "Value List of Variable" dialog box, that allows editing the existing list.

PARAMETRICITY OF CUSTOM DIALOGS

In the course of reviewing this example, it was mentioned that control elements, just like other T-FLEX CAD elements, possess general system parameters "Level", "Layer", etc. This allows creating parametric dialogs, whose appearance changes depending on the kind of a model.

As an illustration to this statement, let's continue with the current example. Let's create a modification of the dialog. The respective file in the library of samples is "`Sample parametric dialog.GRB`".

Delete the elements "Preview" and "Picture" from the dialog page. Then select the group box of the deleted elements and double-click . In the coming up "Control Parameters" dialog box, modify the group box title by assigning the parameters "Caption" the string "Drawing Title".



Let’s place in this group box the control elements for handling the drawing title. The variable “flag_3”, “size” and $text will define, respectively, the visibility, size and text string of the title. To control the title visibility, the variable flag_3 needs just two values (“0” when the title is displayed, “-1” – title hidden). Therefore, let's create a “Check Box” control element for this variable.

Position the control as shown on the diagram. In the window “Insert Variable”, select the variable flag_3. In the “Control Parameters” dialog box, assign the “Caption” parameter the string “Display Title”. Enter the value “0” for the parameter “On”, “-1” - for “Off”.

Next, let's create control elements for defining the title font size. Since the variable "size" does not have a list of values, one can use an “Edit Box” control. In the window “Insert Variable”, select the variable "size". Put the explanation text “Font size:” before the control, using the “Static Text” element.

The next step is creation of control elements for modifying the title text.

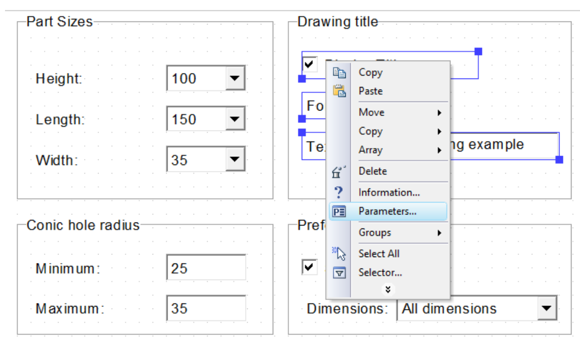
The variable $text can be conveniently controlled by an “Edit Box” element as well. To provide support for defining a multiline title, turn on the flag “Multiline EditBox”, using the option . Then specify the element position and select the variable $text in the “Insert Variable” window.

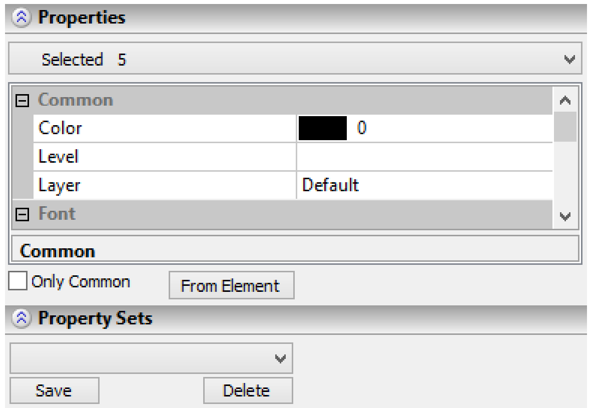
> To go to the new line in the multiline editor, use the key combination <Shift><Enter>. The control sequence “\n” can be used for separating lines as well.

Put the explanation text “Text:” before the “Edit Box” control, using, as before, the “Static Text” element.

Thus, we have created the interface for controlling the title text in our dialog. However, it is not yet parametric. Let's modify it in such a way that the control elements for modifying the font size and the title text string were displayed in the dialog only when the flag “Display Title” is turned on. To get this, let's relate the “Level” parameter of the relevant elements to the variable flag_3.

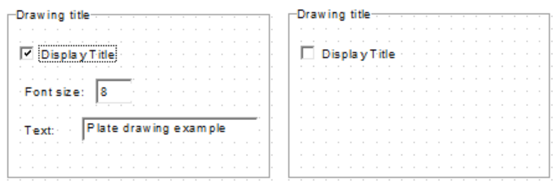
Select by box the font size and the title string control elements and click .





In the context menu, select the item **Parameters...**. The dialog for editing properties of the selected elements will appear in the properties window. In this dialog, clear the marks against all parameters except the "Level" parameter, and assign the latter a new value: flag_3 == 0?0:-1. Click . The entered expression means: if the variable flag_3 is equal to 0, then the parameter "Level" will be assigned the value 0, otherwise - (-1).

Thus, the dialog is created, and we can call it by the command **Parameters** > **Model.** Calling the command brings up this dialog box on the screen.



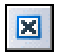
When the flag "Display Title" is turned on, the parameters "Font size:" and "Text:" are present in the dialog. If the "Display Title" flag is off, the controls for defining the font size and the title text string are omitted in the dialog. Therefore, the created dialog box sample has parametric properties. One can create various parametric dialogs of more complex structure, using the same principle.

Working with Multiple Dialogs

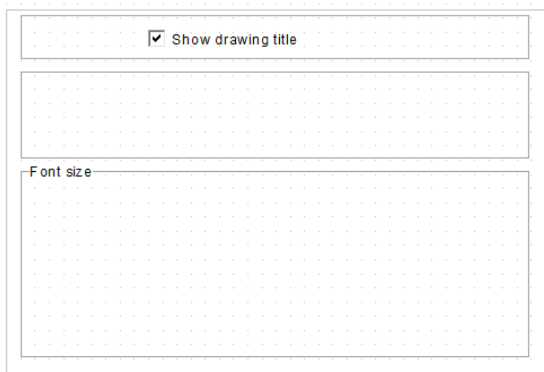
As was mentioned above, multiple dialogs can be created within one model. Let's get back to the dialog instance with a preview pane and a picture. In addition to the existing dialog, let's create a dialog with controls for the title of the drawing. The file of this instance is *"Sample dialog with tabs.GRB"*.

Call again the command **TR: Create Control**. Using the option , create a page for the second dialog. Rename it with the help of the command **Pages**, setting the name "Drawing title".

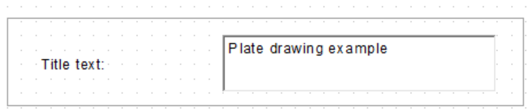
As it was mentioned, the variables "flag_3", "size" and $text control the visibility, size and the contents of the drawing title text string, respectively. For each variable's control element, let's create the three respective "Group Box" elements, as shown on the diagram.

As before, create a "Check Box"  for switching the values of the variable flag_3.

Place this control inside the first group, as shown on the diagram. In the window "Insert Variable", select the variable flag_3. In the "Control Parameters" dialog box, assign the parameter "Caption" the string value "Display Drawing Title". Set the parameter "On" to "0", "Off" – "-1".



We will use the "Edit Box" control for modifying the title text, with the flag "Multiline EditBox" turned on.



We need to put the explanation text "Title Text:" before the "Edit Box". To do this, use a "Static Text" element.

Finally, let's create the control elements for defining the title font size. Those will be placed in the group box "Font size".

We will pre-define three choices: small font (size=6), medium (size=8) and large (size=10). For controlling the value of the variable "size", we will use three "Radio Button" elements: one per each value of the "size" variable.

Call the option:

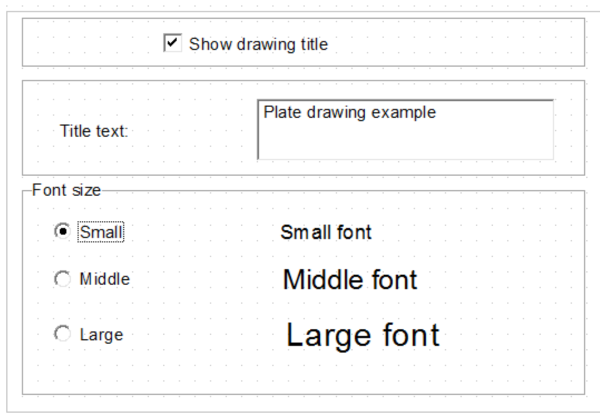
	<R>	Radio Button

Specify the position and the size of the switch. In the "Insert Variable" window, select the variable "size". In the parameters dialog box set the "On" parameter value equal to "6" and the parameter "Caption" as "Small". Similarly, create another two "Radio Button" controls, positioning those exactly beneath the first one.

Their related variable should also be 'size". The "On" parameter settings for these elements should be "8" and "10", respectively.

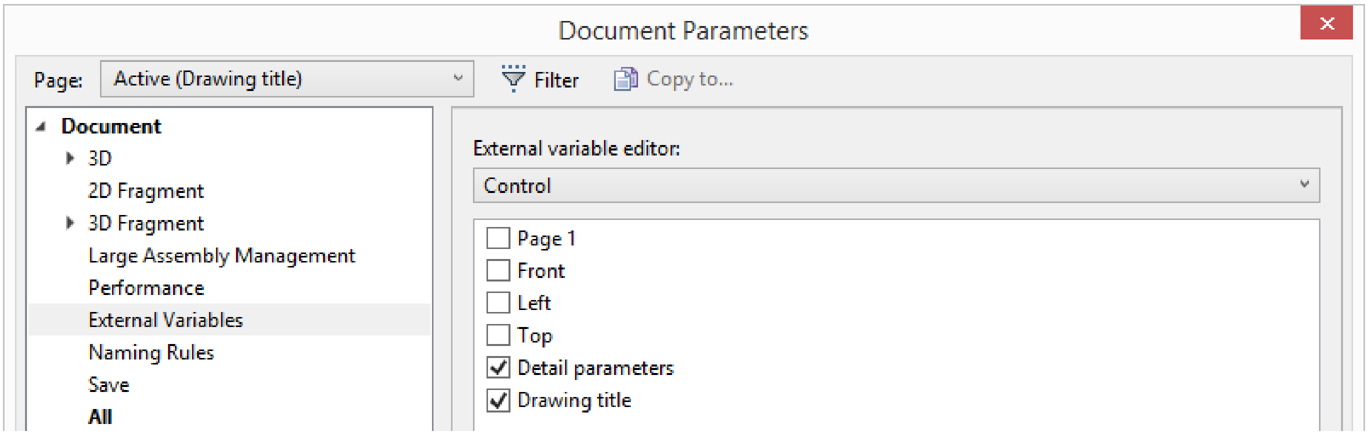
The name of the second switch is – "Medium", the name of the third one – "Large".

One can put "Static Text"-type control elements next to the switches to provide explanation, mentioning the respective font size of the text (6,8 and 10) in the "Control Parameters" dialog box on the tab "Font". Once all the control elements are created, make sure to check their activation order.

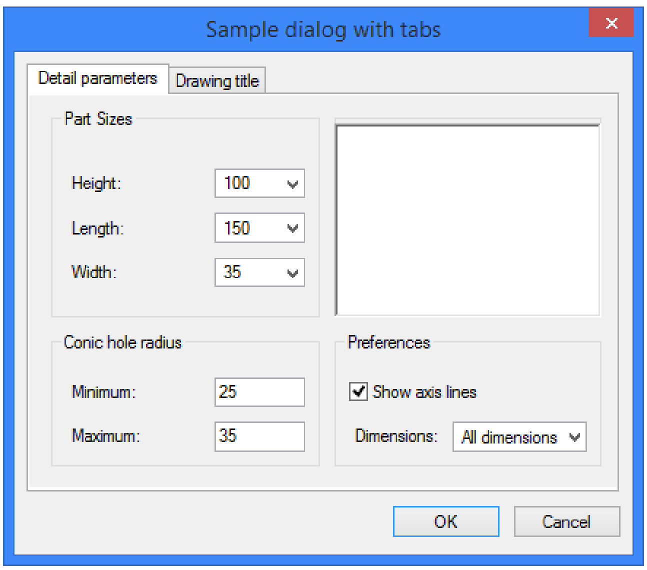


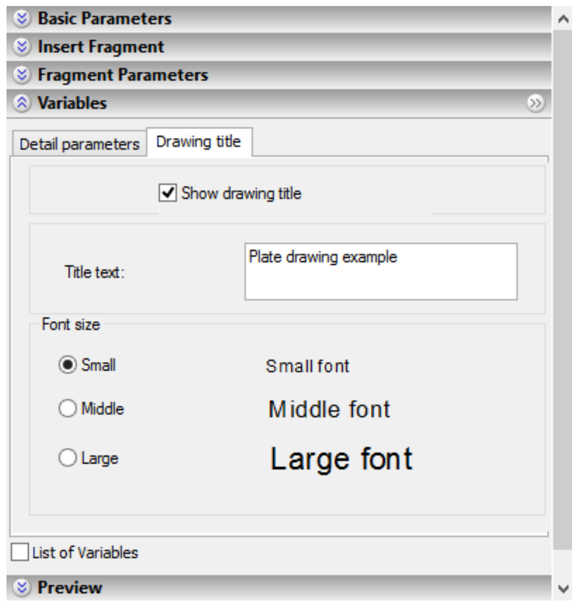
Thus, a second dialog is created in the model – the one for controlling the title of the drawing.

Next, we need to decide, how the two dialogs will interact. Since the option  was used for their creation, both pages were automatically added and check marked in the page list of the command **ST: Set Document Parameters** (the tab **External variables**). Meanwhile, the parameter **External variable editor** switched to the mode **Control**.



In this case, the dialog pages will be automatically joined into one dialog box with tabs. The tab names will be the same as the names of the pages. The first tab will correspond to the first page, the second tab – to the second page, respectively.

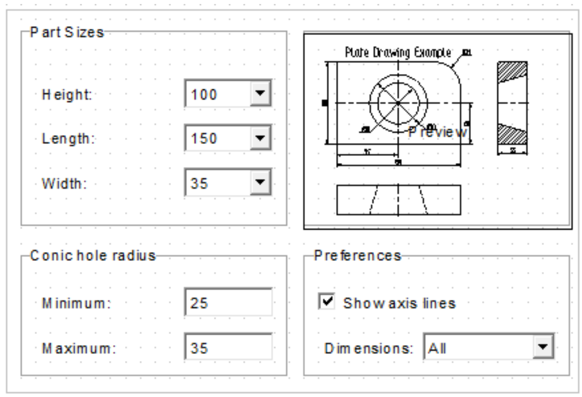




However, there is another way of using multiple dialogs.

Consider one dialog as main. Then this dialog will be launched by the command **Parameters > Model** or when inserting the fragment. Other dialogs can be called, if necessary, using the "Button"-type control elements of the main dialog.

Let's consider this possibility in our example. Select the “Part parameters” dialog box as main. To do this, in the command **ST: Set Document Parameters** (the tab **External variables**) clear the checkmark on the page “Drawing title”. Now, upon calling the command **Parameters > Model** or while inserting the fragment, only the dialog “Part parameters” will be displayed.



The second dialog will be called by one of the controls of the “Part parameters” dialog box. Let's create a button as such a control. Place this button in the “Preferences” group box.

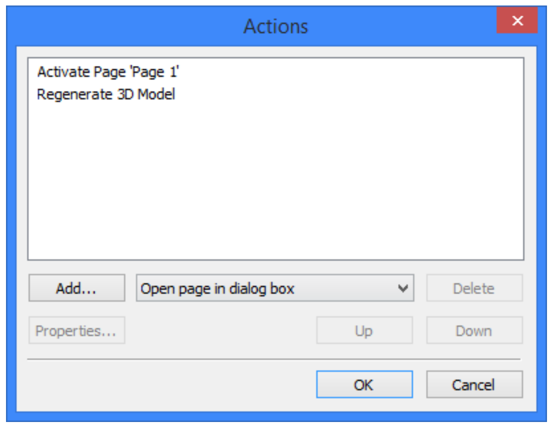
Note that the dialog page is already fully occupied by the existing control elements. To fit the button, let's reduce the size of the “Check Box” element for the variable flag_1. Change the “Caption” property of this element from “Show Axis Lines” to “Show Axes”.

To create the button, again call the command **TR: Create Control**.

Select the automenu option:

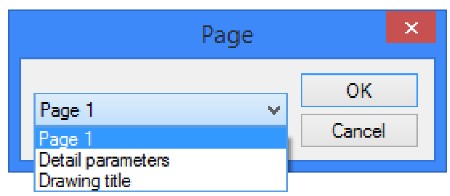
	<B>	Button

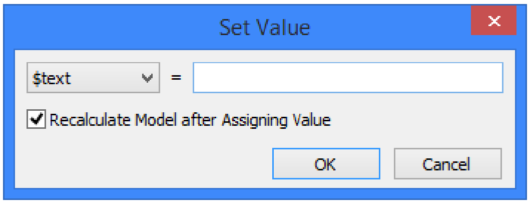
As was mentioned, this option allows creating a button control on a dialog page and define a sequence of actions performed upon pressing the button.



The first step of button creation is defining its size and position in the dialog box. Next, in the “Control Parameters” dialog box, define the button parameters. Then, the “Actions” window appears. Here we need to specify the action (or a sequence of actions) to be performed upon pressing this button.

Adding actions to the list is done as follows. In the pulldown list of actions select the desired one and press [Add...] button. Then, depending on the selected choice, an appropriate dialog box will appear for selecting a page or variable and the value to be assigned to it. After that, the action is added to the list of actions of the button being created.





To make changes in the specified list of actions in the “Actions” window, use the following graphic buttons:

To delete an action - select the desired line in the list of actions and press the button [Delete];

To modify an action - select the desired line in the list of actions and press the button *[Properties...]*. A dialog box will appear for selecting the page or variable and for specifying its value;

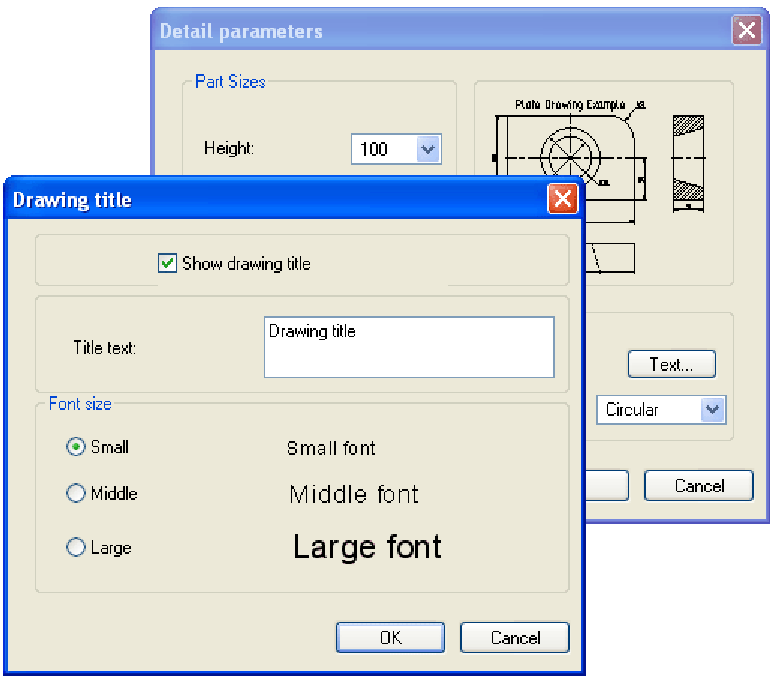
To change the order of actions (which is the same as their order in the list) - select the necessary line in the list of actions and move it, using the buttons *[Up]* or *[Down]*.

> When pressing the button, the actions of performed in the order in which they were defined. Therefore, the order of actions is defined according to the purpose of the button creation. In addition, keep in mind that if the action sequence involves calling a dialog box, the remaining actions will be performed only after closing this dialog box by pressing the [OK] button. If the dialog call ended in pressing the button [Cancel], the remaining actions are ignored.

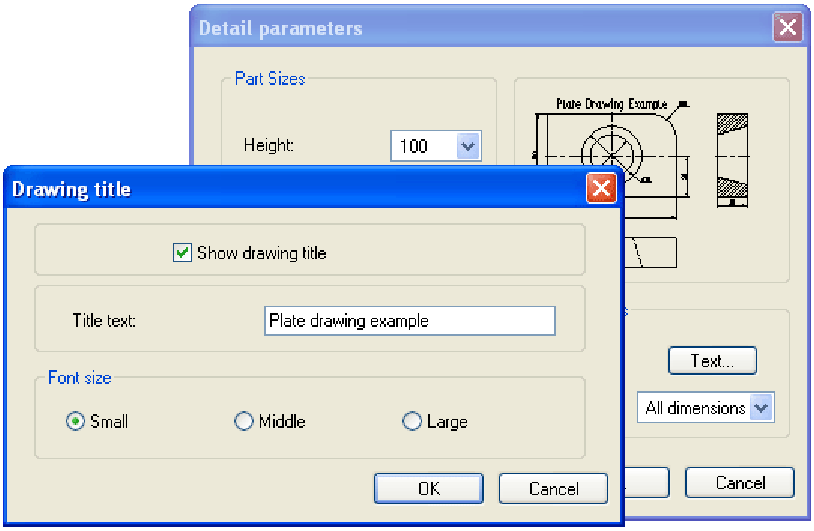
For the current example, define the position of the button on the dialog page as shown on the diagram. In the “Actions” window, define the following sequence of actions:

1. Assign the variable: $text=“Drawing Title”;
2. Show the “Drawing title” dialog.

After creating the button, call the command **Parameters** > **Model**. The “Part parameters” dialog box will appear. Press the graphic button *[Text...]*. The “Drawing title” dialog window opens. The title text changes to “Drawing Title”. To return to the main dialog, just press the *[OK]* button.



In this way of dialog interaction, the dialog box sizes can be different. Let's reduce the "Drawing title" page size using the command **PZ: Set Paper Size**.



The two latter examples are represented in the library of samples by the files *"Sample compound dialog with constant size of pages.GRB"* and *"Sample compound dialog with variable size of pages.GRB"*.

Control Elements Modification

To modify control elements, the control editing command is provided, "**EO: Edit Control**":

Keyboard	Textual Menu	Icon
<EO>	Edit > Draw > Control	

The control editing command can also be called from the context menu.

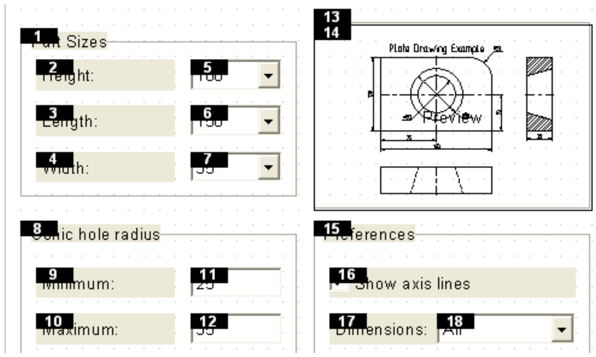
Upon calling the command, the following options become available in the automenu:

	<*>	Select All Elements
	<R>	Select element from list
	<T>	Change Controls Tab Order
	<S>	Set Recommended Page Parameters
	<Esc>	Exit command

The option is used for assigning default parameters of a dialog page to the current page (those set ar creation by the option). The page being modified is assigned the type "Controls". This page is automatically check marked in the list of pages under the **External Variable Editor** parameter in the command **ST: Set Document Parameters**. The parameter itself switches to the "Page" setting. The option is used on the pages created or modified by using the commands **Customize > Pages...**, **Customize > Page Size** or **Customize > Grid**.

The option allows changing the order of control element activation.

Control elements have a certain order that defines transition to the next element upon pressing the keys <Tab> and <Shift><Tab>, and priority, that defines the order of overlapping of the control element images. By default, both properties are assigned according to the order of element creation. That means, the transition will occur from the first created element to the next one, while the priority of each element is higher than that of the previous, and lower than that of the next one.



Upon calling the option , the order number is displayed next to each control element.

If you need to change the order of elements, select them in the desired order. To confirm the changes, press the icon or <Enter>. If you made a mistake when defining the order of elements, press or <Esc>, and then select the option again.

To modify a control element (or a group of elements), select it by clicking (you can also use box selection, <Shift>+ for multiple selections, selection by name from the list using the option , or selection of all elements on the page by the option).

If selecting a single element, you can modify its size. To do this, simply point the mouse to the border of the element rectangular area. The pointer shape changes to . Dragging the mouse with depressed changes the size of the selected element.

To move the selected element (group of elements), simply point the mouse inside the element rectangular area. The pointer shape changes to . Dragging the mouse with depressed changes the position of the selected element.

Upon selecting an element, you can use the following options (the full set of accessible options depends on the element type):

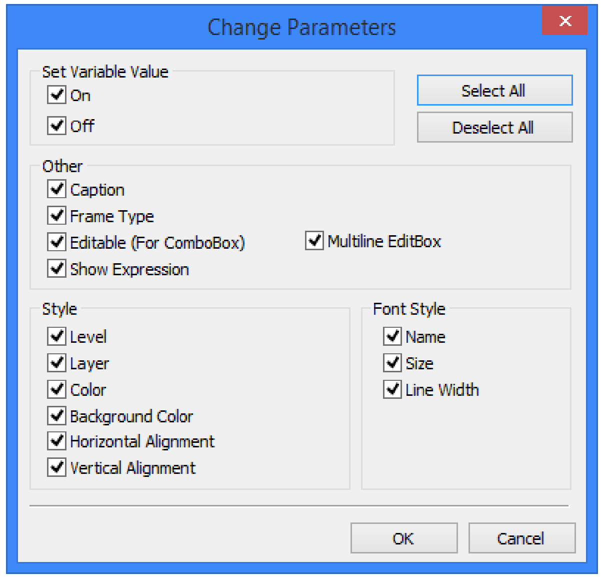
	<P>	Edit Control Parameters
	<V>	Change Control Variable
	<Y>	Create Name for selected Element
	<A>	Edit Actions List
	<I>	Select Other Element
	<Del>	Delete selected Element(s)

	<Esc>	Cancel selection

The option  is only accessible for the control elements that are bound to variables. This option allows reassigning the variable, whose value is controlled by the selected element (all steps in this case are the same as when creating the control element).

The option  is only accessible for the "Button"-type control elements. It allows adding/deleting the actions to perform when pressing the selected button.

The option  calls the property window of the selected control element. For "Frame", "Group Box", "Static Text" and "Preview" elements, this window can be called by double-clicking .



In the case of multiple selections, the option  first brings up the "Change parameters" dialog box. In this dialog box, you need to mark the parameters to be edited. By default, all parameters of the selected elements are subject to editing. If a parameter should not change, clear the check mark before its name. Upon specifying the set of parameters to be edited and pressing *[OK]*, the standard dialog box appears for defining the parameters of the selected elements. The parameters that cannot be modified are made inaccessible. Note that some general parameters ("Layer", "Level", etc.) can be edited using the system toolbar.

Optimization

T-FLEX CAD provides a capability for computing parameters of a 2D drawing or a 3D model by solving an optimization problem under certain constraints on the model variables. The solution to this problem yields a set of values for the existing variables that satisfy the imposed conditions best.

Main Concepts

Optimization of the model is done by using the command **PO: Optimize Model:**

Icon	Ribbon
	Parameters → Tools → Optimize
Keyboard	Textual Menu
<PO>	Parameters > Optimize

The command is accessible only when numerical variables are present in the document.

Calling the command brings up the dialog box "Optimization Task", that contains the list of the defined optimization problems. The column **Name** displays the name of the variable, whose value is optimized by the current task. The column **Comment** contains textual strings entered by the user.

A T-FLEX CAD document may contain any number of optimization tasks.

The graphical buttons at the bottom of the dialog box allow the following actions:

Add. Entering a new optimization task.

Delete. Deleting the task of the current list item.

Properties. Displays "Optimization Parameters" dialog box of the current list item task.

Run. Starts the optimization computations. The system seeks for the solution according to the specified optimization parameters and regenerates the drawing or the 3D model with the found variable values.

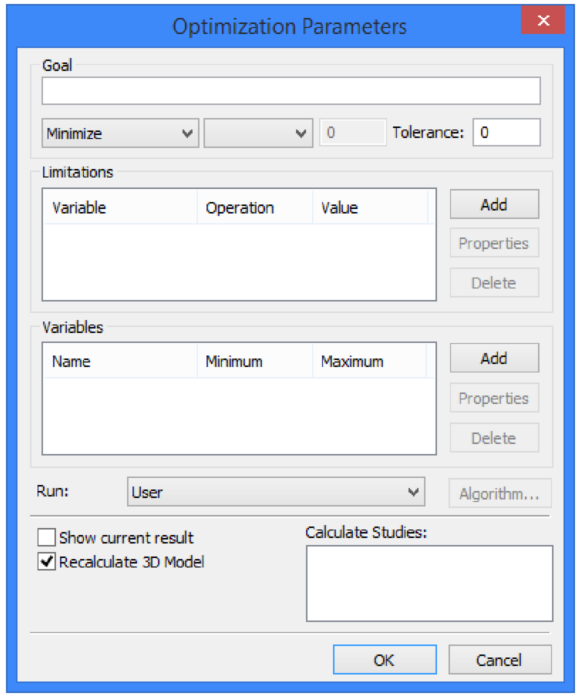
Close. Exits the command.

Optimization Task Definition

Upon pressing the **[Add]** button, the "Optimization Parameters" dialog box is displayed that contains the following fields:

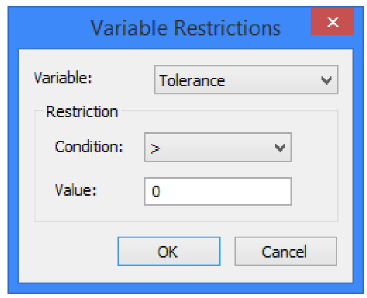
Goal. Keeps the textual string that is the comment of this optimization task. Next are the combo box for selecting the target function (Make Equal, Minimize, Maximize), the variable Name combo box and the Tolerance input box. The variable is selected from the list containing all numerical variables present in the document. If the selected function type is "Make Equal", then the input box becomes accessible for entering the target value of the variable.

The tolerance value defines the admissible range of the values of the target variable within which the optimization task is considered solved.

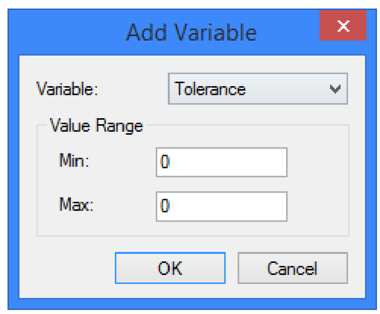


Limitations. Defines the list of restrictions on the model variables when performing the optimization. New limitations are entered upon pressing the **[Add]** button.

In the "Variable" box, set the name of the desired variable (one variable can be subject to multiple restrictions). In the "Condition" combo box, select one of the comparison types (<, >, <=, >=) for comparing the variable against the target value (the "Value" input box). To modify the entered limitations, use the button **[Properties]** that allows editing all fields of the current line in the limitations list. Pressing the **[Delete]** button deletes the current line of the limitations list.

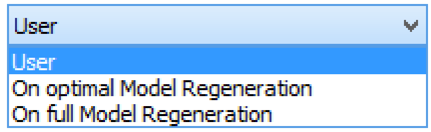


Variables. This is the list of variables whose values will be the subject to the optimization process. The range of admissible values is to be defined for each variable. To be formulated correctly, the optimization task requires defining the range of admissible values for at least one variable. The graphic buttons **[Add]**, **[Properties] and [Delete]** work similar to those described in the previous section. To define a new record, you need to fill in the following entries: Set the variable name by selecting from the list in the "Variable" combo box (each variable can have only one the range of admissible values).



The entries "Minimum" and "Maximum" define the bounding values of the variable's range of admissible values. When solving the optimization task, the values of the variables are tried that satisfy the set of the limitations and fall in the range of admissible values.

Once a limitation is defined for a variable, its name is no longer available for defining the range of values, and vice versa. The variable whose value is the target of the optimization is not included in the lists of variables when defining limitations and the value ranges.



Run. This parameter takes one of the following values:

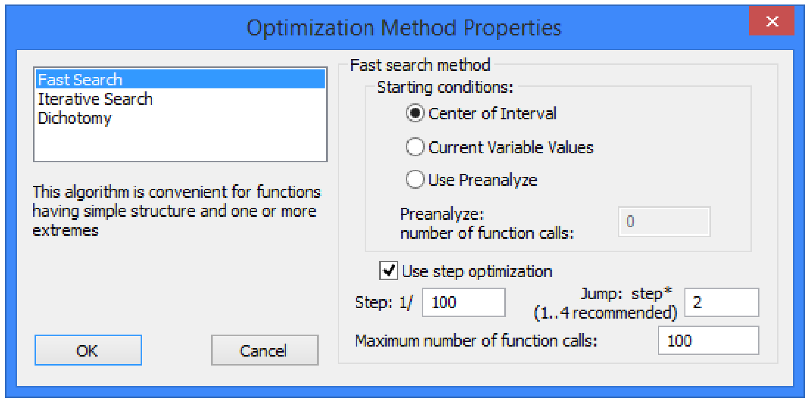
User. The optimization task will run only upon user pressing the bottom **[Run]** in the "Optimization Task" dialog box.

Optimizing may take long time on complicated drawings or 3D models. The described setting allows skipping optimization when regenerating the model.

On optimal Model Regeneration. The optimization task will run upon partial (optimal) model regeneration (when only modified elements are regenerated).

On full Model Regeneration. The optimization task will run upon full model regeneration.

To select an optimization algorithm and to define its parameters, use the graphic button **[Algorithm…]**. Pressing this button brings up the dialog box for defining the algorithm parameters.



The pane in the left side of the dialog box displays the list of available optimization methods:

Fast Search. This method is suitable for functions with one or two extremes.

Dichotomy. This method is suitable for functions of one variable. Not recommended for handling restrictions **Iterative Search**. This method is suitable for functions with complicated behavior and multiple extremes.

The right side of the dialog box contains the set of parameters for the selected optimization method.

The button **[Ok]** closes the dialog box, saving the entered changes. The button **[Cancel]** allows quitting the dialog box without saving changes.

Show current result. If this flag is set, the “Finding Solution” window dynamically displays the variable values as the solution progresses.

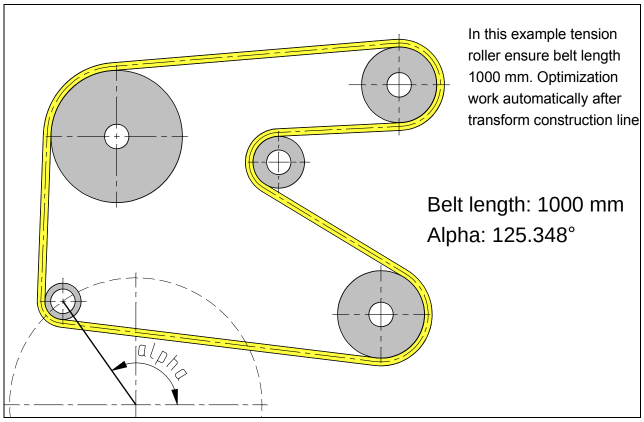
Recalculate 3D model. Setting this flag forces the 3D model regeneration at each step of the optimization algorithm. If the target function of the optimization (the variable) is related to 3D elements, then this flag is required for proper optimization process.

Examples of Using Optimization

Idler Roller Positioning Task

As an optimization example for a 2D model, let's consider the task of finding an idler roller position for fitting the belt of a specified length.

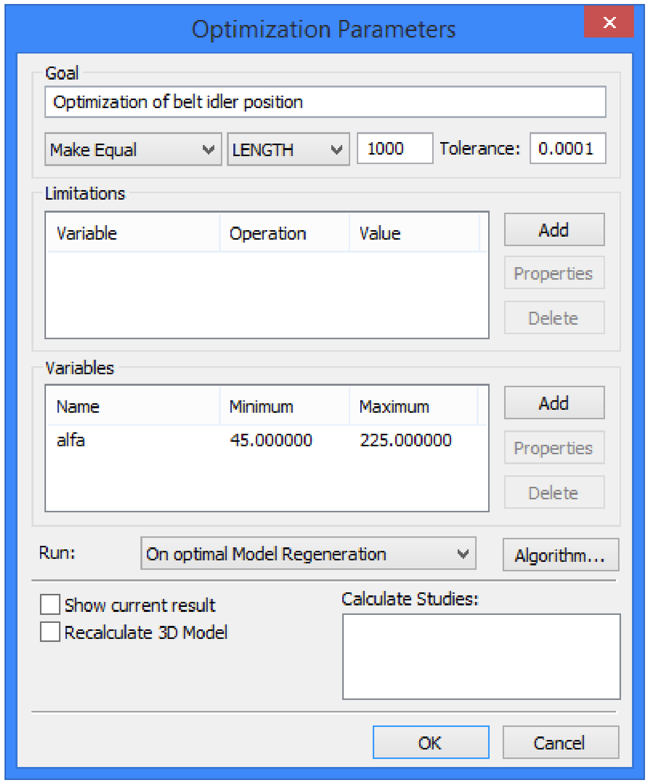
This example can be found in the library "Examples\Additional resources\Optimization\ Belt.GRB ".



The example presents a kinematic system of a generic belt drive. Let's assume one of the requirements for the design of the system being the fixed belt length (1000 mm), adjusted by the position of an idler. The idler position depends on the angle of turning its mounting bracket.

The turning angle of the idler mounting bracket is defined on the drawing by the parameter of the respective construction line created as "through node, at an angle to horizontal". This parameter of turning the construction line was defined by a variable named "alpha". The length of the belt is defined by the variable "Length".

Let's define the optimization task for this model by calling the command **PO: Optimize Model**.



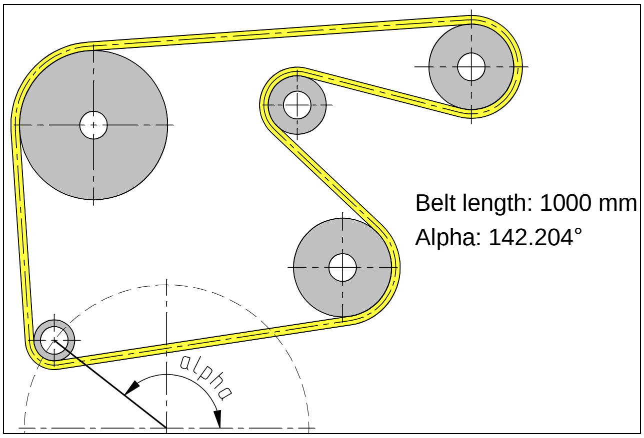
The target condition will be the equality: the variable "Length" is equal to 1000 mm with the tolerance 0.0001.

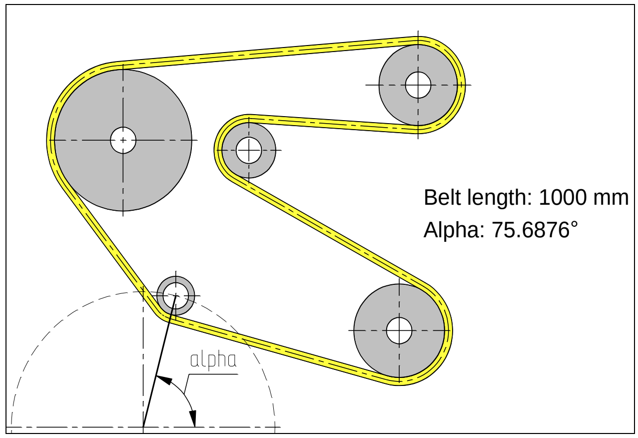
Let's set to the variable "alpha" to be the target of the optimization, with the values ranging from 45^0 to 225^0.

There are no additional conditions on the variable in this model, therefore, no restrictions are applied.

Let's set the optimization run condition "On optimal Model Regeneration". In this way, any model modifications will trigger the optimization, making it work in "transparent" mode. Let's set the dichotomy method as the optimization algorithm.

Upon defining all optimization parameters, any change in the drawing will trigger the optimization. For example, moving any of the rollers or changing any radius requires running the optimization to define the position of the idler. Meanwhile, the belt length is maintained equal or nearly equal to 1000 mm.





Bottle Volume Optimization Task

> This example clarifies use of optimization for 3D models. The file for the example is located in the library "`Examples\Additional resources\Optimization \Bottle.GRB`".

The example presents the solution to the bottle volume problem. In this example, the variable "Volume" is created, equal to the bottle capacity, that is, the volume of its contents. The variable "H" defines the height of the bottle, while "HW" – the level of the liquid in it.

The optimization task is to maintain the constant bottle capacity (0.5 liter = 500000 mm^3) while varying the bottle height and the level of the liquid in it. To achieve this goal, we need to find the value of the variable "D" driving the median diameter of the bottle (the widest portion diameter).

The optimization task "Volume" was defined in the command **PO: Optimize Model** as follows.

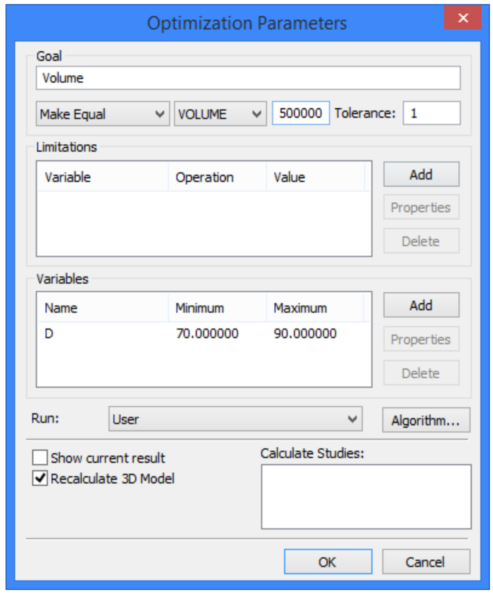
The target function equates the variable "Volume" to 500000 with the tolerance equal 1.

The modifiable variable is "D", in the range from 70 to 90.

No additional conditions are imposed on the model variables; therefore, no restrictions are defined. To visualize the optimization process, the flags are turned on, "Show current result" and "Recalculate 3D model".

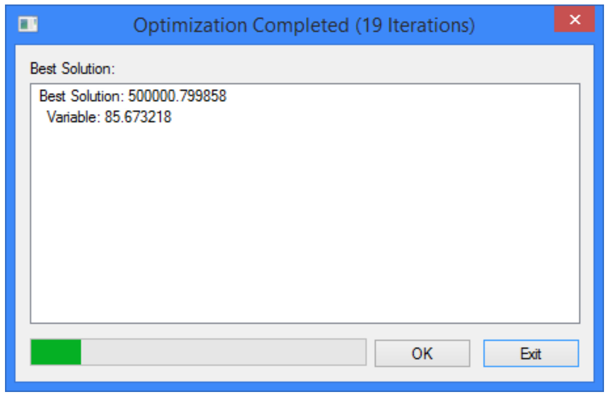
The dichotomy method is selected for the optimization algorithm, with the maximum number of iterations equal to 100.

The parameter "Run" is set to the option "User", so that the optimization is run only upon the user request.

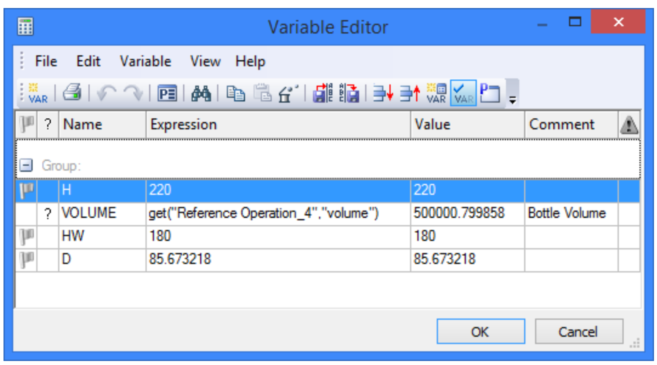


After specifying the optimization task, let's modify the model. For example, let's reduce the bottle height and, respectively, the level of the liquid in it, by changing the values of the variables "H" and "HW". This reduces the capacity of the bottle.

To adjust the bottle diameter, simply call the command **PO: Optimize Model**, select the task "Volume" in the coming up dialog box and press the graphic button **[Run]**. The shape of the model will be changing on the screen according to the current values of the variable being optimized as the solution progresses.



By accepting the found solution by pressing the button **[Ok]**, we get the bottle 220 mm high and 0.5 liter in capacity.



ASSEMBLY DRAWINGS

Basic Fundamentals and Concepts of Working with Assemblies

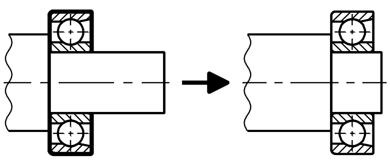
Introduction

Any drawing can be inserted in other drawings in T-FLEX system. For example, you can insert the drawing of the title block template into a part drawing, or, say, a drawing of a bolt into an assembly.

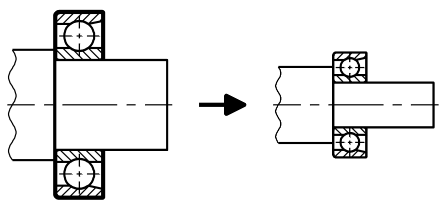
A T-FLEX CAD drawing is called *fragment* when inserted into another document. The drawing obtained by using fragments will be referred to as *assembly drawing*. The assembly drawing keeps only a reference to the original fragment file. Modifications in the fragment file propagate on the respective component in the assembly drawing.

Creation of drawings using fragments brings significant benefits in various cases. Firstly, this simplifies the process of creating complicated drawings, since you can create portions of such drawing first, and then join those. The design workflow of separate fragments can be completely independent, or, alternatively, be conducted in the assembly context, using the associative relations between the fragments and the assembly. Separating the assembly drawing into fragments corresponding to separate parts makes the assembly drawing fully represent the actual assembled mechanism. This approach also provides maximum automation to creating bills of materials of the assembly drawing, and delivers the complete set of detail drawings. Secondly, the use of assembly drawing supports reverse propagation of parametric modifications to any of the assembly parameters from the assembly to its contributing parts. This feature instantly yields a full set of part drawings satisfying those modified parameters. Thirdly, fragments can serve for representing frequently used drawing elements. Standard library elements can also be handy as fragments. For example, the title block template drawing can be added to a part drawing. You could also create special detailing elements to add to drawings.

Effective use of fragments in T-FLEX assembly drawings relies on the following fragment characteristics:

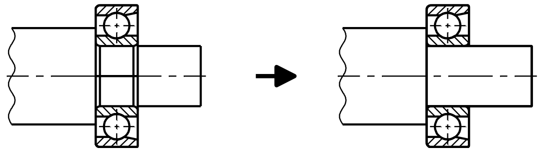


As any T-FLEX CAD elements, fragments can be attached to various elements of the assembly drawing, including other fragments. This allows coordinating fragment position modifications with other drawing elements relocations.

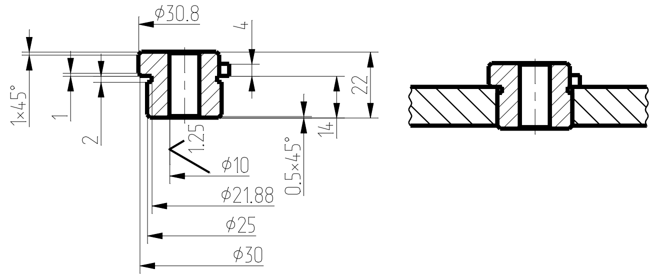


Since fragments are parametric drawings, they dimensions upon assembling are calculated according to the desired parameters of the assembly drawing. Note that the same fragment file can be used in the assembly document multiple times with different parameter values. This feature is especially beneficial when using libraries of standard elements.

Once assembled, fragments often overlap each other and the image of the assembly itself. T-FLEX CAD allows setting up automatic hidden line removal upon fragment overlapping.



You can do selective insertion of only necessary elements from the fragment drawing into the assembly drawing. This capability allows using fully furnished part drawings in assemblies.



Fragments can be variable, meaning that one or another fragment would be inserted in the assembly drawing, depending on certain conditions. This capability allows creating assembly drawings and modeling products in various configurations, when different parts are put in the same assembly, depending on the configuration.

Assembly drawings do not directly incorporate all fragment data. They only keep the necessary images and references to fragment drawings. This helps achieving most compact representation of the drawings in the memory and on the hard disk. Besides, if the same drawing file is inserted in different assemblies, then its modifications will propagate on all documents that use it.

Specifics of Handling Assembly Drawings

This section describes main capabilities and design techniques of fragments and assembly drawings. Detailed description of fragment-handling steps is provided in further chapters of this book.

Assembly Drawing Creation Techniques

Before starting with creation of an assembly drawing, think through its structure. Try to define requirements to its parametric layout: what specifically will be subject to modifications in the future, what parts will make up the drawing, what is the expected hierarchy of fragments. The conclusions made at this preliminary analysis stage will define the preferable technique of the assembly model and fragment creation. Assembly design techniques differ in the ways of creating fragment files:

- "Bottom-up" development. This technique implies creating part drawings first, to be inserted in the assembly, in a conventional way in separate T-FLEX CAD documents. Assembly drawing creation in this case implies subsequent assembling of the necessary fragments. In the course of development, one has to accomplish the task of attaching part images within the assembly drawing.

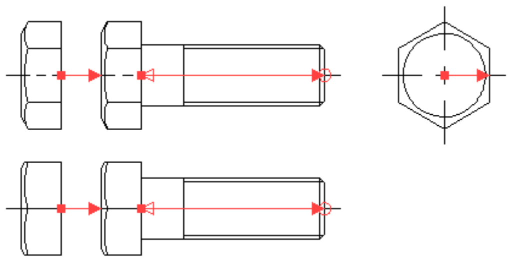
"Top-down" development. This technique implies that the part drawings originate from the assembly drawing. This means, the fragments are created in the assembly context. In this case, development start from creating the assembly drawing. Already created parts of the assembly drawing, including graphic lines and nodes belonging to fragments, can be used for creating new fragments. This approach simplifies creation of associative relations between the assembly fragments and the process of their attachment. The created fragments are saved into separate documents for further refinement and/or use in other assemblies.

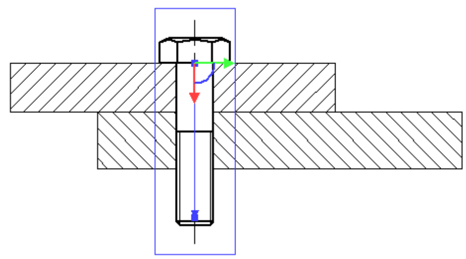
The described techniques can be combined. For example, a fragment created and inserted in an assembly according to the “Bottom-up” approach, can later be edited in the assembly context. Vice versa, a fragment created in the assembly context, can later be used for creating other assemblies within the “Bottom-up” framework.

Ways of Attaching the Fragment Image to the Assembly Drawing

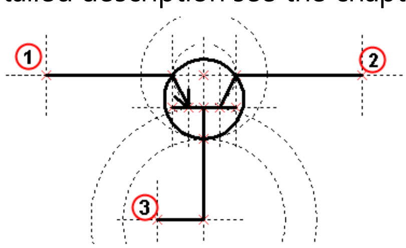
For placing the fragment image at a desired place in the assembly drawing, in the system T-FLEX CAD there are several different methods utilizing various tools depending on the problem being solved:

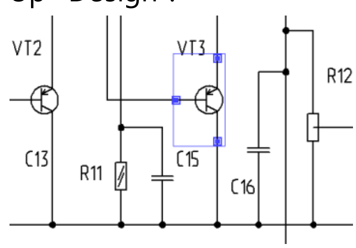
1. **Inserting the fragment with the help of fixing vector**. The fixing vector represents itself a special construction entity which serves as a two-dimensional coordinate system. When inserting the fragment into the assembly drawing, a user specifies location of the fixing vector, and the fragment image, connected to the fixing vector, is transferred along with that. The fixing vector is defined in advance in the fragment document (for more detailed description see the chapter “Bottom-Up Design”).



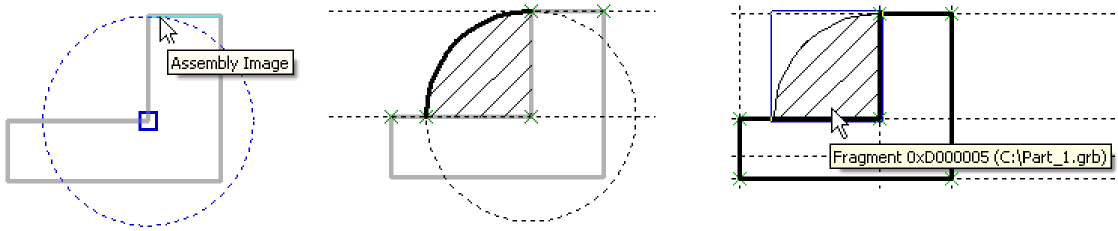


2. **Inserting the fragment by fixing points.** The fixing point represents itself an intersection of the horizontal and vertical construction lines the coordinates of which are specified by a pair of variables with special names x1…x9 and y1…y9. Accordingly, the number of fixing points can vary from 1 to 9. The entire parametric drawing of the fragment is constructed on the basis of the base construction lines determining the fixing points. When inserting such fragment, a user indicates a new location, and the parametric drawing of the fragment gets reconstructed depending on the new location of the base construction lines. This method is used when the fragment image (linear dimensions, shape, topology, etc.) has to be modified depending on the placement in the assembly drawing when position of the base fixing points is changed. For more detailed description see the chapter “Bottom-Up» Design”.



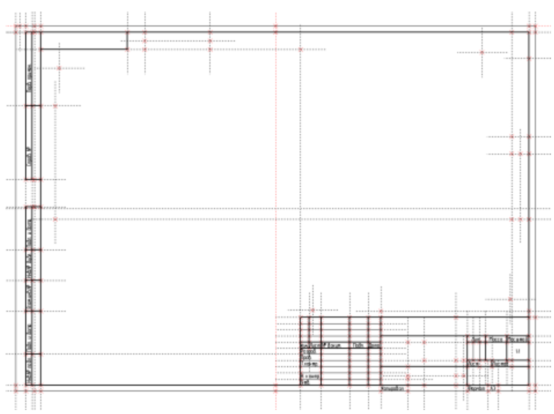


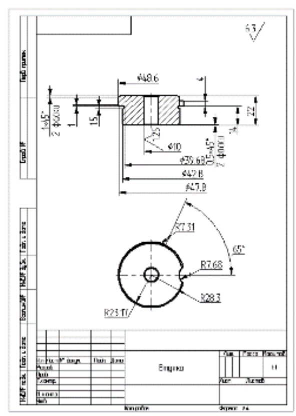
3. **Associative/Non-associative fixing to elements of the assembly drawing.** While designing by «Top-down» approach, upon creating a fragment in the assembly context drawing, a user can perform associative and non-associative fixing of the elements of the fragment to the elements of the assembly drawing (for more detailed description see the chapter "Top-Down" Design").



Without fixing (inserting «as it is», in absolute coordinates). When the fixing vector and the fixing points are absent while working with the «Bottom-Up» approach, and also, when all snaps are turned off while working with the «Top-Down» approach, the system puts the image of the drawing on the page of the assembly drawing without any changes. Each line or node of the fragment gets the same coordinates in the assembly drawing as the coordinates it had in the fragment document.

Use of special functions of variables. In some cases when inserting the fragment in absolute coordinates, the functions of the T-FLEX CAD variables can be used. For example, it is used for making the title block. In the parametric drawing of the title block the special functions that read the coordinates of the page borders of the assembly drawing are utilized. Then the coordinates are passed via the variables to the corresponding construction lines. Thus, the title block automatically takes the desired size in accordance with the specified page size of the assembly drawing.

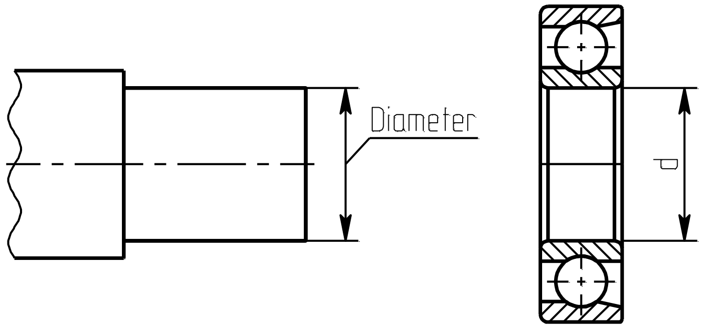




Use of Fragment Variables

When inserting a fragment, one can define the values of the variables that define the fragment drawing. For this, the desired variables must be marked as external when creating the drawing of the future fragment. For example, to be able to define a circle radius of a fragment when inserting it in other drawings, assign an external variable (say, "R") to the circle radius when creating the respective construction circle. After that, any time when inserting this fragment into other drawings, the system will ask for the value of the variable "R", and then adjust the fragment image according to the input value.

The external fragment variables play an important role by relating the fragment parameters with those of the assembly drawing. For example, consider a drawing representing a shaft, whose diameter is assigned the variable “***Diameter***”.



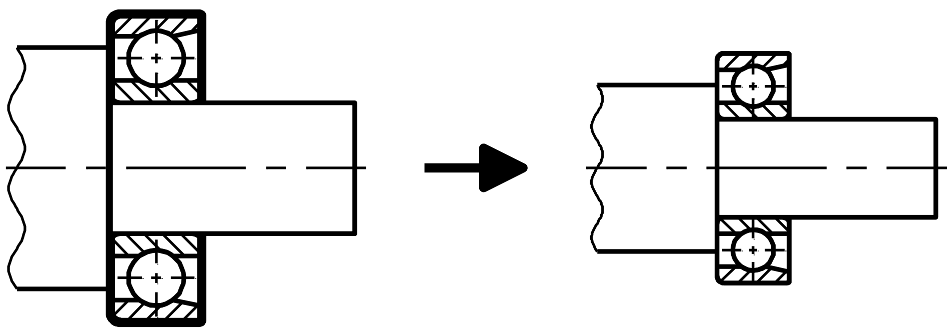
Suppose, we need to mount a ball bearing on the shaft. A variable “***d***” was created in the ball bearing drawing that is responsible for the value of the ball bearing inner diameter. The variable “***d***” is marked as external. All the rest of the ball bearing parameters are interrelated in such a way that they are driven by the value “***d***”.

Variables

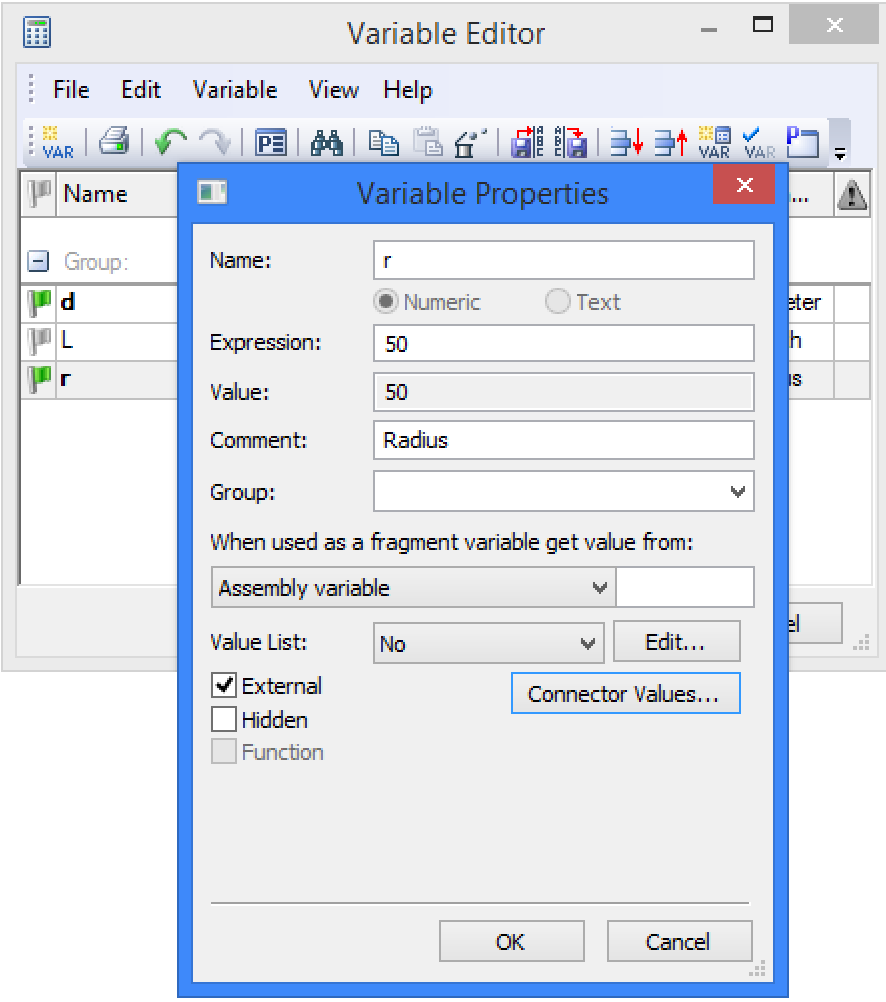
Name	Comment	Expression
d	Diameter	10

Now, as the bearing is inserted in the shaft drawing, the two variables can be related.

After the above steps, modifications to the shaft diameter, that is, to the variable “***Diameter***” of the assembly drawing, will be automatically propagate as changes to the variable “***d***” in the fragment, leading to the desired adjustment of its image.



An assembly variable name can be set the same as an external variable name of the fragment. When inserting a fragment in the current assembly drawing, if a same-name variable is found in the fragment as an assembly variable, the latter will be automatically assigned the respective external fragment variable.



For automatic specification of connections between the values of the fragment variables and the variables of other fragments in the assembly, the mechanism of connectors can be used (see below for details).

When working with a large number of external variables of a fragment, it is convenient to use *configurations*.

Configuration – is a named, stored in the document, collection of values of external variables for the fragments of the document and 3D geometry corresponding to these values. In the 2D document, only the values of external variables are stored in the configuration. The work with configurations will be described in more detail in the chapter "Auxiliary Tools for 3D Assemblies Modeling" of the book "T-FLEX Parametric CAD. Three-dimensional Modeling".

If in the document of a 2D fragment, configurations are created, then, when inserting and editing the fragment, the user can select one of the fragment's configurations. The values stored in the selected configuration will be automatically assigned to all external variables of the fragment.

The name of the used configuration can be specified as a variable. If the value of this variable changes, the system will select the corresponding configuration of the model and automatically modify the values of the fragment's variables.

Visibility Management of Fragment Drawing Elements

A part drawing may contain an image that should not be included in the assembly drawing. Alternatively, we may need to use one or another portion of the same part drawing, depending on circumstances. (For instance, in one case we need to show the top view, while in the other – the front view.) Visibility of the fragment drawing elements inserted in the assembly can be controlled using layers or visibility levels. Two approaches are possible when using layers.

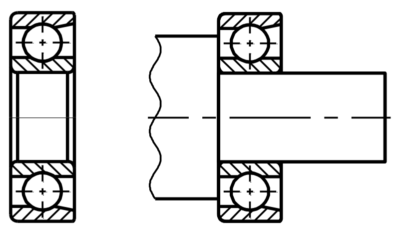
The first one is using a layer's own attributes ("Hidden when model is used as a Fragment", "Visible only when model is used as a Fragment"). This way does not allow defining several configurations of the fragment image based on the same drawing. Nevertheless, it allows hiding/showing the elements of the fragment part drawing that are definitely necessary in the part drawing, yet must be hidden in the assembly (or vice versa). This can be, for example, the part dimensions, the title block, etc.

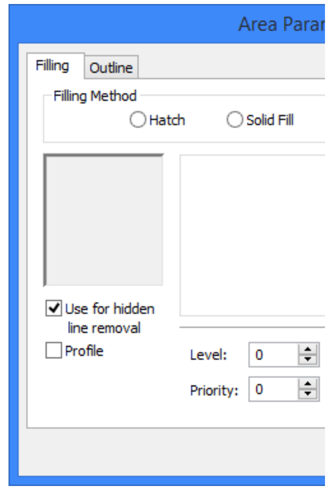
The second approach is more flexible, and can be used when positioning the fragment by a fixing vector. The fixing vector parameters can relate the vector with selected layers (see chapter "Bottom-Up Design"). Thus, a number of part configurations (such as part views) can be obtained from the same drawing by creating several fixing vectors with different types of relation with the layers in the drawing.

Control over the visibility of the fragment drawing elements is done by using visibility levels in a way common across all drawing elements. This approach may require use of external fragment variables. Such external variables can be created in the fragment drawing, and then carried over to the assembly, in order to control the visibility levels of the fragment drawing. The visibility levels of the fragment drawing elements will obey the external variable settings both while editing in the assembly context and in detailing (in the exported assembly fragment instance).

Hidden Line Removal in Assembly Drawing

One of the important advantages of T-FLEX fragments is the hidden line removal mechanism when building an assembly from fragments. This allows, on one hand, creating a complete drawing of the desired part, while, on the other hand, "hide" the drawing lines that fall behind the image of other parts of the assembly drawing.

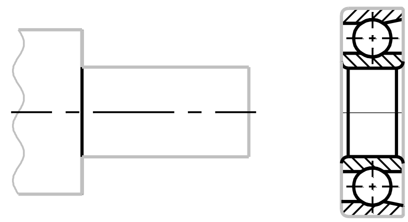




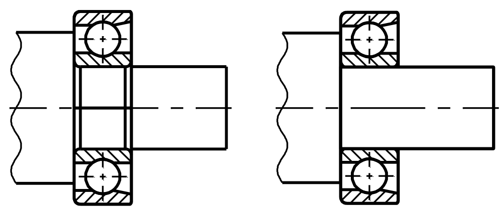
The area in the drawing subject to hidden line removal is defined using a hatch. To remove hidden lines, you can use an existing hatch or create an additional invisible one (using the fill method **Invisible**). In the hatch parameters, set the flag **Use for hidden line removal**. In this case, the invisible hatch will be hiding behind the objects with lower priority.

Control over the visibility of a covered up element is done by setting an appropriate priority in its properties. If a fragment is supposed to cover assembly lines, then the fragment priority should be set higher than that of the assembly elements, which the fragment should cover. Vise versa, if the assembly lines should cover the fragment lines, then one would have to create a hatch in the assembly drawing itself, and then set its priority higher than the priority of the respective fragment.

Let's review hidden line removal on the example of a shaft drawing and a ball bearing fragment.



We need to create a hatch in the shaft drawing for hidden line removal. Upon inserting the fragment in the assembly, the ball bearing fragment image will be overlapping the shaft image. For the shaft image to cover the lines of the ball bearing, we need to set the priority of the ball bearing fragment lower than the priority of the hatch contour on the shaft. This can be done by the immediately at the time of inserting the fragment, or later, when editing it.



Upon modifying the fragment priority, refresh the image by calling the command **RD: Update Model Windows** (by pressing <F7>). As a result, the image will appear as shown on the lower right diagram.

Snapping the Fragment Elements

Even though the lines and other elements of a fragment are not part of the assembly drawing that includes it, those can be used as references for creating various elements of the assembly drawing. Ordinary lines of the fragment image (arcs and circles) can be used at the time when the object snapping is turned on. In this way, one can create new construction lines on top of the fragment graphic lines or dimension those, or attach any other detailing elements.

Besides the graphic lines, one can use the fragment nodes as references. To engage use of fragment nodes, you need to activate certain settings. The flag **Fragment Nodes** must be set in the command **SO: Set System Options** on the tab **Snap > Priority**. This setting allows creating assembly drawing elements that rely on the named fragment nodes or end points of graphic lines. In this way, the new nodes based on fragment nodes are created in the transparent mode.

If the image of the fragment is crowded with various elements, working in transparent mode may become somewhat difficult. In this case, you can turn off the discussed setting and work in the mode of forcing creation of necessary nodes based on a fragment:

1. The command **N: Construct Node** allows creating only the nodes that will later be actually used in the assembly constructions (a detailed description is provided in the respective chapter).
2. Named fragment nodes can be created automatically when inserting the fragment, provided that the flag **Create Named Nodes Automatically** is set in the system settings (the command **SO: Set System Options**, the tab **Fragments**).

Assembly BOM Creation

Creation of bill of materials is one of the important steps in handling assembly models. Detailed description of handling BOMs is provided in chapter **Product Structure, Reports, Bills of Materials**.

For automatic filling of BOM records, you need to make sure, that the parts (fragments) of the assembly document contain sets of the relevant data. BOM data can be defined in the fragment part document in "Product structure" window at any stage of creation. You can specify the way of using the inserted elements (other fragments) and their data contribution in the product structure of assembly. This allows including in the product structure the data about inserted fragments or append data from product structure belonging to a fragment document. Defining an appropriate mode simplifies building bills of materials of multi-layer assemblies.

To obtain the BOM of the assembly document, you need to do the following steps:

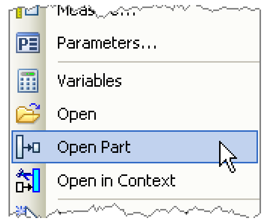
1. Enter the BOM data in the product structure window in the fragment document.
2. In the assembly document, define the way of including the fragment in product structure, either in the fragment parameters or in the command **BI: Include in Product Structure** (**Tools** > **Report/Bill of Materials** > **Elements**).
3. Using the command **BC: Create Report/Bill of Materials**, generate the assembly BOM.

Modifications of the assembly drawing are automatically reflected in the BOM. If necessary, an arbitrary number of BOMs can be attributed to an assembly drawing.

Detailing Drawings Based on Fragments

When building an assembly, the fragment drawings may be modified according to the assembly parameters via modifications of the fragment external parameters or the whole drawing when using associative attachments. In this case, the fragment drawing files are not changed. However, if necessary, one can automatically create separate documents with the drawings of the fragment parts adjusted to the assembly parameters. We will refer to such drawings as detail drawings. One can create detail drawings for the whole set of parts making up the assembly drawing. Once created, a detail drawing does not maintain any relation with the source assembly drawing.

To get a detail drawing, use the option  in the commands **FR: Create Fragment** and **EFR: Edit Fragment** or the command **Open Part** in the fragment context menu.

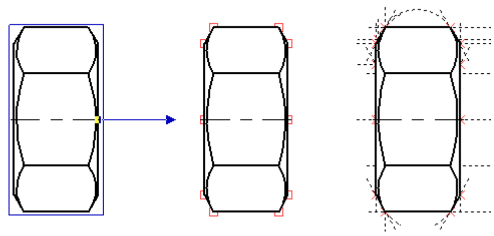


Upon calling the command, a new document window will be opened, with a copy of the fragment drawing loaded in it, with the external parameter values and associative relations inherited from the assembly. The new drawing will be named “Part” with a subsequent number, for example, “Part 1”. This drawing will be treated by the system is a new document, with the system asking to define the name to save the drawing as upon an attempt to close the window.

Usually, detail drawings are used for printing out hard copies of the detail drawings that can be different from the original fragment drawing due to, for instance, modifications of the assembly variables. In such case, you don't even have to save a copy of the detail drawing into a file. This is convenient in the case when a parametric fragment used in the assembly has a corresponding fully furnished drawing. In such a case, upon creating the detail, the user instantly gets a complete set of new documentation for the desired part.

Exploding Fragments

An inserted fragment can be exploded. By this operation, the fragment is removed, while instead the copies of all fragment elements are created in the assembly drawing that were deemed visible at the time of assembling.



The system can explode selected fragments in two ways, creating drawing elements with or without construction entities. In the first case, all parametric relations are preserved between the former fragment elements by carrying over all necessary construction lines for maintaining parametric relations of the former fragment drawing. In the second case, the fragment is transformed to a set of graphic lines attached to free nodes.

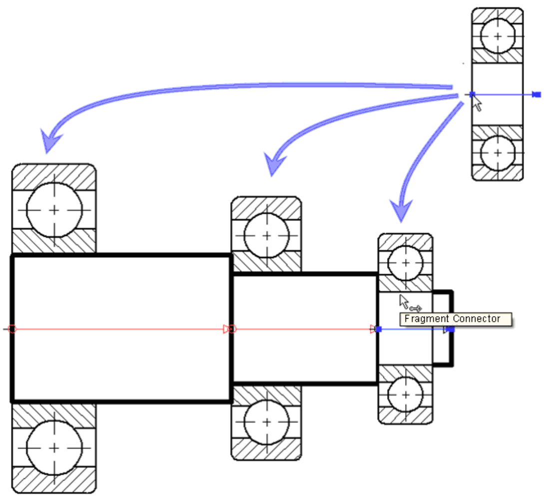
A fragment that contains nested fragments is exploded into the graphic elements and fragments that it contains. After using the exploding option, the obtained elements can be treated as usual, according to the type.

If some elements in the assembly drawing were attached to fragment elements (dimensions, construction lines, etc.), then, upon exploding, those will be re-attached to the elements created by the fragment explosion (nodes, construction and graphic lines).

Use of Connectors

When creating assembly documents, it is often necessary to relate parameters (variables) of the inserted elements with the parameters of the elements to which the attachment is made. Examples are: setting a ball bearing on a shaft, attaching a cap to a ball bearing, a shaft key to its slot, a nut to a bolt, insertion of a screw or being into a hole, etc. When inserting such elements, the user is required not only to define the main dimension parameters (diameter, length, etc.), but also precisely position the elements being inserted with respect to the target element (selection of the attachment point and direction). One of the ways of solving this task is using "Measuring" mechanism. This mechanism, however, often requires large amount of auxiliary preparations. **Connector mechanism** helps significantly simplify the procedure of assembling elements and minimize the number of actions required from the user.

This mechanism is based on the concept of a "connector" which is a construction element serving as a reference for attaching other elements. In fact, a connector is a counterpart of a fixing vector or a target LCS for attaching 3D fragments. Its main difference from a fixing vector is that a connector serves for providing attachment reference to other model elements. Consider, for instance, use of a fixing vector for inserting a bearing image into an assembly drawing. Suppose, the image of the bearing should be attached to an axle. In this case, the connector should be attached to the axle image (fragment).



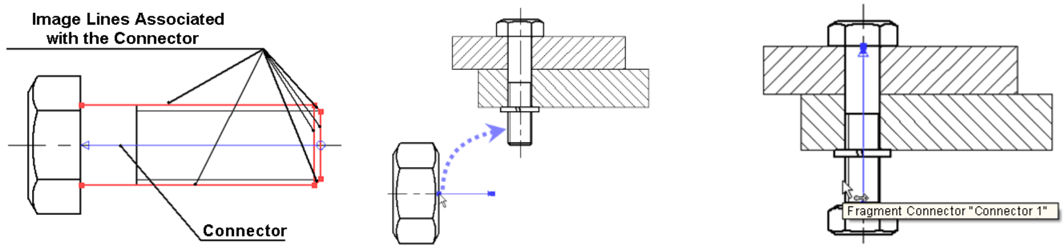
Besides the geometrical positioning (the origin position and the axes directions), a connector may keep additional information necessary for "snapping" to it other elements.

This information is kept in the connector in the form of named values that can be constants or variables. The names of these values are used for specifying the values of the respective external variables of the fragments being snapped to the connector. For example, a connector placed on the axis of a hole, can have such parameters as the hole depth and the diameter. When inserting a pin in this hole, its diameter can be automatically defined by the value D kept in the connector. To achieve this, the external variable defining the in diameter should also be named "D".

There are a number of considerations when using a connector for snapping a fragment:

- A connector is a construction entity and may not be displayed in the main drawing, while it has to be selected when snapping a fragment to it.
- A connector may be located outside the viewable area of the drawing.
- Sometimes, it is convenient to use a connector for placing an element elsewhere. For example, when attaching a cap to a ball bearing, it is convenient to select the outer lines of the ball bearing. Meanwhile, the intended connector is lying on the ball bearing axis.

A concept of "associated elements" is introduced for dealing with such questions and for overall convenience. The list of such elements is stored in the connector. Associated elements are necessary for in-depth utilization of the object snapping mechanism when snapping to a connector. As the pointer approaches one of the image lines associated with the connector, the connector is automatically activated (highlighted in the screen), and the external variables of the fragment receive the values from the connector. The 2D fragment is automatically calculated with the new variable values and is snapped to the attachment point with the appropriate orientation.

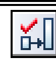


All libraries of the standard T-FLEX CAD elements are already equipped with connectors and ready for their use. The required named values for linking with the connectors have been set in advance for the driving external variables.

Composition Document. Embedded Fragments

T-FLEX CAD provides a mechanism for managing references to other documents (fragments, pictures, external databases, etc.). A T-FLEX CAD document may contain references to external files ("external link") or have the external file data embedded directly within the file of the composition (assembly) document in T-FLEX CAD (“internal link”). This mechanism also helps quickly port an assembly model to another location in the file system, zipping the assembly in one file with the capability of quickly unzipping it later.

The **File > Assembly…** group of commands is often used in the system to manage references. Working with commands of this group is described in the chapter **Links. Managing Composite Documents**.

Keyboard	Textual Menu	Icon
<UL>	File > Assembly > Update Links	
<UA>	File > Assembly > Update Assembly	
<AL>	File > Assembly > Links…	
<AM>	File > Assembly > Move Assembly…	

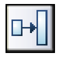
The command **UL: Update Links** redownloads data from the external files of the first level which are included into the composition document. The command **UA: Update Assembly** re-downloads data from the external files and embedded documents as well.

The command **AL: Links** is provided for managing the assembly document links to other documents. Each of the links can be assigned a type defining the position of the external file.

The command **AM: Move Assembly** is provided for moving an assembly document to another location in the file system or for zipping the assembly into one file. When moving the file, the moving options can be specified (embedding, substitution), as well as the name and the path to the target document of moving the assembly.

List of Commands Used in Assembly Design

The commands for inserting and editing fragments:

Keyboard	Textual Menu	Icon	Brief Description
<FR>	Draw > Fragment		The main command for inserting fragments and creating new fragments in assembly context
<EFR>	Edit > Draw > Fragment		The command for editing fragments. Is used for modifying attachment parameters, defining properties, editing fragment variables, etc.

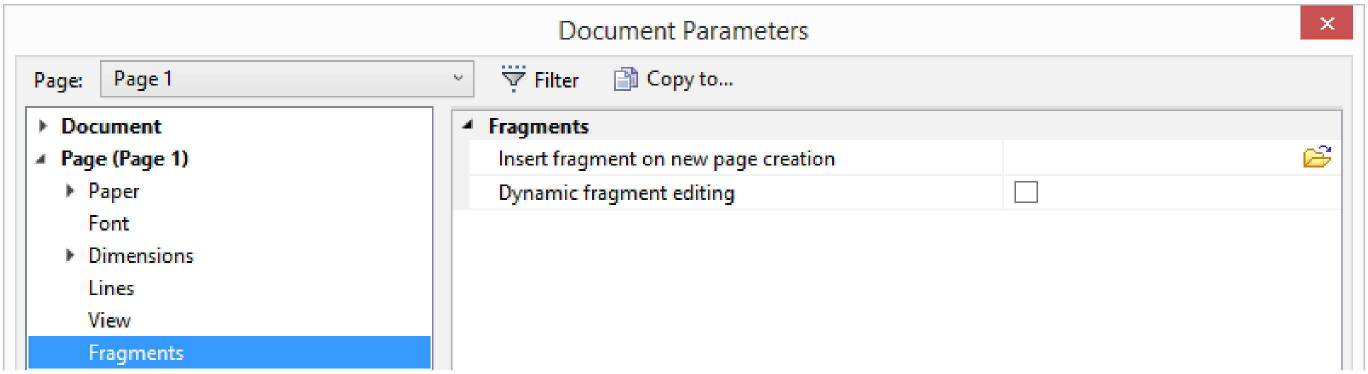
The commands for "Top-down" design:

Keyboard	Textual Menu	Icon	Brief Description
<FM>	File > Fragment > Create		The command for creating new fragment in assembly context.
<FX>	File > Fragment > Extract Fragment Drawing		The command for creating new fragment from assembly drawing elements. Moves selected elements to fragment file.
<FF>	File > Fragment > Apply		The command for finishing fragment editing in assembly context with saving fragment.
<FQ>	File > Fragment > Cancel		The command for finishing fragment editing in assembly context with or without saving fragment.
<FG>	File > Fragment > Update Files		The command for updating fragment created by "Top-down" approach, per changes in the assembly.

The command for handling the assembly structure:

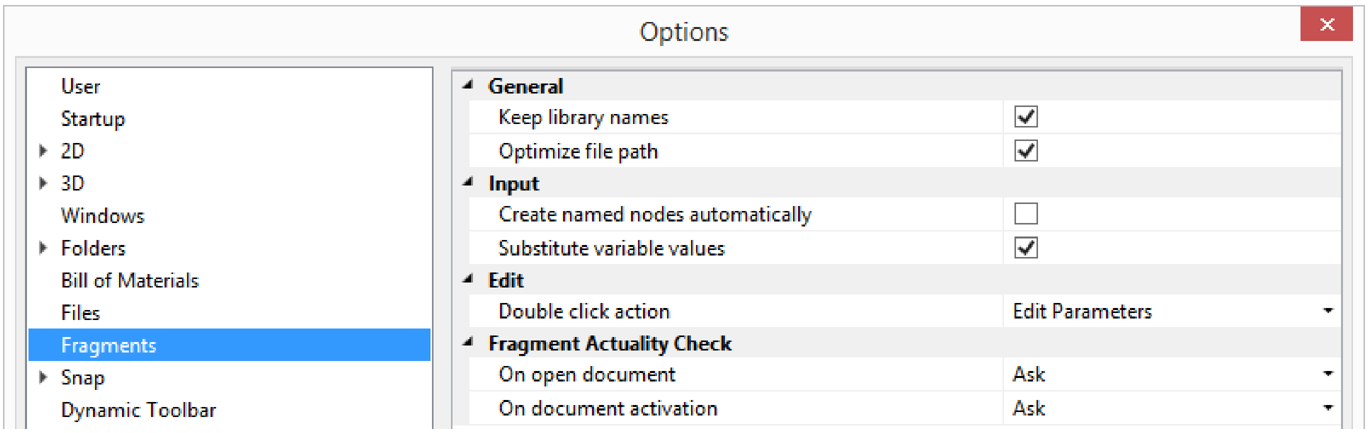
Keyboard	Textual Menu	Icon	Brief Description
<SS>	Tools > Special Data > Structure…		The command for managing the assembly model structure. Displays all fragments contributing to the document as a tree in a special window, or in a text file (embedded fragments are included).

The current document settings are done by the command **ST: Set Document Parameters,** the tab **2D Fragment**. These settings will be applied to the assembly drawing when using the current document as a fragment.



On the tab **Fragments**, the mode **Dynamic Fragment Editing** used upon creating and editing a fragment can be turned on. When this mode is turned on, the variables editing or the fragment attachment results in redrawing the elements connected to the fragment. While doing it, preliminary image of the fragment being inserted is displayed with the same quality as the final image. Dynamic modes enhances the clarity of the editing process. It can be useful for editing schemes, plans, etc.

The system settings for handling fragments are accessed by the command **Customize** > **Options**, the tab **Fragments**:



The command for starting the application «Document converter»:

Icon	Ribbon
	File → Document Converter
Keyboard	Textual Menu
<AC>	File > Document Converter

This command allows carrying out automatic or forced, complete recalculation, diagnostics or saving of the entire assembly together with the files of the fragments included into the assembly. Documents created under the previous versions of the T-FLEX CAD are adjusted in accordance with the format of the current, newer version of the software. When the work is finished, the report is generated which can be used for diagnostics and error search in the structure of the assembly model. Also, the converter is used for transferring the documents to the T-FLEX DOCs system. Details on working with this application can be found in the chapter “Converting Documents Created in Earlier Versions of T-FLEX CAD”.

«Bottom-Up» Design

When using the "Bottom-up" approach, the assembly model design starts from creation of separate assembly elements – fragments. A fragment drawing is originally created as a standalone T-FLEX CAD document. In the process of creation, it is necessary to follow the certain rules that will allow in future to "snap" the fragment to the elements of the assembly drawing.

A special mechanism is provided in T-FLEX CAD for correct positioning of the fragment drawings in the assembly – **Fixing points or vectors**. Such elements should be created in a fragment drawing in advance, before inserting it into an assembly drawing.

When inserting a fragment, you need to specify the position of the reference elements in the assembly drawing, which will define the fragment position, orientation and size. The fragment image will be built according to the specified fixing points or vectors. If no attachment elements are created in the fragment drawing, such fragment will be attached to the coordinate system of the assembly drawing page according to its initial coordinates in this page. The position of such fragment can be modified only by editing. To get associated relations between the fragments and the assembly drawing, attached fragments to the nodes of the assembly drawing (including the nodes of other fragments).

Ways of Attaching Fragments

When designing assemblies by «Top-down» approach, two ways of positioning the fragment on the drawing are primarily used in the T-FLEX CAD system:

Defining fixing vector. In this way, first create the drawing, and then define the necessary number of fixing vectors. Each fixing vector defines the origin and the positive direction of the X-axis of one local coordinate system in the drawing. An arbitrary number of fixing vectors can be defined within a fragment drawing. The fixing vector defines the fragment position and orientation one the assembly drawing and controls elements visibility.

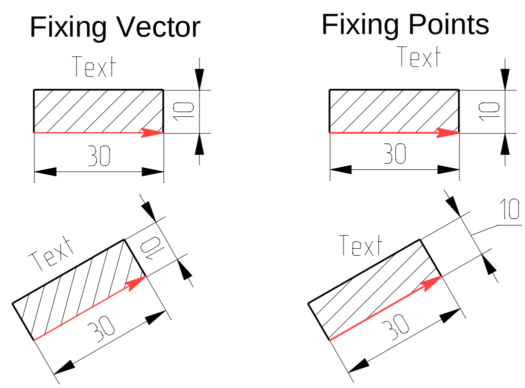
Defining fixing points using variables. In this way, the drawing is created first, following certain rules. The base vertical and horizontal lines are assigned the parameters with the reserved variable names. Upon assembling, the system will identify the intersection point of such lines as a fragment fixing point. Multiple fixing points are allowed. All the rest of fragment constructions are done with respect to the base lines defining the fixing points. The fixing points can define the position, orientation and size of the fragment within the assembly drawing.

The main difference between the fixing vector and fixing point approaches is in the sequence of creation steps of the fragment drawing. When using fixing vectors, you need to create the drawing first, and then specify the fixing vectors. When using the fixing points, one has to create fixing points first, and then build the drawing of the part based on the fixing points.

There are also differences in the uses of fixing vectors and points. Fragments with fixing vectors are created as follows: first, the image of the fragment is formed, taking into account the visibility layers.

Next, the complete image is moved to the specified point and rotated by the specified angle without distortions. Fragments with fixing points are created differently: upon specifying the fixing points, all construction elements dependent on those points are rebuilt first, and then the image of the fragment is obtained.

This difference in forming the fragment image on the assembly drawing leads to differences that can be illustrated by the following example. Let's create identical fragments with different fixing provisions. Insert these fragments into a drawing, rotated by the same angle.



The fragment that uses a fixing vector, exposes a change in the hatch angle, nonstandard dimension placement (with the automatic dimension text orienting functionality disabled), and a changed angle of a text. The fragment that uses fixing points shows the hatch angle preserved, the dimension “10” rotated per the standard, and the text angle maintained as well.

The mentioned differences shall not be considered shortcomings; rather, those are features that can be exploited in certain design situations.

Fixing Vectors. Connectors

A fixing vector and a connector are auxiliary elements of the model which are used when attaching the fragments. These construction elements are constantly displayed on the screen and can be hidden together with other construction elements by the command “Hide Construction”. Context menu is available for those elements, providing the deletion, editing and property modification commands.

To use a fully furnished drawing as a fragment, one needs to create a fixing vector. To prepare the environment for fast “snapping” of other fragments, one needs to create a connector. The connector does not have to be created in the fragment drawing. It can be created in an assembly drawing.

Fixing vectors and connectors have different purpose, however, are created in the same command **FV: Construct Fixing Vector**:

Icon	Ribbon
	Draw → Insert → Fixing Vector
Keyboard	Textual Menu
<FV>	Construct > Fixing Vector

Upon calling the command, the following options appear in the automenu:

	<F>	Create Fixing Vector
	<C>	Create Connector

	<A>	Set Snap Elements
	<N>	Select Node
	<F4>	Execute Edit Fixing Vector Command
	<Esc>	Exit command

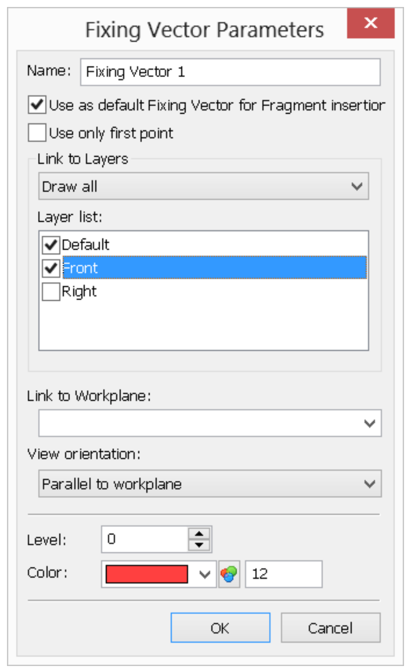
Fixing vector properties

There are two types of fixing vectors: a fixing vector defined by two points, and a fixing vector defined by one point. Fixing by one point is used for fast attachment of parts whose image does not change under rotations or parts not requiring rotation.

When creating a fixing vector, in its parameters specify the layers of the drawing to be displayed in the assembly. This allows, for example, inserting different views of the same part in an assembly.

If the 3D model exists in the document, then the fixing vector can be bound to a specific workplane. This allows using 2D fragments in three-dimensional assemblies for defining the position of a three-dimensional fragment in the space with respect to the final position of the fixing vector and the selected workplane. This assembly creation technique is called "Layout". Detailed information on this technique can be found in the three-dimensional modeling manual, the chapter "3D Assemblies Creation". This kind of relation allows inserting a 3D fragment automatically when the respective 2D fragment is assembled. To enable this feature, set the flag **Auto create 3D Fragment** in the command **FR: Create Fragment**.

Fixing Vector defined by two points



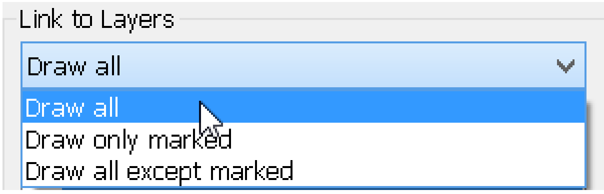
When inserting a fragment with such fixing vector, the position of the first point (the vector start) will be defining the fragment placement position within the assembly drawing, while the second point position (the vector end) - rotation of the fragment in the assembly.

To create such vector, subsequently select two nodes by clicking . Once the second node is selected, a dialog box appears in the screen. You can enter a comment for the fixing vector in this dialog. The comment will help selecting the fixing vector when inserting the fragment into an assembly drawing.

The attribute **Use as default Fixing Vector for Fragment insertion** defines what fixing vector will be offered by default when inserting the fragment into an assembly drawing. Although multiple fixing vectors are allowed in a fragment drawing, only one fixing vector can be defined as the default.

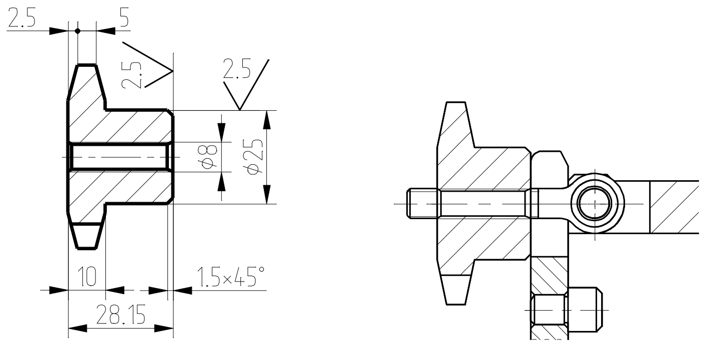
The group **Link to Layers** serves for managing display of the drawing layers upon assembling. The parameters in the group affect only the display of 2D fragment elements. They do not affect visibility of 3D fragments placed on the respective layers.

Layer list. This pane contains the list of all layers existing in the drawing. Here you can mark those that will be displayed when assembling the document as a fragment using this particular fixing vector.



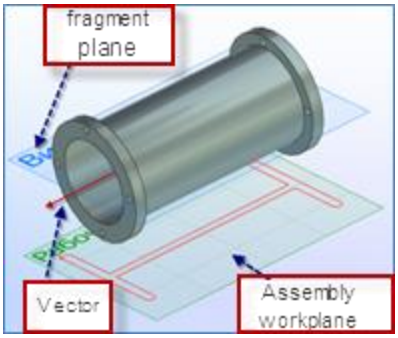
On the diagram below, highlighted are the fixing vector and the drawing elements to be carried over to the assembly drawing. Those elements are assigned a new layer. Exactly this layer must be check marked in the fixing vector parameters.

After that, when assembling these drawing as a fragment, only the check marked layer will be displayed in the assembly drawing, and, therefore, only the elements that lie in this layer will be displayed.



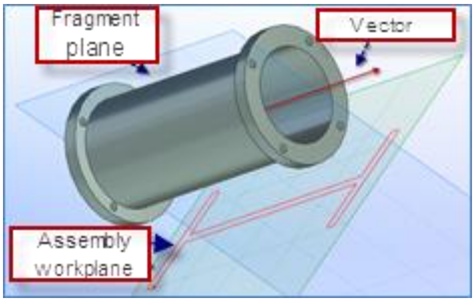
If a fixing vector has linked layers, its context menu will provide the command "**Apply Layer visibility**". Calling this command shows or hides the layers in the way in which they will be displayed when assembling the document as a fragment, using this fixing vector.

Link to Workplane. This parameter has a meaning only in the 3D version of the system. It servers for defining the workplane to which the given fixing vector will be bound. This functionality is used when creating layouts (see the chapter "3D Assemblies Creation" in the volume "Three-dimensional modeling").



View orientation. View orientation is used when creating layouts (2D fragment on a workplane by 3D fragment). It allows us to select condition for which the schematic view will be added to the assembly's plane. Only three options are possible:

Parallel to workplane – after addition of the model to the assembly, the model's schematic image will be created only under condition that the workplane of the assembly is parallel to the workplane in the file of the fragment with which the vector is linked.

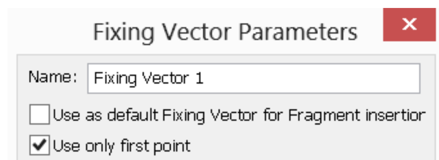


Axial symmetry – it is used for models of body of revolution type. After addition of the model to the assembly, the model's schematic image can be created on any plane of the assembly which is parallel to a line on which the fixing vector is lying.

Arbitrary orientation – it is used for bodies with central symmetry or having identical pseudo-graphical display of several views. After addition of the model to the assembly, the model's schematic image can be created on any plane.

Fixing Vector defined by one point

To create such fixing vector, select a node in the drawing by clicking and press <End> or in the automenu. The same dialog box will appear in the screen as the one described for the other type of the fixing vector. In this dialog box, check the flag **Use only first point**.



When inserting a fragment, only one point will be asked. It won't be possible to define rotation for such fragment. If the flag **Use only first point** is not set, then a two-point fixing vector will be created. The direction of such fixing point will coincide with the X-axis of the fragment drawing.

Connector

To create a connector, upon calling the command **FV: Construct Fixing Vector**, activate the automenu option:

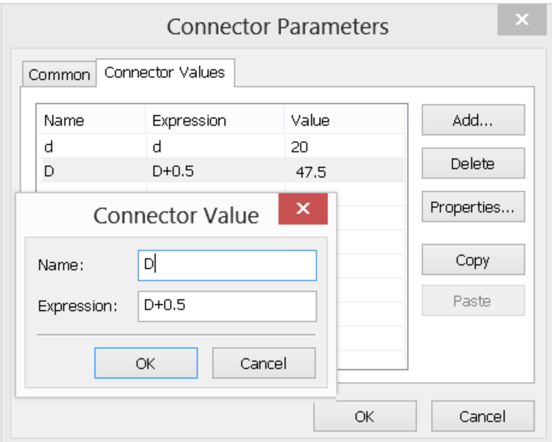
	<C>	Create Connector

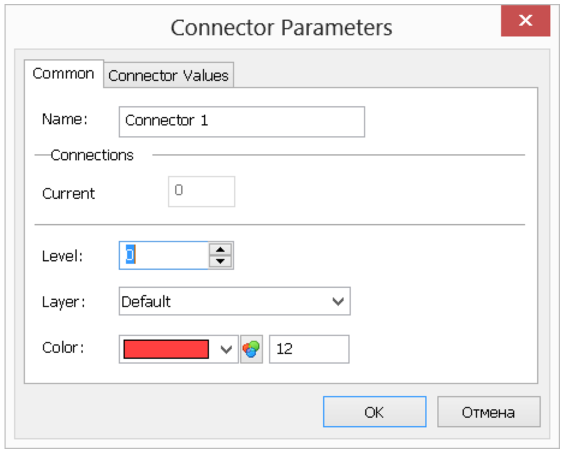
The system will then go into the mode of defining nodes. A connector can be defined by one or two nodes. If only one node needs to be specified, you can use the icon after selecting the first node.

If a connector is defined by one node, then its drawing image will be a point. This leaves the only possibility of geometrical attachment to it of the fragments with the fixing vector defined by one point (only the origin, no direction). For easier fixing by such a connector, we recommend establishing the references between such connector and several drawing elements (see below).

If a connector is defined by two nodes, then, upon snapping to it a fragment with a two-point fixing vector, the fragment will be automatically positioned by bringing together the two vectors.

Upon specifying the nodes, the connector parameters dialog box automatically appears for defining the variables and the common properties of the connector. The connector variables are defined on a separate tab of the dialog box.





Initially, a new connector does not contain any variables. At the time of creation, the user normally already knows the purpose of creating this connector and, in particular, what parameters of the connector fragments to control with it. If, for example, we create a connector on the axis of a shaft, we will be connecting to it the fragments of parts to be mounted on the shaft. Therefore, we need to pass the shaft diameter value to the fragment being connected (say, a ball bearing).

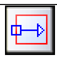
To create a new variable in the connector , press the button **[Add]**. In the coming up window, define the name of the connector variable and its expression. The name of the connector value is used subsequently upon attaching another fragment to the given connector. The name of the external variable of the fragment may not coincide with the value of the connector. At the moment of attaching the system uses a special parameter of the fragment external variable called «**Connector values**» (see the chapter «Variables» for details). If in the list of values of the external variable connector the name coinciding with the value stored in the connector, to which the fragment is being attached, will be found, then the value for the external variable will be automatically read from the connector. Link with the connector will be retained. Also, when attaching to the connector there is a possibility to manually link any fragment external variable directly with one of the connector values (see below "**Specifying values of fragment external variables**").

The number (or the text in quotation marks for a text variable), expression or the name of the current document variable can be put into the field "**Expression**". In our example, the value of the diameter, which is driven by a certain variable (for example, 'D') in the shaft model, has to be passed to other fragments. When creating the named value for the connector, it is possible to assign the name 'Diameter', and put the variable 'D' into the expression field. When creating the ball bearing model, in the properties for the variable driving the mounting diameter, it is necessary to specify the new value of the connector with the name 'Diameter'.

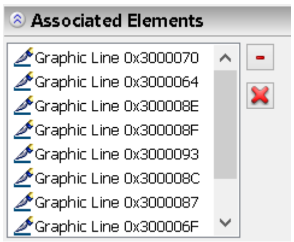
The tab "Common" allows defining basic properties of a connector as an auxiliary drawing object – name, level, layer and color. Besides, a special box in this tab tracks information about the number of connections to this connector (if the dialog of the connector properties is open directly in the assembly drawing).

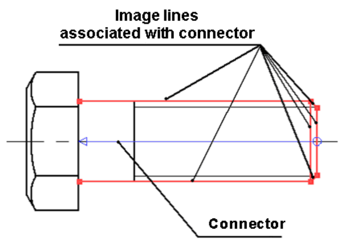
Defining associated elements for connector

To define associations of graphic lines with a connector, use the option in the automenu:

	<A>	Set Snap Elements

Upon selecting the option, the system enters the mode of selecting lines in the drawing that will activate the connector once pointed at by the mouse. To add elements to the connector, select those by the mouse in the drawing window. The selected elements are included into the list of associated elements in the properties window.





Insertion rules

So-called «insertion rules» can be additionally defined for a connector. These are additional transformations of translation and rotation with respect to the coordinate axes of the connector which the system will prompt a user to carry out when attaching a fragment to a given connector. For example, when attaching a nut to the bolt connector, in practice it is always required to specify additional translation of the nut along the axis of the bolt. That is why when designing the bolt model, in the insertion rules for the bolt connector the necessity of translational motion along the X-axis of the connector has to be specified.

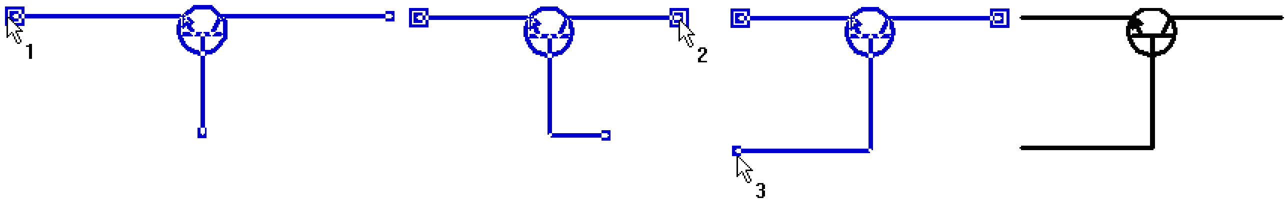
Fixing Points

A fixing point is created as an intersection of a vertical and horizontal lines, whose parameters are defined by variables with reserved names.

The variables reserved for the vertical lines are: x1, x2, x3, x4, x5, x6, x7, x8, x9.

Those for the horizontal lines are: y1, y2, y3, y4, y5, y6, y7, y8, y9.

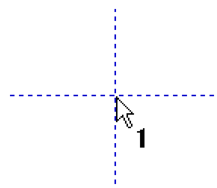
The numeric postfix of X and Y corresponds to the number of the fixing point and must match between the vertical and horizontal lines that define one fixing point. Up to nine fixing points are supported in a drawing.

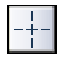
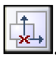
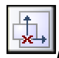


An example of fragment with three fixing points (a transistor)

When using the fragment fixing method based on fixing points, one has to determine the necessary fixing points before creating the drawing. All the rest of the drawing elements must be created relative to the lines defining those points. For example, if we want to use a circle as a fragment, that we could the later place in various locations in the drawings, we should first create a horizontal and the vertical line, and only after that the circle at their intersection point.

Fixing point creation



Fixing points can be created automatically, using the option  of the command **L: Construct Line**. Upon calling the option, move the pointer to the desired position and press <Ctrl+1> (<Ctrl+2>, <Ctrl+3>, ...) or use the automenu icon . When creating a point by option , a number will be attached to the pointer, indicating the order number of the fixing point being created.

As a result, two intersecting construction lines will appear in the screen. Besides, a pair of external variables will be created, x1 (x2, x3, ...) and y1 (y2, y3, ...). The parameters of the horizontal and the vertical construction lines will be given by these variables. Thus, a fixing point is produced in the drawing. When inserting this fragment in an assembly drawing, the system will ask you to specify the fixing point.

If necessary, a fixing point can be defined manually. To do this, just create manually two intersecting lines, a vertical and a horizontal one, and assign the reserved variable names to their parameters, x1 (x2, x3, ...) and y1 (y2, y3, ...), respectively. The variables must be marked as external in this case.

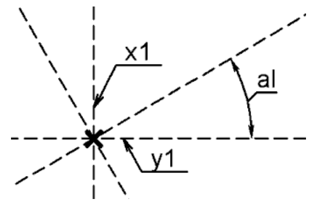
Throughout the following description, the expression **Create first (second, ...) fixing point** will imply performing all steps of a fixing point creation described above.

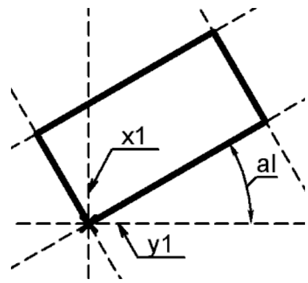
Let's review most common techniques of creating fixing points of fragments and local coordinate systems in a drawing.

Fragment with one fixing point without rotation provision

Create a fixing point. After that, create all construction lines relative to the vertical and horizontal construction lines that define the fixing point. In the course of the drawing creation, do not use the lines of types "vertical" and "horizontal"; rather, use the types "parallel" and "under angle". If you follow those rules, you'll get a drawing whose local coordinate system originates in the fixing point, and axes collinear with the X-axis and Y-axis of a drawing in which the current fragment is being inserted.

Fragment with one fixing point with rotation provision





Create a fixing point. Then, create a construction line passing through the node of the fixing point, at an angle to the horizontal line. Assign the slant angle of the line the variable *"al"*. When creating the variable, mark it as external and enter value other than 0 (entering "0" will make the lines coincide with the horizontal line on the drawing, complicating further constructions). The variable *"al"* will be an external variable of the drawing. Upon inserting the drawing as a fragment, the system will be asking for the value of the variable *"al"*.

The construction line passing through the node at an angle to the horizontal line defines the X-axis of the new local coordinate system in this drawing. To create the Y-axis, make a construction line passing through the node and orthogonal to the slanted construction line that is passing at the specified angle to the horizontal line.

The slanted line and the line perpendicular to it define a new local coordinate system in the drawing. All further constructions must be based on these lines. As a result, a parametric fragment will be built with one fixing point and the

variable *"al"*, that will be defining the slant angle of the fragment coordinate system with respect to the coordinates of the assembly drawing.

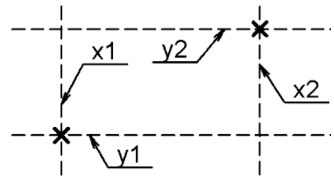
To make further constructions with respect to the new coordinate system easy, we recommend doing the following steps. Set the level value equal to "-1" for the vertical and horizontal lines passing through the fixing point. The lines will disappear from display as the construction lines visibility levels are defined by the range from 0 to 127 in the command **SH: Set Levels**, while these lines are assigned the level -1 and do not belong in the range. After that, call the command **ST: Set Document Parameters**, switch to the tab **View** and select the value **Visible only** in the parameter **Element selection**. By doing so, you set the drawing into the mode in which the elements will not be selectable that are not shown on the drawing.

Fragment with two fixing points with rotation provision

The described fragment can be created into different ways. In the first way, the second fixing point defines the fragment rotation and changes the size of the fragment. In the second the way, the second fixing point defines the fragment rotation only, not changing its size.

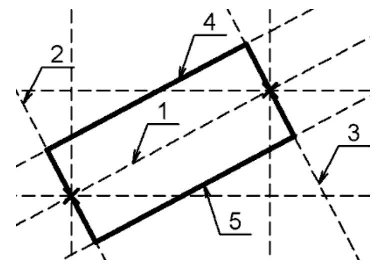
First approach:

Create two fixing points of the would-be fragment.

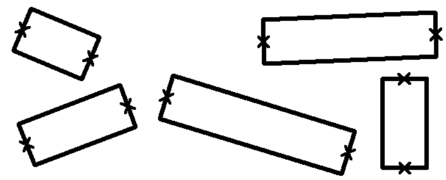


After that, you need to create construction line 1 passing through two nodes - the fixing points. This line will be defining the X-axis of the new local coordinate system in this drawing. To create the Y-axis of the local coordinate system, you need to make construction line 2 passing through the node and orthogonal to the line 1. The line passing through the two nodes and the line orthogonal to it define the new local coordinate system in the drawing.

All further constructions must be performed relative to the lines defining the new coordinate system in the drawing. Create line 3 parallel to line 2 and passing through the node defining the second fixing point. Next, create lines 4 and 5 parallel to line 1. After that, one can create the necessary graphic lines.

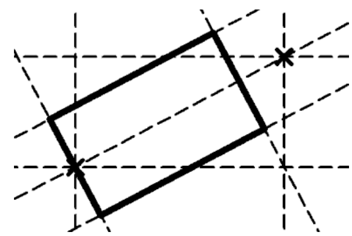


As a result, a parametric drawing with two fixing points is achieved, with the point positions defining the position of the fragment coordinate system with respect to the coordinate system of the assembly drawing. Besides, the second fixing point will be also determining the size of the rectangle.

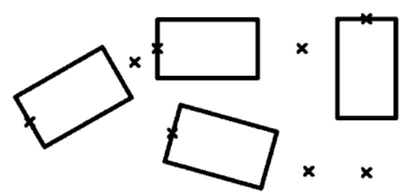


Second approach:

All constructions within the second approach repeat those in the first approach, except that line 3 does not snap to the node of the second fixing point, however staying parallel to line 2.



When inserting such a fragment into an assembly, its size will be constant, with the second fixing point defining only the rotation angle of the fragment.



> Since T-FLEX system always treats variables *x1*, *y1*, *x2*, *y2*, etc. as defining fixing points, these names may not be used for defining other drawing parameters.

INSERTING FRAGMENTS INTO A DRAWING

There are several ways of inserting a fragment into an assembly drawing:

1. By using the command **FR: Create Fragment**.
2. Within the **Library Explorer** or library window.
3. By creating a fragment directly in the assembly context.

The following description relates to the first two techniques of inserting a fragment into a drawing. The third technique is described in the dedicated section **Top-Down Design**".

Enter the command **FR:Create Fragment** for inserting a fragment into an assembly drawing:

Icon	Ribbon
	Draw → Insert → Fixing Vector
Keyboard	**Textual Menu**
<FR>	Draw > Fragment

The following options will appear in the automenu:

	<O>	Select File
	<C>	Create new Fragment in Assembly Context
	<G>	Extract Fragment Drawing
	<P>	Set Fragment parameters
	<R>	Repeat previous Fragment (accessible upon a repeated call to the command)
	<F>	Select Fragment to create its copy
	<F4>	Execute Edit Fragment command
	<Esc>	Exit command

The controls for customizing parameters of the fragment will appear in the properties window. They are put into several sections, their number and structure depend on the fragment creation approach. The sections of the properties window allow selecting the file of the fragment being inserted, a way of attaching the fragment, specifying external variables of the fragment, etc.

> Before selecting the fragment file the dialog in the properties window contains only three sections: **Basic parameters**, **Fragment parameters**, **Options**. The values of the parameters, defined in the properties window before file selection, are automatically stored as default values for all subsequently inserted fragments.

The first step when inserting the fragment is the selection of the file of a fragment. The dialog window for selecting a file can be called with the help of the option or by pressing the button in the section **Basic parameters** of the command's properties window. In the window for file selection it is necessary to indicate the document file of the fragment. After selecting the file, a dynamic image of the inserted fragment attached to the cursor will appear in the drawing window. The contents of the automenu will be changed. The new state of the automenu will depend on the fragment attachment approach specified in the properties window: by fixing points or by fixing vector.

The fragment attachment approach is specified in the section **Insert Fragment**, with the help of the parameter **Fixing**. In case the fragment is attached by the fixing vector, the required fixing vector can be chosen from the drop down list of this parameter. The required page of the fragment document can be selected here as well (in case of multi-page document).

In the section **Fragment parameters** of the command dialog it is possible to specify various settings for fragment insertion. A detailed description of these parameters is presented in the section “Fragment parameters”.

The values of the fragment external variables are specified in the section **Variables** of the properties window. This section is present in the dialog of the command's properties window if the fragment being inserted has external variables.

In the section **Preview** of the command dialog the inserted fragment is dynamically displayed in accordance with the specified values of the external variables, fixing elements (fixing points and fixing vectors) are indicated.

When defining fragment insertion parameters, you need to specify the placement points of the fixing vector or fixing point in the drawing area (see the section **Defining Fragment Placement in the Assembly Drawing**). The coordinates of the specified points are shown in the section **Coordinates** of the command's properties window.

After attaching the fragment on the current drawing, it is necessary to finish creating the fragment by pressing in the automenu or in the command's properties window. Before that a series of additional operations can be performed with the help of the options appearing in the automenu of the command at this stage of the fragment insertion:

	<K>	Input Fragment insertion points
	<P>	Set Fragment Parameters
	<Y>	Create Name for selected Element
	<Ctrl+O>	Open Part
	<O>	Open Fragment in Context of Assembly
	<D>	Automatic Explode

	<S>	Explode with Construction
	<F4>	Open Fragment file for editing
	<C>	Select Clipping Hatch
	<I>	Select Other Element
	<Esc>	Deselect all elements

The option starts the mode of respecifying the fixing point of the fragment.

The option allows assigning the name for the inserted fragment. The fragment name can be used, for example, for search, for automatic creation of nodes of the fragment in the assembly, for obtaining the values of the fragment variables in the assembly drawing with the help of the function get:

get («Fragment name»,« Variable name»),

where «Fragment name» – the name that was specified for the fragment, and «Variable name» – the name of the variable from the fragment drawing the value of which has to be obtained in the assembly drawing.

To explode the fragment automatically the options and are used (see the section **Exploding Fragments** in the chapter "Basic Fundamentals and Concepts of Working with Assemblies"). The options and allow opening the file of the inserted fragment straightway for editing, after its creation has been finished automatically.

If necessary, for example, for fast insertion of several fragments in a row, it is possible, by setting on the flag **Auto Create** in the section **Options** of the command's properties window, to turn on the mode of finishing fragment insertion automatically after specifying its fixing points. In this case, the fragment insertion will be finished automatically right away after specifying all fragment fixing points or its fixing vector.

> Detailed description of all steps of inserting a fragment (selecting the fragment file, defining external variable values and placement, attachment of the fragment to the assembly drawing) can be found in the respective topics of this section.

If you need to insert another instance of one of already inserted fragments, you can conveniently use the options and (described below, in the topic "Repetitive fragment insertion").

Selecting Fragment File

For specifying the fragment file the standard window for file selection is used. It appears after using the option:

	<O>	Select File

Both your directory structure and the installed T-FLEX CAD libraries can be the source for this selection. Switching between these two sources is carried out with the help of drop down list found at the upper left corner of the file selection window.

When you are in the fragment insertion command, it is also possible to indicate the library element in the service window "Library Explorer".

Selecting Fragment from Library

If a library element is used as a fragment, it can be assembled directly in the "Library Explorer" window or in the auxiliary library window (see the chapter **Libraries**), bypassing a direct call to the command **FR: Create Fragment**. In this approach, you can use the context menu commands or the Drag&Drop mechanism. To use the latter, select a fragment in the "Library Explorer" or library window by pressing on the fragment and drag it into the drawing area, while holding the left mouse button down. This will automatically activate the command **FR: Create Fragment**. You can get the same result by right-clicking over a fragment in the "Library Explorer" or library window, and then selecting the context menu item **Insert**.

Defining Fragment External Variables

If external variables exist in a drawing, you will be asked to define the values when inserting this drawing as a fragment into assemblies. When attaching to the connector, necessary values can be specified automatically.

Let's review defining fragment external variables in the example of a ball bearing drawing that we will be inserting into a shaft drawing. An external variable "d" is created in the ball bearing drawing that drives its inner diameter.

After choosing the name of the fragment-bearing, in the command's properties window the tab "Variables" will appear which allows specifying the values for the fragment external variables. The standard list of variables or user-defined dialog with the control elements can be used for specifying the values (see the chapter "Control Elements. Creating User Defined Dialog Boxes").

> If the fragment drawing has text fields with an external variable inserted into them (for example, title block), then the values of such variables can be changed directly on the assembly drawing.

Let's consider first defining the fragment external variables using the list of variables. In this way, variables may be assigned data of three types:

1) Constant (a number in the case of a real variable, or a string in the case of a text variable);

2) An assembly variable;

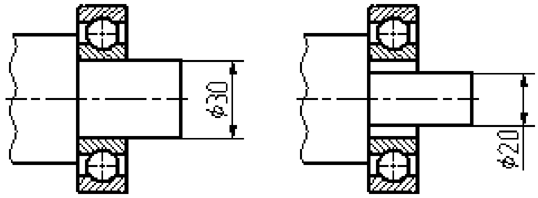
3) None (the variable entry left blank).

Type 1: Assigning constant value

Let's set a particular value of the ball bearing diameter, for example, *"30"*.

Comment	Name	Value
Inner diameter	d	30

In this case, as the shaft diameter varies, the inner diameter of the ball bearing will not change. To modify the value of the ball bearing external variable, you will have to edit the fragment.



> Fragment external variables can be pre-defined with default values when offered for defining. The default values are copied from the respective variables in the original fragment drawing file, if the flag **Substitute Variable values** is set in the command **SO: Set System Options** on the tab **Fragments**.

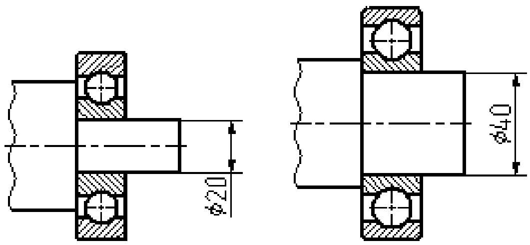
Type 2: Assigning assembly variable

Instead of a fixed value of the ball bearing external variable, let's type in the name **Diameter** of an existing assembly variable. If a user puts in the name of a non-existing variable, the system will prompt a user to create a new variable.

Comment	Name	Value
Inner diameter	d	Diameter

Suppose, at the insertion time, the variable **Diameter** equals **20**. Let's modify the variable **Diameter** to *"40"*. In this case, the ball bearing image will adjust automatically.

An assembly variable can be selected from the list of existing variables by pressing the key <F8> (see the chapter "Main Concepts of System Operation", the topic "Context menu for dialog input boxes").



> If a fragment external variable was named after an assembly variable, and such assembly variable actually exists in the current assembly drawing, the latter value will be automatically assigned to the fragment variable (see the chapter **Variables**).

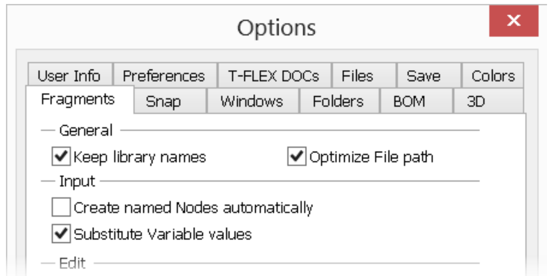
Type 3: Undefined value

You can skip defining the external variable altogether, leaving the entry blank.

Comment	Name	Value
Inner diameter	d	

In this case the value read from the fragment file will be put into. To change now the ball bearing diameter, it is necessary to download its file as a separate drawing, specify the desired diameter value in that file, and after saving the file return to the assembly drawing. This scheme is used in practice rarely, when it is required to modify the assembly drawing by changing the values of the variables in the fragment file.

For filling in unspecified values of the variables automatically, in the System options a special parameter **Substitute variable values** is turned on by default. In this case, upon inserting the fragment, all variables automatically get constant value read from the fragment.



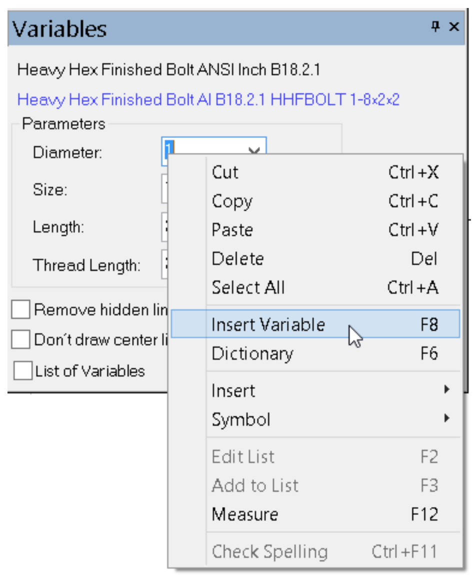
Working with user-defined dialog

User-defined dialog with control elements represents itself a more visual tool for working with external variables of the fragment. The set of different control elements allows specifying the values for the variables with the help of various toggles, buttons and also by entering the values from the keyboard or working with the list of values. There are two regimes in which the dialog fields for the value entry can operate. Either the value of the variables or the expressions themselves can be displayed in the fields of the dialog. In the latter case, the values of the expressions are shown to the right of the entry field. The second regime works in case the parameter **Show expressions** has been set on in the properties of the entry field. Details of creating user-defined dialogs are described in the chapter Control Elements. Creating User Defined Dialog Boxes.

In any case, a user can write both values and expressions into the entry fields. After finishing the data entry (when the focus of input is shifted to another field), the fragment external variable which is related to the given entry field will be set equal to the expression.

To define a relation between a fragment external variable and an assembly variable, you can use the context menu commands.

You can call the context menu command “Insert Variable” or, instead, simply press <F8>. The standard “Insert Variable” dialog box will be displayed in the screen. Upon selecting an assembly variable, its value will be assigned to the fragment external variable. A checkmark before the respective command of the context menu will be indicating that the current fragment variable has a relation with an assembly variable. Direct editing of the dialog entries will be modifying the respective assembly variable values.



To cancel such a relation, call the same context menu command again. When the connection with the variable is established, it is possible to continue working with the drop-down lists of the values. While doing it, the assembly variable connected to the given fragment variable will be assigned the new value from the drop down list.

If necessary, it is possible to switch to the mode of working with the list of variables instead of working with the user-defined dialog. For switching to another regime, use the flag **List of Variables** found at the bottom of the dialog.

For loading one of the saved configurations (predefined set of values for the fragment external variables), use the drop-down list **Configuration:** at the bottom of the user-defined dialog.

The command for working with the configuration can be invoked by the following means:

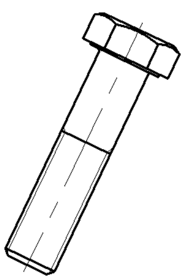
Icon	Ribbon
	Parameters → Tools → Configurations and variations
Keyboard	Textual Menu
<FCE>	Parameters > Model Configurations and variations

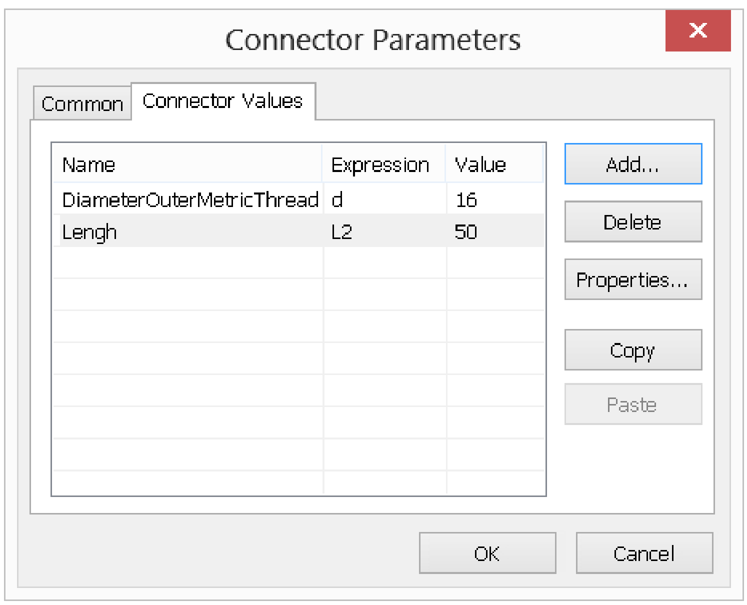
Details of the work with the command **Model Configurations and variations** can be found in the chapter “Auxiliary Tools for 3D Assemblies Modeling” of the book “Three-Dimensional Modeling”.

Working with the connector

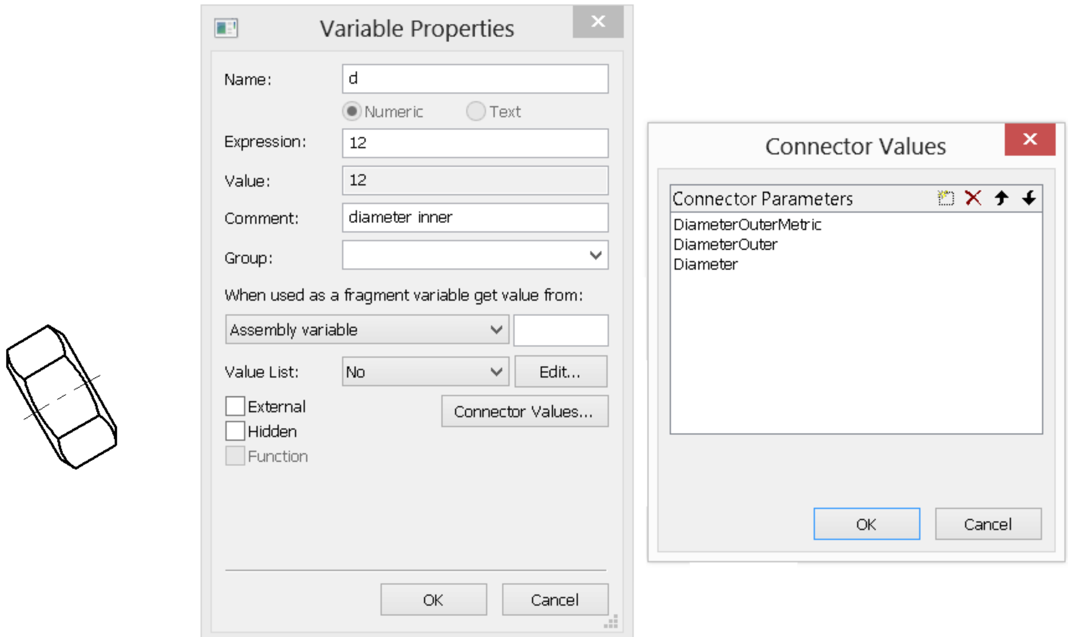
Upon attaching the fragment (for example, a nut) to the connector of another fragment (for example, a bolt) the values of the variables can be obtained automatically. For successful automatic connection of the variables with the connector, the following conditions must be met:

1. In the connector the named values have to be specified. These named values will subsequently serve as a link between the variables of the fragment having a connector (bolt) and the variables of another fragment (nut) which will be attached to the given connector in the assembly.



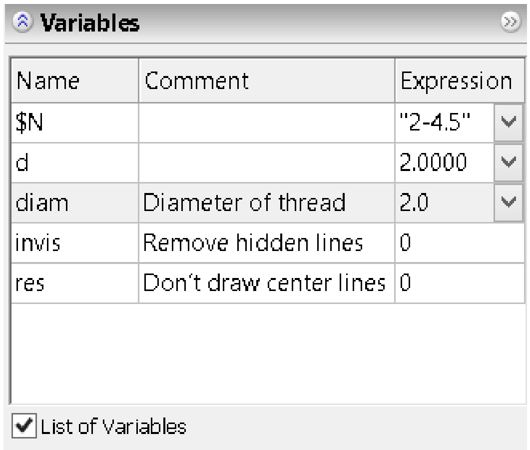


2. For corresponding variable of the fragment the list of names for the values or at least one name of such sort must be specified for linking with the connector. The name of the value for the variable has to be specified in advance in the variables editor in the fragment file by modifying the properties of the variable. The name of the connector value specified for the variable must coincide with the name of the value specified in the connector. Only in this case the automatic connection is possible.



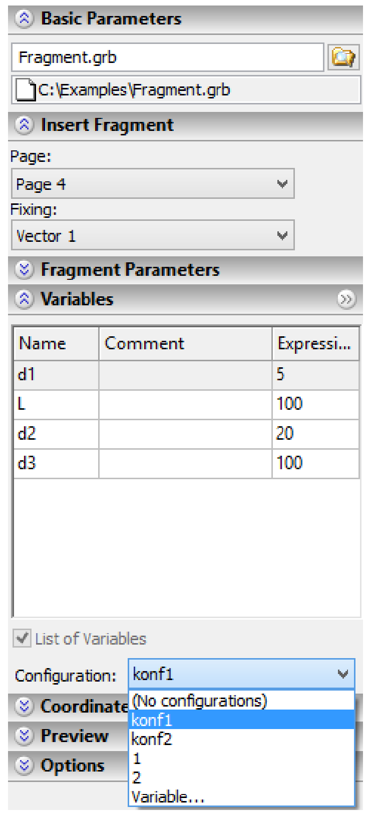
3. Upon inserting the fragment, the connector must be chosen. The variables of the new fragment automatically take the values straightway after the connector selection. The image of the fragment takes the form in accordance with the values of the variables.

The connection between the fragment variables and connector values can be determined manually. To do it, resort to the list of the fragment variables in the command of creating and editing the fragment. If the user-defined dialog is used for controlling the variables, switch to the list of the variables. The manual setup of the connection between the variable and the connector value is needed in case the fragment is attached to the connector but for its variable the connector value name is not designed or that name does not coincide with the value in the connector. In this case, clicking in the field **Connector Values...** will show the list of the connector values and upon selecting one of the values, the connection will be established. The field will be marked by a special symbol.



Using configuration

For specifying the values of the fragment's external variables, it is also possible to use configurations created in the document of the fragment.

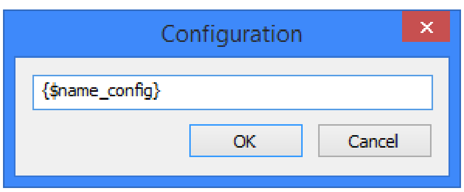


When configurations exist in the fragment's document, the parameter "Configuration" is available in the fragment's parameters dialog. Its drop-down list contains the names of the configurations created in the fragment's document. When selecting a configuration, all fragment's external variables are automatically assigned the values stored in the selected configuration.

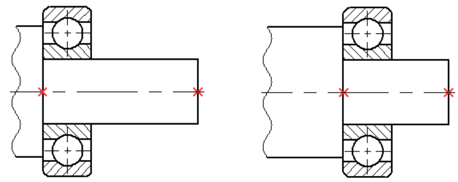
In addition to the names of fragment's configurations, the list of the parameter "Configuration" contains additional items:

No configurations – this option is used to prevent the use of configuration;

Variable… – allows specifying the name of the used configuration with help of the variable. After selecting this item in the list, the window for specifying the variable's name will pop up on the screen. The variable's name can be specified either manually (in curly brackets) or with the help of the context menu command "Insert Variable… F8".



Defining Fragment Placement in the Assembly Drawing

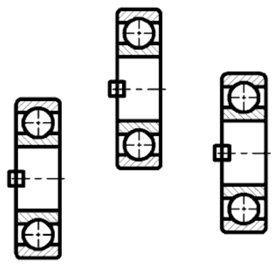


When specifying location of the fixing vector or fixing points, the nodes of the assembly drawing can be picked for carrying out fragment attachment to the elements of the assembly drawing. When changing location of the specified nodes of the assembly drawing, the fragment placement will be modified as well.

If necessary, the fragment can be inserted by specifying the absolute coordinates for the fixing vector and points. This becomes possible upon clicking at the desired place of the drawing in the absence of object snap. The object snap can be turned off via the toolbar **View** or by pressing the key <Ctrl> at the time of the click. In this case, with the help of the section **Coordinates** in the fragment properties window the adjustment of the coordinates of the fragment insertion can be performed.

Fragment placement using fixing vectors

When inserting a fragment by the fixing vector the system asks to define the vector fixing points.



If the fixing vector was defined in the fragment drawing by a single point, then you need to specify only one point in the assembly drawing to place the fragment. Once that is done, the image of the fragment will be placed in this point. The fragment image cannot be rotated in this case.

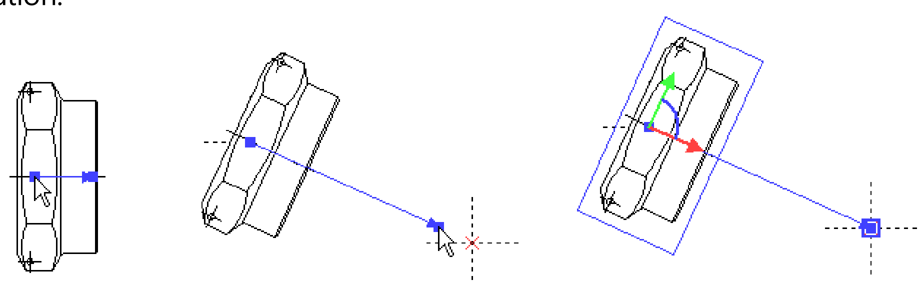
If the fixing vector was defined by two points in the fragment drawing, then the fragment can be placed in several ways.

The first way implies specifying two points, the first defining the location of the fragment image, and the second - rotation of the fragment image about the first point.

The option for handling this way appears upon specifying the first point:

	<M>	Fixing using two points

When proceeding in this way and fixing the vector start to one node and the vector end to another, modifying the position of the first node will change the location of the fragment image, while the second one - the orientation.

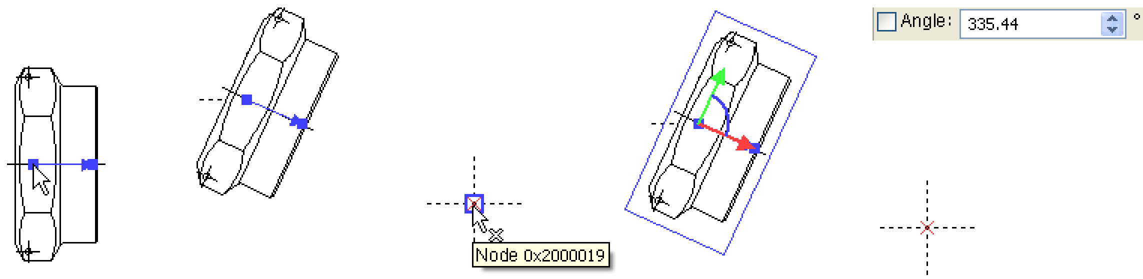


The second approach differs from the first one as follows. The first point still defines the location of the fragment image, while the second point, instead of maintaining the rotation, sets just once the rotation angle of the fragment image with respect to the X-axis of the assembly drawing. The angle of rotation for the fixing vector can be specified as a variable.

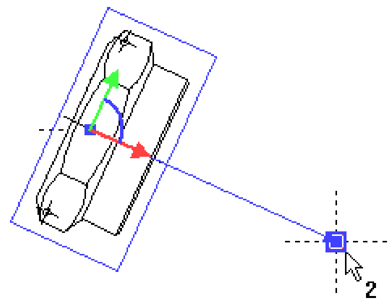
This way is handled by the option that appears upon defining the first point:

	<M>	Fixing using one point and angle

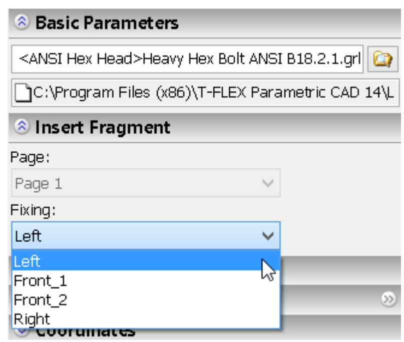
With this type of placement, modifications in the position of the first placement node will cause change in the fragment location only, not affecting the rotation angle of the fragment image.



Before confirming the fragment insertion or in the fragment editing command, already inserted points of the fixing vector can be easily changed. For modifying location of the desired point, bring a cursor to this point. The cursor will take the form corresponding to the point number (1 or 2). Next by pressing 🖱 select the point, and the system will be ready for modifying the point location. Choose the new position and press 🖱.



The fragment being inserted can have several fixing vectors. For selecting the desired vector, the section **Insert Fragment** in the properties window should be used. In the drop down list «Page» it is possible to select the desired page of the fragment document (if the fragment contains several pages), and then for the selected page the fixing vector is picked in the drop down list “Fixing”.



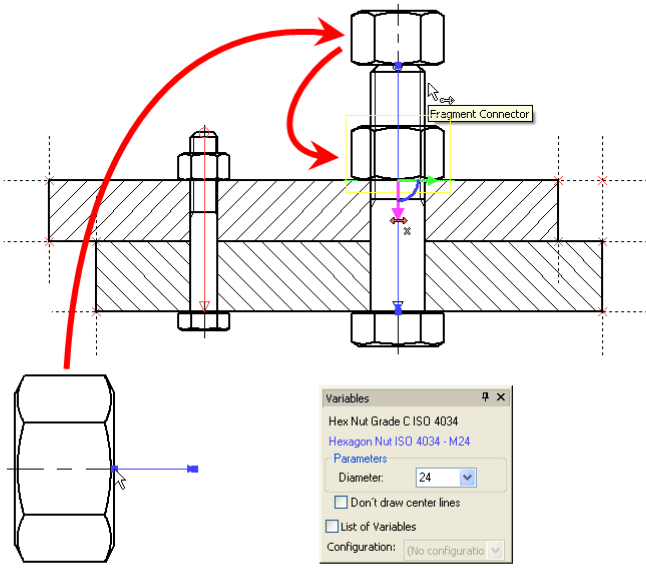
The section “Preview” of the properties window can be also used for changing the fixing vector. Here, on the preliminary image of the fragment, the active fixing vector is highlighted with a red color, the rest are blue. The fixing vector can be selected by clicking in the preview window.

Attaching to connector

When inserting a fragment with a fixing vector, turn on the mode of snapping to connectors. This mode is activated by the automenu option:

	<C>	Mode of snapping to Connectors

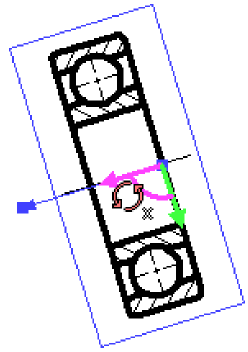
The option appears in the automenu immediately after selecting a fragment file, if the system determines that the current fragment is placed using a fixing vector. In this mode, as the pointer approaches a connector or a graphic line referenced by the connector, the variables of the fragment being inserted automatically assume the values from the same-name connector variables. This reflects on the changing shape of the fragment. At the same time, the updated fragment automatically snaps to the connector.



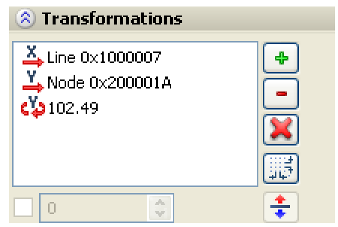
When attaching to the connector the system may require specifying **additional transformations** (see below).

If you need to define the values of other variables, this can be done in a conventional way.

Additional transformations



When attaching the fragment with the help of the fixing vector, sometimes it is required to make the fragment location more precise – move or turn the fragment with respect to the axes of the fixing vector, i.e. define additional transformations. Transition to specifying additional transformation can be carried out straightway after specifying the points of the fixing vector. While doing it, a special dragger in the form of the coordinate system will appear on the fragment image. If the cursor is brought to the elements of the dragger (the coordinate axes and the arc between the axes), it will be taking the form in accordance with the offered transformation – , rotation or , translation along one of the axes. When specifying the rotation, one of the dragger axes is also selected (namely the nearest one) for possibility of defining exact direction to the selected point on the drawing.



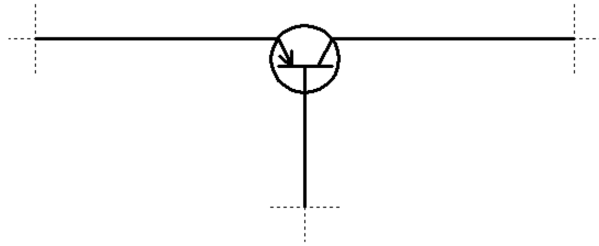
If at this moment is pressed, the fragment will start moving following the cursor. For fixing the current transformation, it is necessary to select the object on the drawing or specify the numeric value of the transformation. Selection of the object of the drawing will create its associative link with the transformation. This means that upon changing the object location, the transformation will be automatically adjusted as well. The numeric value for the transformation is specified in the properties window or by clicking at the free space of the drawing without selecting the object.

It is possible to define an unlimited number of transformations successively. All specified transformations are put into the properties window. The transformations of similar type (for example, several translations along the same axis) are automatically summed up.

When attaching the fragment to the connectors, the requirement of defining additional transformations can be brought into the fragment file already at the connector creation stage. In this case right away after the connector selection (when attaching another fragment) and reading the values of the variables, the system will automatically turn to specifying the required additional transformation. This approach is frequently used in the libraries of the standard elements since it is not always possible to automatically determine the location of the future fragment. For example, when putting a nut on a bolt, it is almost always necessary to place this nut on the surface being fastened, the attachment to which is specified by a user while defining additional transformations.

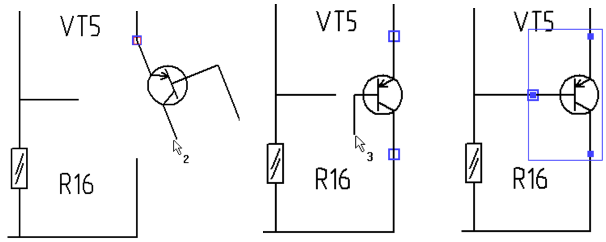
Fragment placement using fixing points

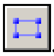
If the fragment drawing contains external variables *"x1"*, *"y1"*, *"x2"*, *"y2"*, etc., then upon inserting the fragment into the assembly drawing you will have to specify as many placement points as you have pairs *"x"* and *"y"* with the respective numbers in the fragment drawing.



For example, suppose we want to locate the image of the transistor, the drawing of which has three fixing points. Each point specifies location of one transistor contact. When inserting the fragment on the assembly drawing, the system will successively prompt a user to specify location of each point.

When specifying the fragment fixing points, the current fixing point number will be drawn next to the cursor.



The dynamic image following the cursor helps to evaluate the appearance of the future fragment. If necessary (in case of complicated for dynamic imaging fragments) the dynamic image can be switched off with the help of the option .

There are two possibilities for specifying locations of the fixing points: attach the fixing point in absolute coordinates (independent of the assembly drawing) or snap to the node on the assembly drawing. Attachment in absolute coordinates is carried out by pressing  while the object snaps are turned off or by pressing simultaneously the key <Ctrl>. Snapping to a node is done via  with simultaneous node selection or with the help of the option <N>.

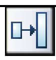
Repetitive Fragment Insertion

Several options are provided in the fragment creation command **FR: Create Fragment** for repetitive insertion or duplicating fragments. Use of these options speeds up insertion of identical fragments.

Multiple insertions of the last created fragment are supported by the option:

	<R>	Repeat previous Fragment

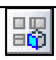
Selection of a fragment to duplicate in the drawing is done by the option:

	<F>	Select Fragment to create its copy

Upon calling one of those options, the system asks you to specify just the placement of the fragment in the drawing. In this case, the external variables will have the same values as the fragment being duplicated.

Adding Projections and Schematic Drawings of Models to Assembly Drawing

This option allows us to add projections or schematic drawings of models onto a plane.

	<W>	Project 3D fragments onto a workplane

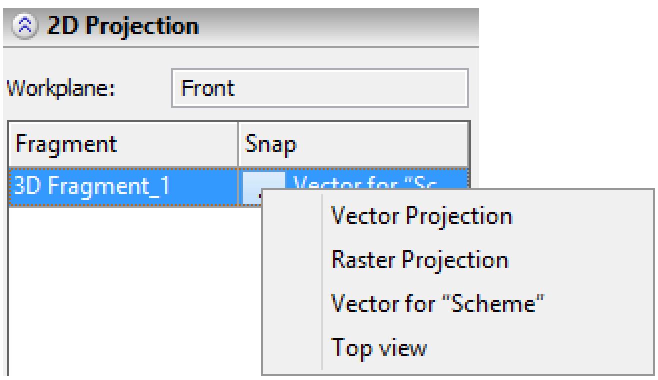
After the command is activated, the automenu appears.

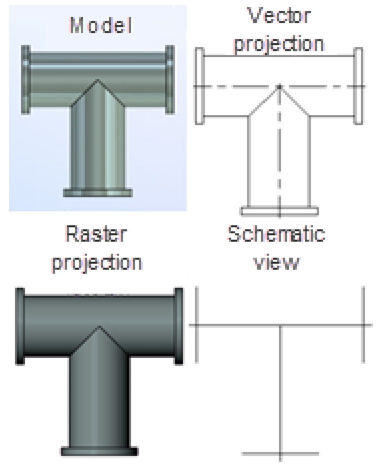
	<W>	Select workplane for specifying projection direction
	<F>	Select 3D fragment
	<A>	Select all 3D fragments in scene
	<Esc>	Exit from the command

Using the command:

1. Select a plane onto which the model will be projected . The selected plane will be displayed in the operation's properties in the «Workplane» column;
2. Select one or several models for projection or ;
3. Select projection type or schematic view.

The source for the 2D fragment being created can be selected in the list in the «Fixing» field:





- Vector projection – projection of the selected model is created on the basis of the lines of the model' image;
- Raster projection – projection of the selected model is created as a 2D fragment;
- Fixing vector – schematic image created earlier is added to the plane. Only those vectors are added to the list which satisfy the workplanes compatibility condition.

The use of this command is described in more detail in the «3D Assemblies – Creation of assembly 3D models» subsection of the «Using 2D-fragment as «projection» for 3D fragment» section.

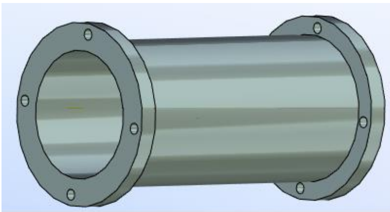
Using 2D-fragment as «projection» for 3D fragment

This method allows us to create, on the drawing, instead of projection of a 3D fragment, a 2D image prepared in advance in the fragment's file (the term "the schematic view" is used below).

This can be a simplified image or a drawing of the workpiece. In most cases the assembly's drawing created in this way will be recalculated faster than the drawing obtained by projection of the assembly.

This method can be conveniently used when creating layouts of industrial shops, creating electric circuits and in other cases that require schematic display of 3D elements on the drawings. Furthermore, schematic view can contain conditional graphical views which can not be foreseen on projections (for example, direction of door opening).

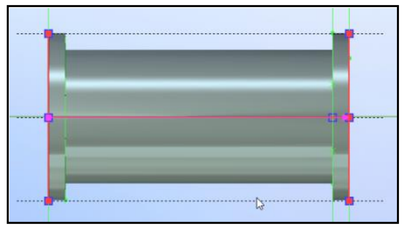
Preparing simplified image in fragment's file



First we open or create a 3D model of the fragment. In this example we consider an element of a pipeline.

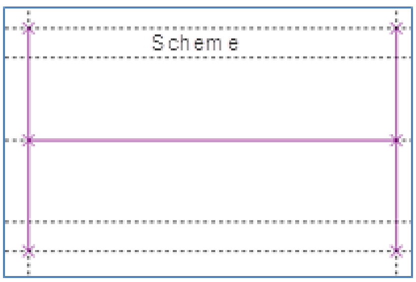
On the «Top view» workplane we create schematic view of the given element.

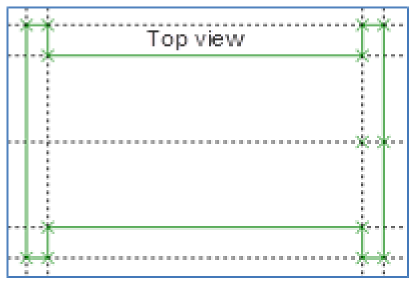
It is important that the overall dimensions of the drawing and 3D model be the same. If the model is parametric, the image being created must also be reconstructed when changing parameters. For example, if the length of the pipe on the drawing will be smaller than that in a 3D fragment then the fragment will not be connected with other elements of the pipeline upon the assembly.



It is possible to create several different drawings of the workpiece on the same or different workplanes and then select any drawing among them upon creation of the assembly's drawing.

Let us create one more drawing. Its construction lines must be built independently of the graphic lines of the previous drawing in order to translate drawings independently of each other. Let us call the resulting drawings as «Scheme» and «Top view».

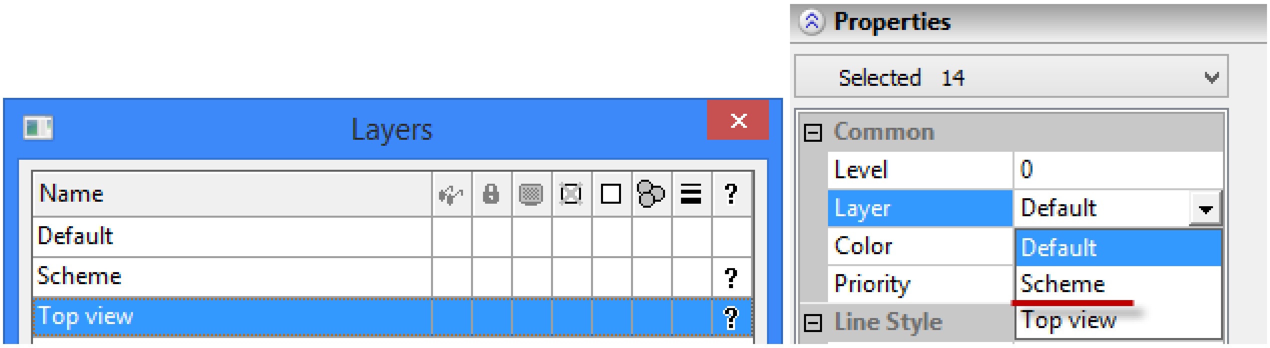




Graphic lines of each drawing must be located on different layers for them to be added independently of each other upon creation of the assembly's drawing even if they are located on the same workplane. With the help of the command

Icon	Ribbon
	Edit → Document → Layers
Keyboard	Textual Menu
<QL>	Customize > Layers

We create two new layers. For convenience, we will call these layers «Scheme» and «Top view».

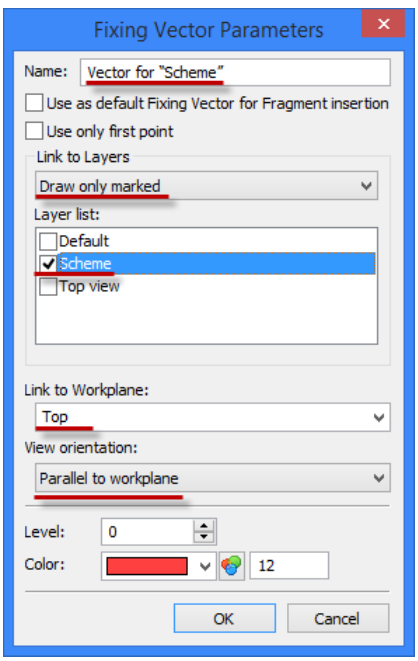


Next we select the graphic lines of each drawing and choose one of the created earlier layers in the properties windows. Now selected graphic lines will be associated only with that layer.

At the next step we create the fixing vector with the help of the command:

Icon	Ribbon
	Draw → Insert → Fixing Vector
Keyboard	Textual Menu
<FV>	Construct > Fixing Vector

Now we indicate the start and end point of the vector on the drawing. The fixing vector is used for creating connection between the 3D model and the drawing.

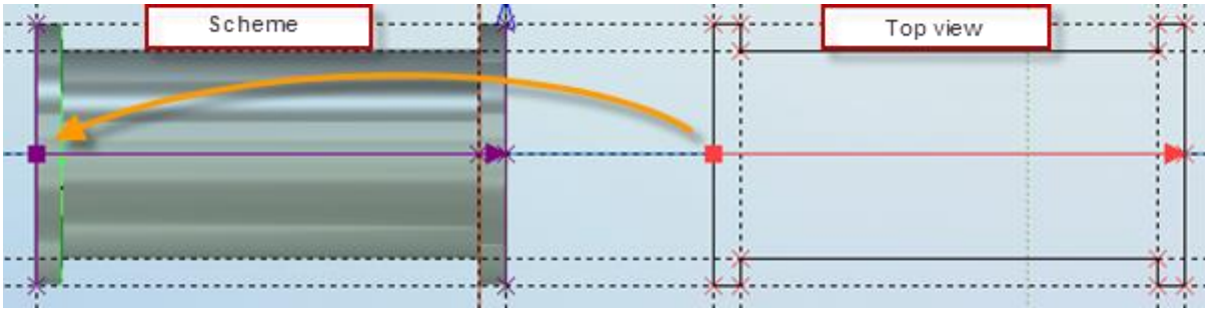


In the «Fixing vector parameters» window that appears we specify the name of the vector «Vector for "Scheme"». In the «Connection with layers» field we select the «Draw only marked items» option, so that upon rendering only the drawing located on the selected layer is displayed as a fragment. From the list of layers we select the «Scheme» layer.

In the «Connection with workplane» field we select the plane with which the fixing vector will be linked. In the «View orientation» field we select the option of axial symmetry. The same sequence of actions should be repeated for the second drawing, by indicating another layer.

Creation of fixing vectors is described in more detail in the «Ways of attaching fragments» section of the «2D–fragments – Bottom-Up Design» chapter.

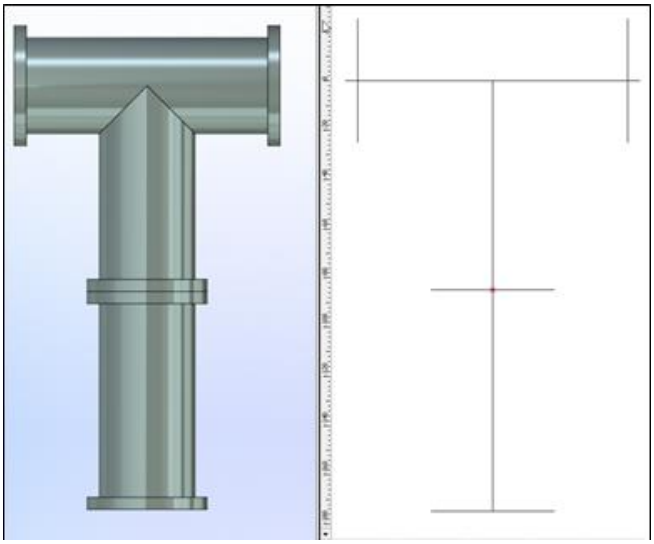
As a result we have two schemes for insertion into the assembly's drawing. Projective connection must exist between the model and the scheme, i.e. schematic image must be located at the same place where the projection of the model would be located. In our example the schemes are created on the same page, that's why before adding the model to the assembly the schemes must be combined (place the «Top view» drawing above the «Scheme» drawing).



Save the file of the fragment.

Creating schematic image of 3D fragment in the assembly

Open the assembly's file. It contains two connected elements of the pipeline. The elements of the schematic view were created for them in advance.

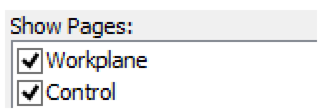


Go to the page of the «Top view» workplane.

For the pages of the workplanes to be displayed in 2D window, the command must be invoked

Icon	Ribbon
	Edit → Document → Pages
Keyboard	Textual Menu
<PG>	Customize > Pages

This command can be invoked only when 2D window is active. In the window that appears enable the «Workplane» flag. Now at the bottom of 2D window the tabs of the workplanes will be displayed.



Invoke the command

Icon	Ribbon
	Assembly → Assembly → 3D Fragment
Keyboard	Textual Menu
<3F>	Operation > Insert 3D Fragment

After the 3D fragment has been inserted into the assembly we invoke the command

Icon	Ribbon
	Draw → Insert → Fixing Vector
Keyboard	Textual Menu
<FR>	Draw > Fragment

Now we activate the automenu option

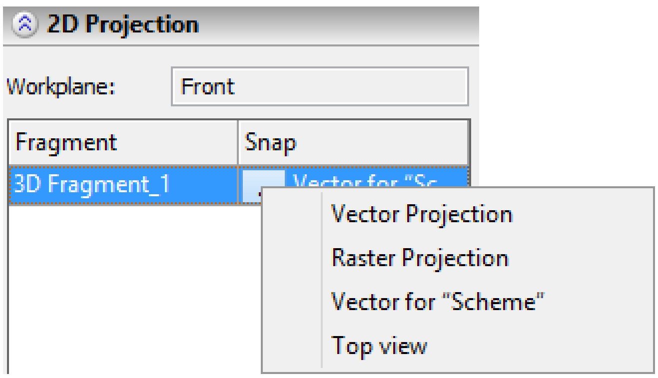
	<W>	Project 3D fragments onto a workplane

We select the plane on which will be created the image and the body, for which this image will be created. There is a capability of selecting simultaneously several bodies manually or all bodies in the scene with the help of the option

	<A>	Select all 3D fragments in the scene

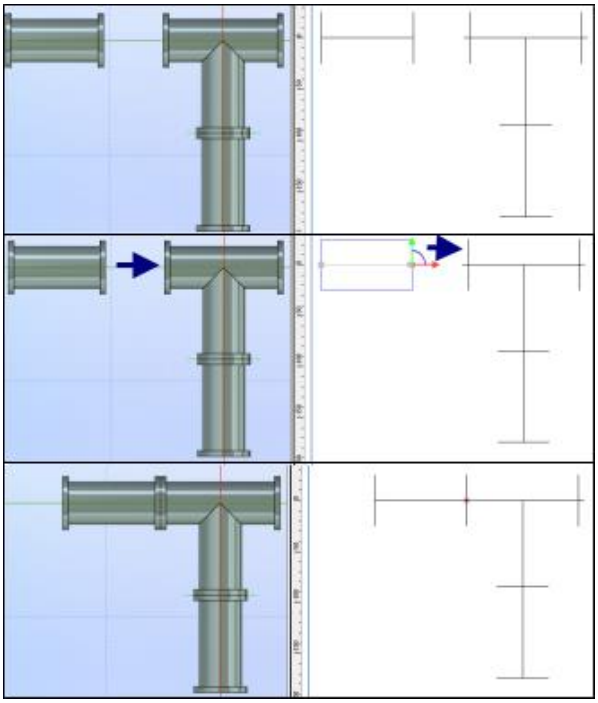
For a current 3D fragment, the source for the 2D fragment being created can be selected from the list of the «Fixing» field:

- Vector projection – projection of the selected model based on its graphic lines is created;
- Raster projection – projection of the selected model as a 2D fragment is created;
- Fixing vector – schematic view created earlier is added to the plane. Only those vectors that satisfy the workplanes compatibility condition are added to the list.



We select «Vector for "Scheme"» and press «Finish input». On the «Top view» plane the new schematic image has been created.

3D fragment and schematic image will be synchronically moving in space. For example, when translating a 2D-fragment, the 3D fragment will also change its location. At the same time some degrees of freedom of a dragger can be automatically suppressed upon this translation (become grey), in order to prevent the break-up of connection with the schematic views on the other planes.

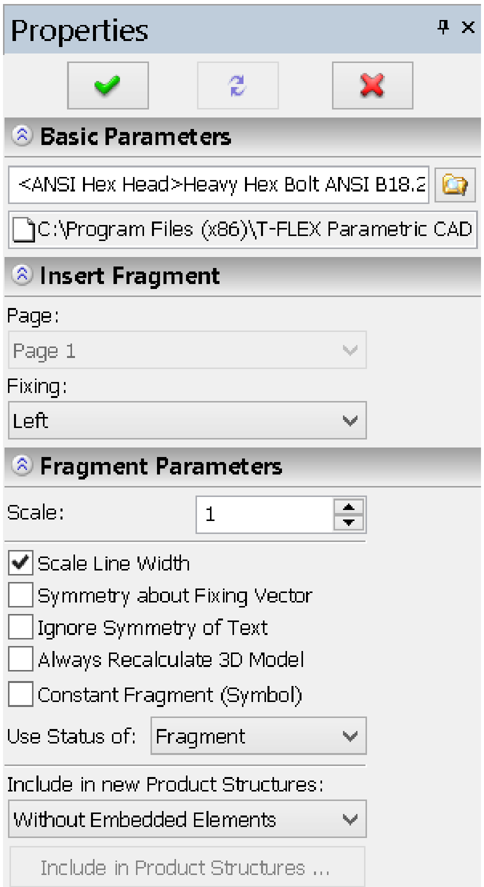


Fragment Parameters

The section **Basic Parameters** contains data about the fragment file. The button  can be used to open a window for selection of the fragment file. In the field available for editing, the text with the reference to the fragment file is written down. The reference can be full or relative. The second line shows the full path on the hard disk through which the written reference is functioning. The relative reference can be written manually or automatically in the following form:

- `..\..\folder\file_name.grb` – in this case, the record is made relative to the file of the assembly drawing. The symbol ..\ denotes going along the folders by one level up with respect the assembly file.
- `<library>file_name.grb` – in this case, the reference uses the drawings library connected to T-FLEX CAD.

It is recommended to use the relative references which operate in a more flexible way and are not tied to a specific place on the hard disk. It allows a user, for example, to move the assembly with all documents contained in it from one place to another without any losses.



The section **Insert Fragment** allows a user to select the desired page of the document being inserted as a fragment (parameter **Page**). This parameter is not available if there is only one page in the fragment. Also, in the drop down list **Fixing** found in this section, the attachment approach is specified: either by fixing points or by fixing vector.

The section **Fragment Parameters** contains different parameters of the fragment insertion:

Scale. Defines the scale of the fragment being inserted. This is used only when inserting fragments by fixing vectors.

Rotation Angle. Defines the angle of the fragment rotation. This is used only when inserting fragments by fixing vectors.

Scale line thickness. This parameter specifies whether to apply the assigned scale to the fragment lines thickness. This is used only when inserting fragments by fixing vectors.

Symmetry About Fixing Vector. When set, the fragment image will be mirrored about the fixing vector. This is used only when inserting fragments by fixing vectors.

Always regenerate 3D model. With this flag set, the 3D model of the fragment will be automatically regenerated upon changes to the 2D fragment parameters.

Constant Fragment (Symbol). Setting this parameter makes the inserted fragment saved in the assembly document as a picture. This allows speeding up the work with the assembly since now there is no necessity to resort to external files. The fragment file is not read when opening the assembly. Owing to that, the fragment can be displayed in the assembly even in case its source file is absent. However, since this is a picture, other elements of the drawing cannot be fixed to it. Also, it is not possible to modify variables of such fragment from the variables editor of the assembly drawing (when the connection between the assembly variable and the fragment variable exists). Modifying variables is possible only upon editing such fragment on condition that the source file exists on the hard disk at the indicated reference.

Use Status of. As any T-FLEX drawing, the fragment has its own drawing settings defined in the commands **ST: Set Document Parameters** and **SH: Set Levels**. Those include the line thickness, the font size, levels, etc. One of the two status settings can be selected for the fragment being inserted into the current drawing:

Fragment. In this case, the fragment will be inserted with the settings defined in the fragment drawing.

Current Document. The inserted fragment will adjust to the settings of the current document. This choice is used when the assembly drawing is required to maintain a uniform style. Besides, by changing the ranges of visibility levels you can "turn on" or "turn off" certain portions of the fragment drawing when it is used in an assembly. For instance, one can remove "extra" drawings or dimension symbols.

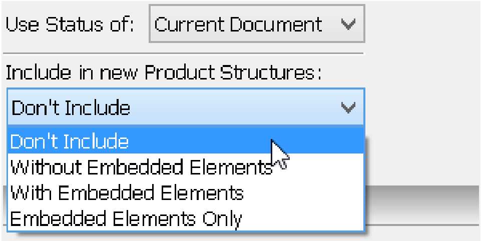
Include in Product Structures. This parameter manages incorporation of fragment data into the bill of materials of the current drawing. It can take the following values:

Don't Include - the fragment is not entered in the BOM table.

Without Embedded Elements - the fragment is entered in the BOM table. If the fragment is an assembly itself, then only the fragment proprietary data is entered into the BOM. The information about the nested elements (lower-level fragments) is not included.

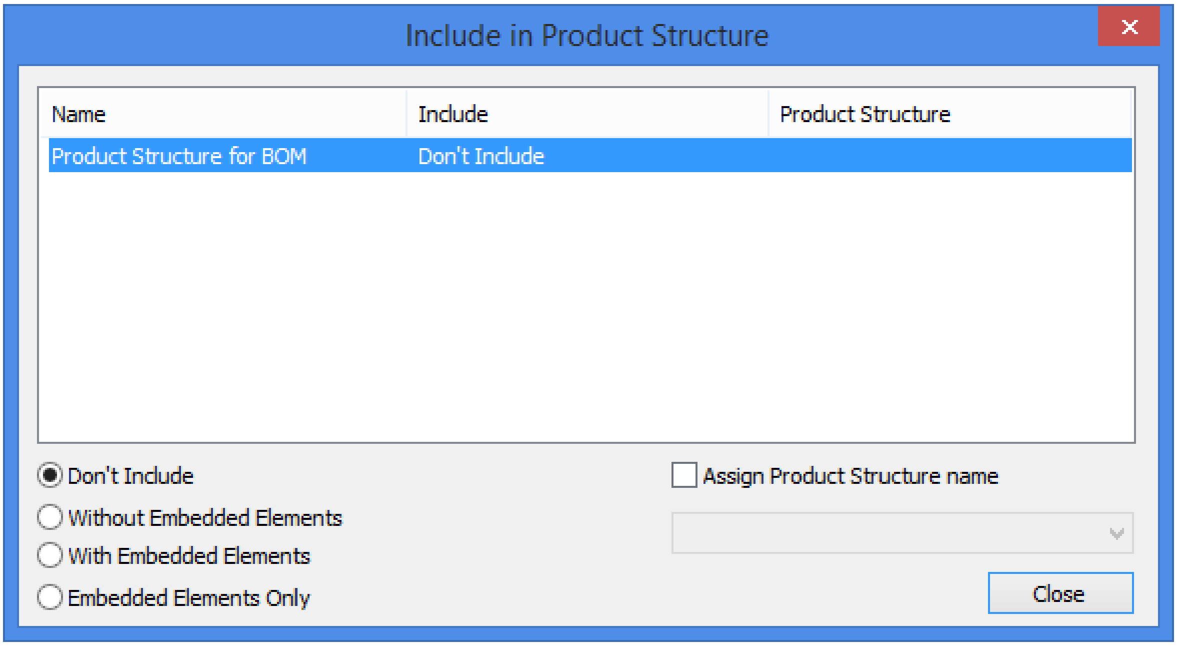
With Embedded Elements - the fragment is entered into the BOM together with the nested elements.

Embedded Elements Only - only the nested elements are entered into the BOM.



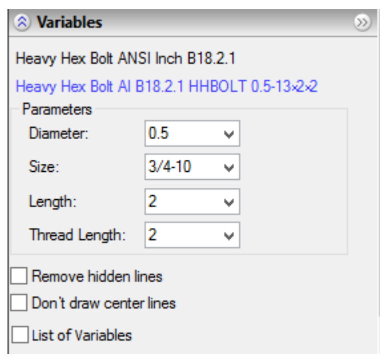
Additionally, a graphic button **[Include in new Product Structures]** will be provided for displaying the window with the list of BOMs existing in the current document, and the settings of the fragment contribution to each of them.

By activating the **Specify workpiece's structure name** option, one of the structures of the workpiece which are present in the fragment's file can be selected. This option works only when the **with embedded elements** and **only embedded elements** parameters are enabled.



The way of including the given fragment into the assembly's structure can also be specified with the help of the command:

Icon	Ribbon
	Bill of materials → Product structure → Included fragments
Keyboard	Textual Menu
<BI>	Tools > Report/Bill of materials > Included fragments

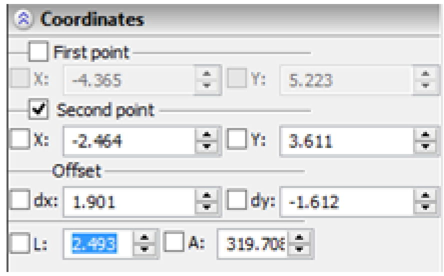


The section "Variables" allows a user to specify the values for the fragment external variables.

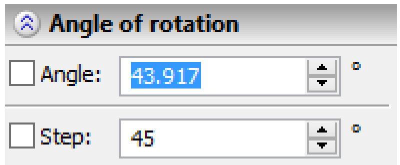
This section is present in the command's properties window only if the fragment being inserted has external variables.

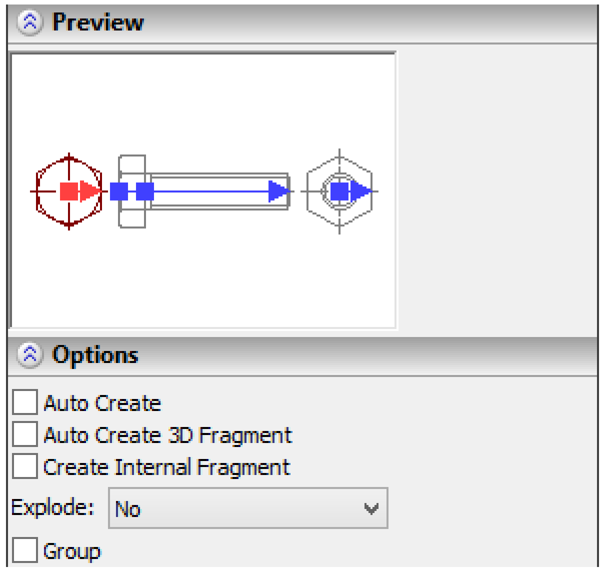
Both the list of variables and the user-defined dialog of the fragment parameters (if it has been created in the document of the fragment being inserted) can be used for specifying external fragment variables. The choice of the desired way to work with the variables is made with the help of the flag "List of variables".

To make the work with user-defined dialogs more convenient, the section «Variables» in the command's properties window can be brought out as a separate window by pressing the button . For bringing the dialog for specifying the external variables inside the general dialog of properties window, it is enough to close it (with the button ).



The section "Coordinates" can be used for specifying precise coordinate values for the points of the fixing vector or the fragment fixing points if these points are not being fixed to the elements of the drawing. The work with the fields in this section is similar to that described in the chapter "Sketch. Creating a Non-parametric Drawing. Automatic parameterization mode" of this manual. The structure of the controls in the section "Coordinates" changes depending on the situation:





Section "Preview". In this section of the properties window the diminished image of the fragment on the selected page is shown. The fixing vectors are also shown in this window if they exist. The desired fixing vector can be selected in this window by pointing at it. The selected fixing vector is shown with red color. The rest of the vectors are blue.

Section "Options" contains the following parameters:

Auto create 3D fragment. (Available only for 3D version). In case this flag is set on, right away after inserting the new 2D fragment the corresponding 3D fragment is automatically created with attachment by default (without using the coordinate system).

Auto create. When turning on this flag, the process of inserting the fragment will be finished automatically straightway after specifying the fixing points or the points of the fixing vector of the fragment.

Create internal fragment. This parameter allows a user to create an internal fragment, the contents of which are stored in the assembly drawing, but not in the external file.

In the dialog of the fragment parameters (option ) the following parameters can be specified additionally:

Priority. Defines the priority of the fragment in the assembly drawing. This is used for changing the order of drawing fragments within the current document and for the fragment hidden line removal. Priority is represented by an integer, positive or negative. Fragments with lower priority will be drawn in the screen prior to the fragments with higher priority. The priority of the current (assembly) drawing is equal to "0" by default.

Level. Sets the fragment level.

Layer. Sets the fragment layer.

The button Options. Provides a shortcut to defining the system parameters for handling fragments. The parameters in this dialog are the same as those on the tab **Fragments** of the command **SO: Set System Options**.

See the chapter “Customizing System”.

"Top-Down" Design

When using the "Top-down" design approach, separate parts-fragments are created directly within the assembly drawing window while working with the assembly. There are two approaches to creating a fragment when using the "Top-down" method - fragment **extraction** and **working in the assembly context**. In the first case, a fragment can be created by extracting into a separate file the necessary elements of the assembly drawing. The second case allows creating a new part drawing with the provision for referencing existing elements of the assembly drawing.

The fragments created in this way can be attached to the elements of the assembly itself or other existing fragments. This helps excluding or significantly reducing use of external variables and simplifies handling of an assembly. Meanwhile, some values of the original model parameters can be obtained directly from the assembly context. This approach significantly simplifies relating elements with each other and provides parametric relation between those. If dimensions or position of one of the parts is modified, then all related model elements will adjust automatically.

Working in the assembly context simplifies in certain cases the design process of the assembly module. This also facilitates development of the complete documentation suite of such a module, including detail drawings of all contributing fragment parts. Upon modifications to any assembly document, either the assembly drawing itself or one of its fragments, the changes propagate to all documents of the assembly (automatically or by the user request). As a result, modifications to one part cause update of the full suite of new documentation for the assembly, including the assembly drawing itself and detail drawings of all contributing part fragments.

The "Top-down" design approach may not be suitable in all cases of designing assemblies. The method has certain shortcomings that limit its use:

- More complicated organization, compared with the approach "from part to assembly";
- Lesser robustness to topology changes. For example, once an assembly line is referenced by an introduced fragment, it can no longer be deleted, otherwise the fragment associative reference will be lost;
- This approach is less convenient in terms of reusing fragments in other assemblies, since modifying a fragment may be complicated without the availability of references to the original assembly;
- Upon an attempt to extract a fragment, if it is impossible to "detach" an element from the assembly drawing, additional copies of the necessary elements are created in order to preserve parametric relations in the assembly drawing;
- Somewhat higher computational resources are required.

The icons for handling the "Top-down" approach can be found:

- In the textual Menu **File** > **Fragment** > **...**;
- In the automenu of the command **FR: Create Fragment:**

	<C>	Create Fragment in Assembly
	<G>	Extract Fragment Drawing

Managing Fragments in Assembly Context

When using the option , the first step will be specifying the name of the fragment being created, using the "Save As" dialog box. After that, all construction elements of the assembly will be hidden in the drawing window, and the graphic elements drawn in halftone. While in this mode, all newly created construction and graphic elements will belong to the new fragment. As you create drawing elements, you can use one of the following modes of snapping to assembly elements:

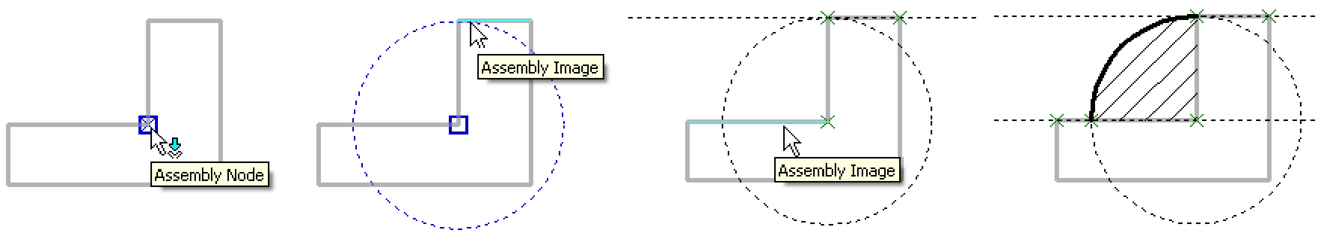
Associative snapping (the icon must be in the pushed state). In this case, the fragment elements can be snapped to the graphic lines and nodes of the assembly. By assembly nodes, we mean the joint points of the graphic lines and the attachment points of the detailing elements. This ensures two-way relation between the assembly and the fragment file. In other words, changes in the assembly drawing can be propagated, upon the user request, into the fragment file, and, vice versa, modifications in the fragment file cause the assembly document update. As the pointer approaches assembly nodes in this mode, those are highlighted and marked by the tooltip "Assembly Node", while the graphic elements – "Assembly Image".

Non-associative snapping (the check box pushed). In this mode, snapping to assembly drawing elements is also available. However, in this case, snapping is done to the current "snapshot" of the assembly. The future modifications of the assembly lines won't affect the fragment image.

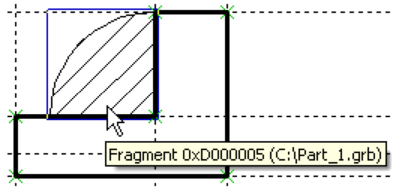
No snapping (both of the above modes should be undone). This mode is none different from conventional drafting. The assembly drawing displayed on the screen does not interact with the fragment elements in any ways.

The icons of the snapping modes can be found in the context menu and also in the textual menu "Customize|Snap|...".

The following diagrams show fragment creation in the assembly context, using associative snapping. In this mode, the assembly construction elements are hidden, while its graphic lines are shown in the halftone. When creating fragment construction lines, snapping to assembly elements is engaged. The graphic elements of the fragment drawing are created last.



Upon finishing working with the fragment, the system returns to the normal mode of working with the assembly drawing. If the created fragment was saved, its image will appear in the assembly.



To finish working with a fragment, use the options in the context menu:

	<FF>	Save Fragment and Return to Assembly
	<FQ>	Close Fragment

The option completes working with the fragment either with saving the results or without saving, at user's choice

Extracting Fragment from Assembly Drawing

The option serves for creating a new fragment by moving or copying into a separate file already existing elements of the assembly drawing. Upon the call, the following options appear in the automenu:

	<End>	Finish Fragment Creation
	<M>	Select Mode
	<M>	Deselect Mode
	<F>	Assign Fixing Vector
	<V>	Select Variables for copying into Fragment
	<D>	Delete or hide selected elements after creating Fragment
	<Esc>	Exit command

Fragment extraction is the action opposite to that of the option “Explode Fragment”. To create a fragment, the user just needs to select a set of graphic elements (lines, dimensions, hatches, etc.) of the assembly drawing to be carried over into the separate fragment. When creating a fragment, besides the graphic elements picked in the assembly drawing, their respective construction elements are also created.

The option allows picking the assembly drawing elements, adding those to the contents of the fragment. (The selected elements are highlighted.)

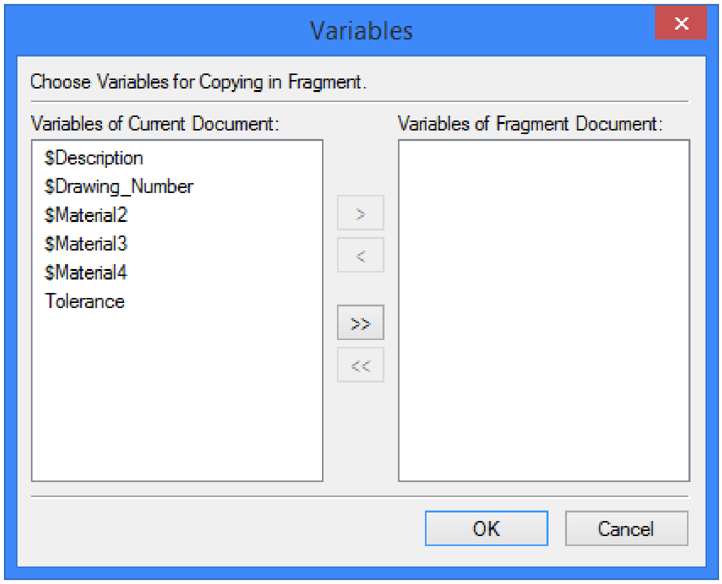
The option excludes elements from the selected set.

The option toggles between the modes:

- If the icon is in the pushed state, then, after creating the fragment, the elements included in the fragment are **deleted** from the assembly drawing. Exception is the parent elements of other elements. In this case, the system will not delete the element. Instead, it will make it hidden (invisible) by assigning a special attribute;

- If the icon is not pushed, then the fragment contents are formed by the copies of the selected elements.

Creation of a fixing vector (the option) is optional. However, if you need to make a provision for modifying the attachment of the fragment being created or placing its duplicate at other locations, then you should perform this step. A fixing vector can be created in one of the ways described above.

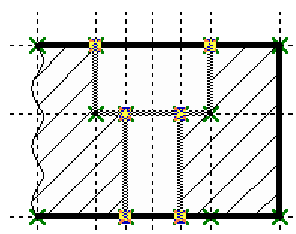


The option brings up the dialog box for selecting variables existing in the assembly drawing, to be copied to the fragment.

The list in the left-hand side pane contains all plain variables. Variables defined by an expression cannot be copied into a fragment. The variables can be carried from the left to the right pane and back using the graphic buttons [>], [<]. The buttons [>>], and [<<] allow carrying over the whole list. All fragment variables that originated from the assembly drawing are automatically deemed "external".

The fragment creation can be completed using the icons:

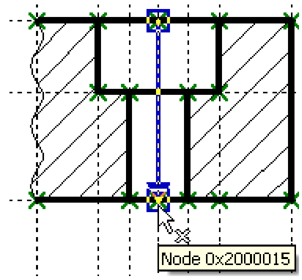
	<End>	Finish Fragment Creation
	<Esc>	Exit command (without saving the fragment)



Calling the option brings up the dialog box for defining the fragment filename. If variables to be copied to the fragment were not defined in the course of the fragment creation (as they would have been by the option), the dialog for selecting variables will appear prior to the filename defining dialog.

The diagrams on the right hand side show the fragment extraction.

The first step is selecting the elements to include in the new fragment.



After that, a fixing vector is created.

Next, the necessary variables are selected, and the filename is defined. As a result, the selected lines are placed in the assembly drawing by the 2D fragment possessing the set of the specified parameters.

Whenever possible, we recommend creating all elements directly in the new fragment, rather than carrying those over from the assembly drawing. This approach enhances productivity.

Editing Fragments

General Information about Fragment Editing

Fragment editing can be performed in order to modify its parameters (references to file, image settings, the way it is included into the bill of materials, etc.), fixing, variables. In some cases in order to make changes, it is enough to turn to the dialog of the element parameters. Sometimes it is convenient to use the fragment-editing command **EFR: Edit Fragment**. It can be called as follows:

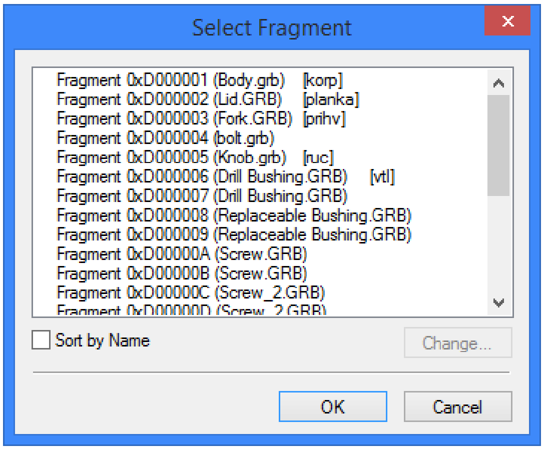
Keyboard	Textual Menu	Icon
<EFR>	"Edit > Draw > Fragment"	

As for any other T-FLEX element, the common selection and editing rules apply to a fragment. That means, selection of the necessary fragment for editing is done by clicking , while the option <P> or is used for changing its parameters, etc.

To select multiple fragments, one can use the key combination <Shift>+ (for adding elements to the selected list), <Ctrl>+ (excluding an element from the list) or the box selection.

To select a fragment by name, use the option:

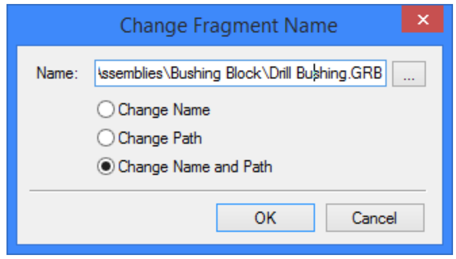
	<S>	Select Fragment from list



This option can be helpful when the fragment with the known name is hard to find in the drawing. Besides, the option may be very helpful for editing the reference paths of several similar fragments simultaneously.

A dialog box will appear in the screen, where you can select the desired fragment.

By default, the fragments appear in the list in the order of their insertion into the assembly drawing. The flag **Sort by Name** alters the order of fragments in the list. The graphic buttons **[Change]** allows changing the filename, the reference paths or both, of the selected element (or a group of elements) in the list.



The fragments which because of some reasons lost connection with the external file are marked with the question sign.

Selecting the option **Change Name** allows modifying the filename only, without changing the path. Selecting the option **Change Path** modifies the path to the fragment without changing its name.

Selecting the option **Change Name and Path** modifies both the Name and the path of the fragment file.

> We recommend using relative paths to the assembly or library rather than full paths, when inserting fragments. Using relative paths simplifies porting assembly drawings to different file systems.

In some cases, the system may automatically record the relative path to a reference with respect to the assembly file. Depending on the relative location of the fragment and the assembly file, the relative path to the reference may appear differently. The following table shows examples of relative path formats for fragment references:

Assembly location	Fragment location	Fragment reference path
C:\Assembly.grb	C:\Drawings\Parts\Part3.grb	Drawings\Parts\Part3.grb
C:\Drawings\Assembly_1\Assembly.grb	C:\Part2.grb	..\..\Part2.grb
C:\Drawings\Assembly_1\Assembly.grb	C:\Drawings\Assembly_1\Part1.grb	Part1.grb
Any	Library "Bolts", file "Bolt_1.grb"	<Bolts>Bolt_1.grb

Follows are the rules that are accepted for the fragment reference relative path formats:

1. If the assembly is located at a higher level in the file tree, than the fragment, then the reference will be always relative to the assembly file. In this way, the portion of the full path to the reference, identical with that of the assembly file path, is removed, resulting in the relative path to the reference.
2. If the assembly is located deeper in the file tree, then the fragment, yet in the same tree branch of the file system, then T-FLEX CAD system will be able to create the relative path for the reference. In this way, to step up one level in the file tree, the system will use the notation "..\" in the beginning of the reference path. However, if the reference requires switching to another branch of the file tree, the user will have to specify the reference path manually, as the system won't be able to do this automatically.
3. If the assembly file and the fragment file are in the same folder, then the reference name will consist of just the fragment file name.
4. If the fragment file was inserted from an opened library, then the reference relative path will contain the library name, in brackets, and the fragment file name.

All fragments at once can be selected by the option:

	<*>	Select All Elements

Upon selecting multiple fragments, the following options are available:

	<P>	Set Fragment Parameters
	<Del>	Delete selected Element(s)
	<X>	Explode Fragment
	<Ctrl+X>	Explode with Constructions
	<R>	Select Fragment from list
	<I>	Select Other Element
	<U>	Update Fragment model
	<Esc>	Cancel selection

The options and explode the selected fragments, turning those into sets of drawing elements, with or without the original constructions.

The option call loads the changes from the fragment file.

The available editing options for a single selected fragment are:

	<K>	Input Fragment insertion points
	<P>	Set Fragment parameters
	<Y>	Create Name for selected Element
	<Del>	Delete selected Element(s)
	<Ctrl+O>	Open Part
	<O>	Open Fragment in Context of Assembly
	<H>	Update Fragment File
	<X>	Automatic Explode
	<Ctrl+X>	Explode with Constructions
	<T>	Open Fragment file for editing
	<U>	Update Fragment model
	<C>	Select Clipping Hatch

	<V>	Inner Fragment Variables
	<Z>	Change Fixing Vector
	<R>	Select Fragment from List
	<I>	Select Other Element
	<Esc>	Cancel selection

Upon selecting a fragment, it will be marked by an outlining rectangle. All fixing points or the fixing vector of the fragment will be highlighted.

The option allows defining the fragment name. The fragment name can be used, for example, for search, automatic creation of the named nodes of the fragment in the assembly, for accessing the values of the fragment variables in the assembly drawing by the function get:

get ("Fragment name","Variable name"),

where "Fragment name" – the specified name of the fragment, and "Variable name" – the variable name in the fragment drawing, whose value is being accessed in the assembly drawing.

The selected fragment document can be opened for editing in a separate window, using the option . In this case, the parameters of the opened drawing will be those defined at its creation. The option starts fragment editing in the assembly context.

Option allows switching between fixing vectors defined in the fragment being inserted.

Option allows editing external variables of embedded fragments without opening of source fragment. After activation, you need to choose one of embedded fragments.

A detail drawing of the fragment can be obtained by the option . In this case, a new window is opened, with a copy of the fragment drawing loaded in it with the parameters corresponding to the assembly. The thus created drawing can be edited and saved under a new name, if necessary.

Ways to Edit Fragments

Modifying Fragment Attachment

If a fragment has several fixing points, to change all their positions (or those of the fixing vector), use the option . After that, just as when inserting the fragment, subsequently define the position of each fixing point.

Modifying a particular fixing point (or a placement point of the fixing vector) can be done right after selecting the fragment. The order of steps in this case is described below.

The case of fragment attachment by fixing points

To modify a placement point, move the pointer to it and click . The fragment will start changing dynamically together with the cursor until the new location of the point is specified by clicking or by one of the automenu options:

	<U>	Dynamic Preview
	<I>	Select next Fragment insertion point
	<N>	Select Node
	<Z>	Change Fixing Vector
	<Esc>	Cancel

To speed up handling of large fragments, the fragment rubberbanding can be turned off by the option .

The case of fragment attachment by fixing vector

To modify placement of a fixing vector point, move the pointer over it and click . The respective end of the fixing vector will start rubberbanding with the pointer. Set its new position by clicking or by one of the automenu options:

	<W>	Dynamic Preview
	<M>	Fixing using two points
	<M>	Fixing using one point and angle
	<C>	Mode of snapping to Connectors
	<Z>	Change Fixing Vector
	<N>	Select Node
	<I>	Select next Fragment insertion point
	<Esc>	Cancel selection

Clipping Fragment by Hatch

The option is provided for selecting a hatch, whose contour will be used for clipping the fragment image. Upon selecting a hatch, only the portion of the fragment will be displayed that is within the contour of the selected hatch. If the image of the clipping hatch is not needed on the drawing, the hatch should have the parameter “hidden” set on.

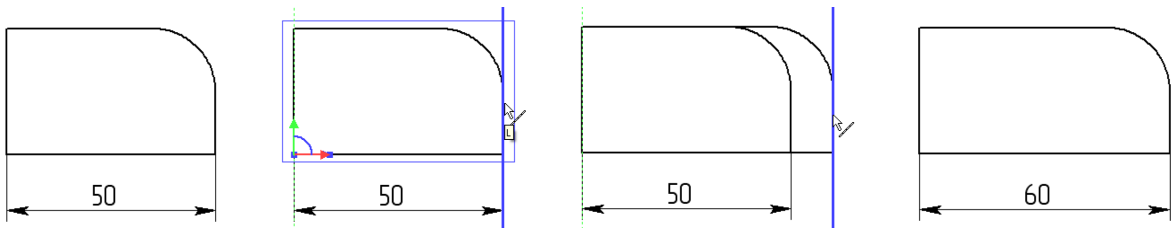
Clipping by the hatch can be used if you need to use a hatch pattern not included in the standard set. Perform the following steps:

- create a new document with a set of graphic lines representing a "hatch" pattern;
- in the document in which it is required to draw a nonstandard contour, create an invisible hatch the contour of which corresponds to the required one;
- insert the document with the hatch pattern created at the step one, as a fragment;
- call the fragment editing command and select the hatch fragment. Then, with the option  select the contour of the invisible clipping hatch.

The option  ceases hatch clipping (this option appears only when editing a fragment clipped with a hatch).

Editing External Variables Using Draggers

For additional convenience, the system allows varying the fragment external variables dynamically by the mouse, using specially provided draggers. Draggers appear as thickened images of construction lines that are driven by external variables. This mode is on by default. Switching it On/Off is done within the document customizations on the tab "Preferences", the graphic button "Fragments".

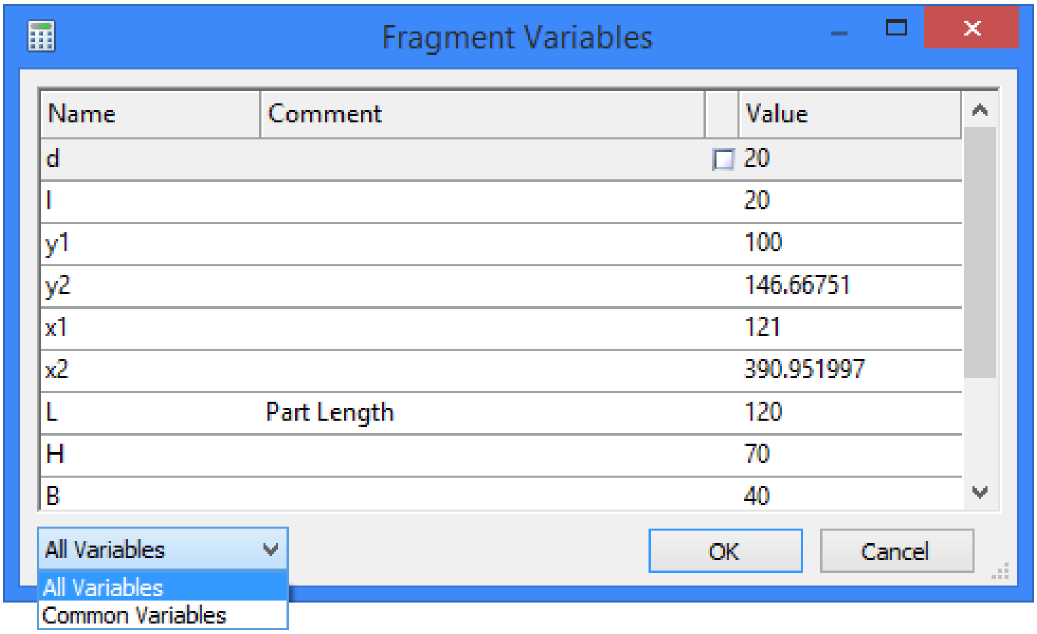


Variables Editing for Several Fragments

The system allows the simultaneous modification of the variables with the same name for several fragments at once. For modifying the fragment variables it is necessary to perform the following actions:

- select fragments on the drawing;
- invoke the context menu by pressing the right mouse button and choose the command **Variables**;
- introduce modifications in the dialog which appears.

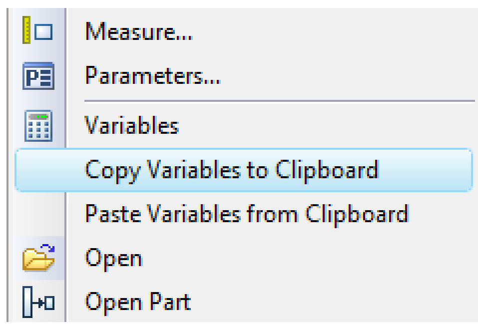
Upon calling the command of the context menu **Variables**, the dialog with the list of variables of the selected fragments appears. This list may include either all variables of all fragments or only common variables (i.e., the variables with coinciding names). Control is carried out with a toggle **Show variables** at the bottom of the dialog.



The common variables in different fragments can have different values. In this case, for such variables opposite their value there will be a special control element – a flag which when being switched on will let a user change the value. Modification of the value for the common variable will be applied to all selected fragments.

Using Clipboard for Fragment Variables

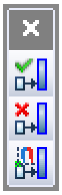
When selecting a specific fragment with , the commands for copying variables from one fragment to another will be found in the context menu. The command "**Copy variables to clipboard**" copies values and expressions for the variables into the internal clipboard. The command "**Insert variables from clipboard**" replaces the values of the variables having the same name for the selected fragment with the values from the clipboard.

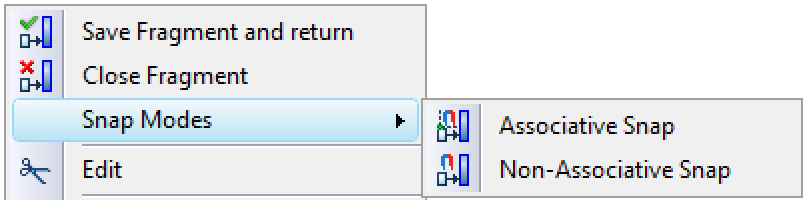


Editing Fragment in Assembly Context

The option for editing in the assembly context is available in the context menu of the selected fragment and in the automenu of the command **EFR: Edit Fragment**.

Upon calling the option, all elements of the assembly drawing other than those belonging to the fragment are drawn in halftone, and the fragment elements become editable. As in the assembly-context fragment creation, the user can create and/or edit the fragment drawing. With the associative or non-associative snapping to assembly elements turned on, you can use nodes and lines of the assembly drawing as references. The commands for modifying the snap types and exiting the mode of work in the assembly context can be found in the context menu.





Updating Fragments Files

The options **Refresh Fragment File** and **Refresh All Fragments Files** (the icon ) are provided for updating the fragment document per the changes in the assembly when working by "Top – down" approach or in the assembly context. To update a single fragment, you can use the respective option in the automenu of the command **EFR: Edit Fragment** or the command **Update**, accessible in the context menu. To update all fragments per the changes in the assembly drawing, use the option "Refresh All Fragments Files" in the textual menu item **File** > **Fragment** > **Refresh Files**.

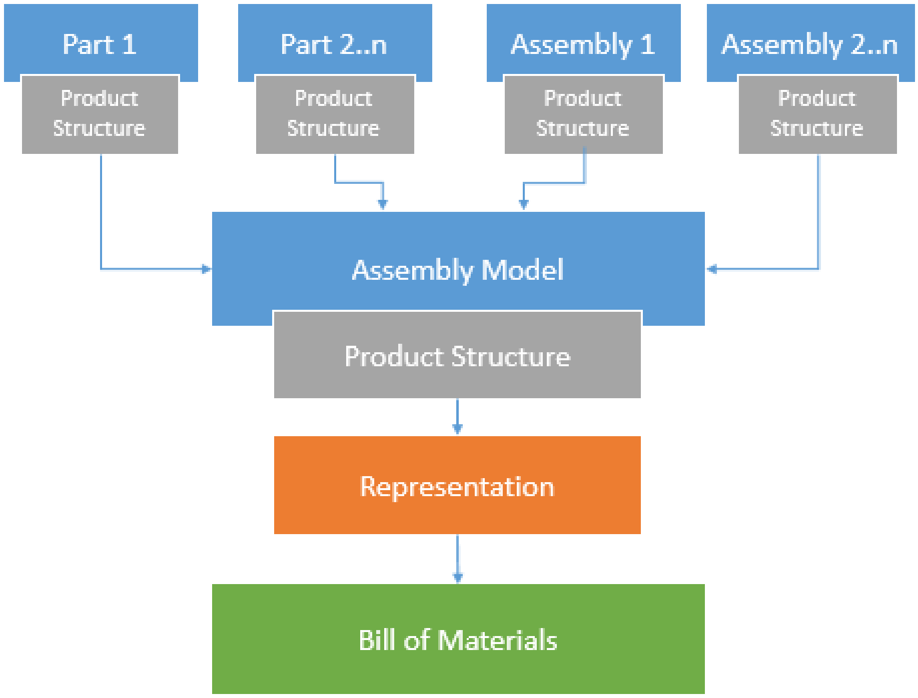
Bill of Materials

PRODUCT STRUCTURE, REPORTS, BILLS OF MATERIALS

T-FLEX CAD provides convenient set of tool for creating standard and custom BOMs and reports.

Report is a flexible mechanism for creating technical documentation in customized formats. This mechanism can be used for a standard BOM creation as well as for your own reports based on the product structure. Thereby program technical documentation tools can be adapted for various specific industries.

BOM in program is a table containing data about fragments (parts, subassemblies) inserted into the main assembly. If necessary, one drawing document may contain several BOMs.



BOM data is automatically copied from fragment files or added manually by user. In addition to fragments, the data from other sources can be included in the product structure – from 3D objects or drawing elements.

BOMs and reports can be created on the base of prototypes or report templates.

The BOM prototype is a program document that contains empty BOM table with preset properties. The BOM prototype describes structure of columns and sections for the created table. Prototype-based BOMs is a legacy mechanism and is not recommended for usage in the new documents.

The report template is a more flexible mechanism with extra options comparing to BOM prototypes. When you work with the product structure, the templates usage is preferable because of various advantages.

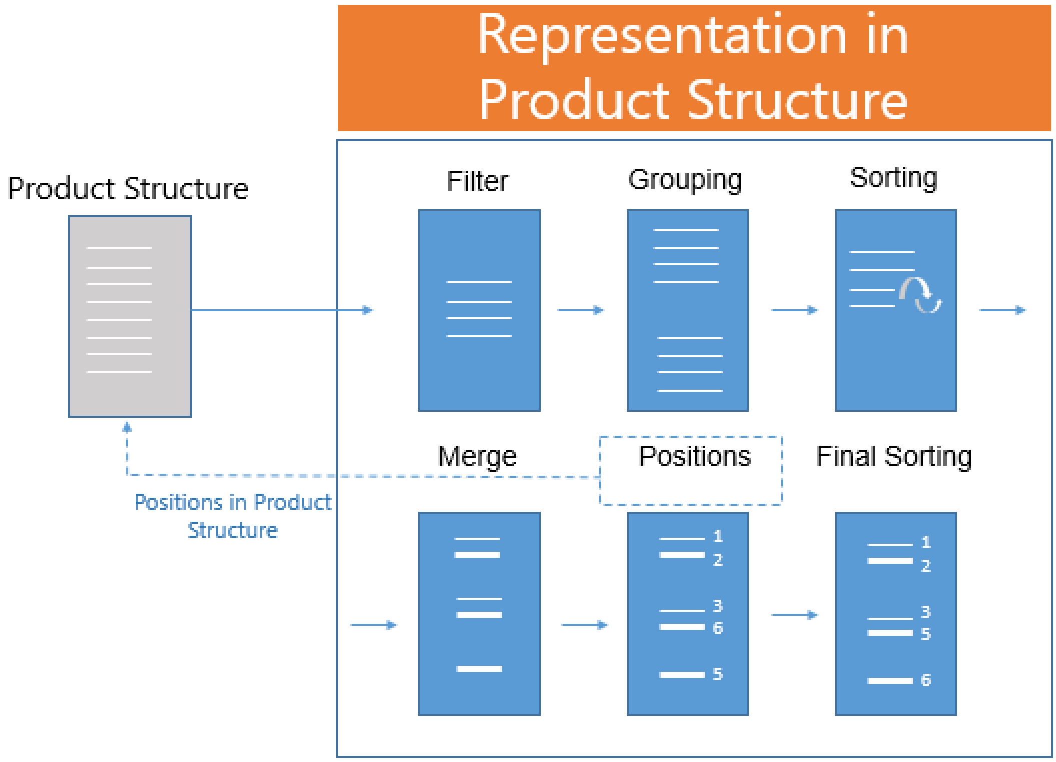
The report template is a program document file. BOM format and its data structure are described in this file according to certain rules. Tables and data description are inserted as text entities that are created in the template.

The template may contain macros (special microprograms) for additional template data processing, for example, empty rows adding.

Each macro has some number of attributes. Their values can be set for inputting when you create a report. Report is filled according to the product structure and the report template.

Report and BOM data is displayed in **Product structure** window.

The **Product structure** is used to collect technical documentation data. The data about the structure and the hierarchy of a product can be added to product structure manually or automatically. You can specify sorting, grouping and displaying rules for data in the table. There are commands for callouts creation, export to Excel, report generation, etc.



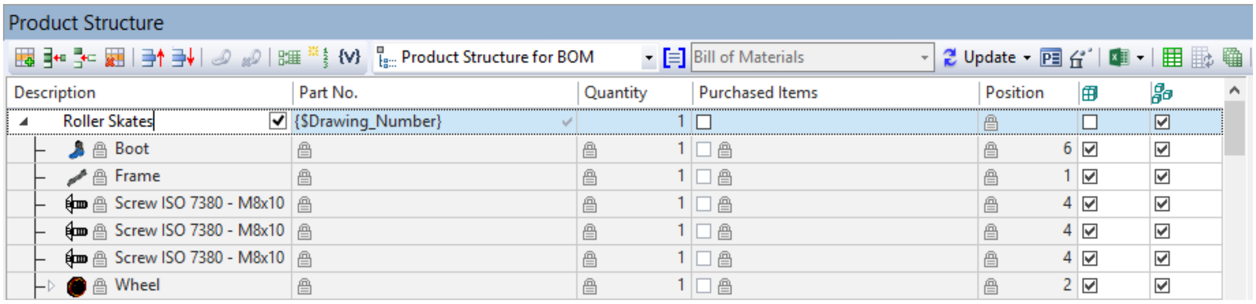
New product structures are created on the base of product structure types either included in the program installation or custom. Each type specifies various properties and column composition.

Several product structures can be created for one product. Each of them will display appropriate data for further generation of reports and BOMs.

PRODUCT STRUCTURE

Product Structure Window

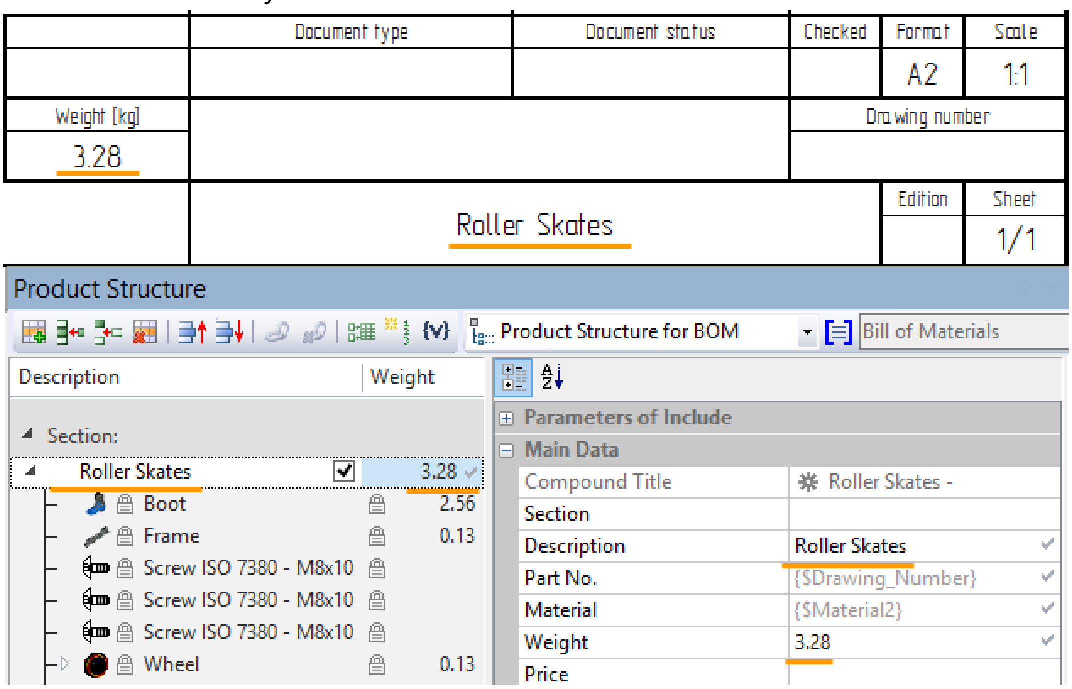
Product structure window is used for managing the product structure.



To open this window use option:

Icon	Ribbon
	Bill of materials → Product structure → Product structure
Keyboard	Textual Menu
<Alt+9>	Tools > Report/Bill of materials > Product structure

BOM data, collected from fragments and other objects or inserted manually, is displayed in **Product structure** window. Also there are additional tools for assigning data. When you fill in the title block, all its data is automatically included in the document product structure, because title block and product structure fields are associated by variables.



Product structure for assembly documents is automatically compiled on the base of product structure data of parts.

> Data from various assembly sources can be added to the product structure. It can be fragments, bodies, 2D elements, etc. By default fields of the product structure records are set as automatically filled to use data from the source elements.

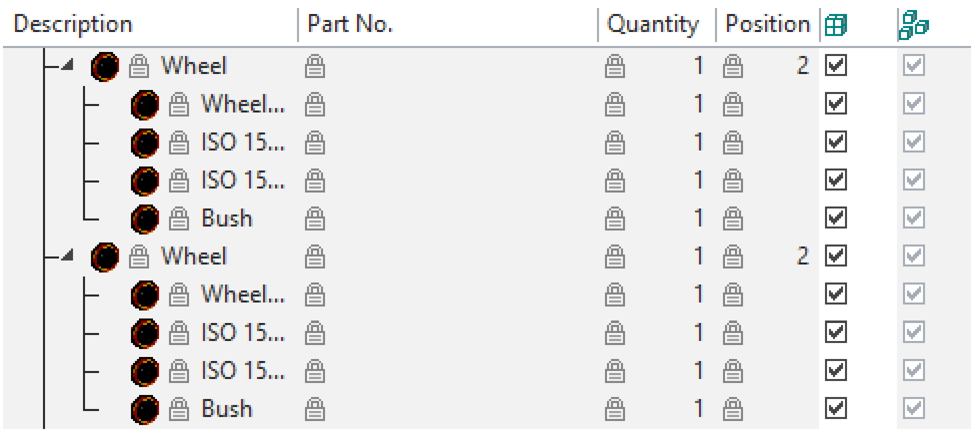
Icons indicate the field filling method. Automatic fields are indicated with icon. Manually assigned fields have icon. The third field type appears for specific fields that are also assigned automatically – titles, zones, page format, etc.

> If you need to edit the content of a field manually, click appropriate icon. Thus «Field is filled by user» mode activates and icon will change to .

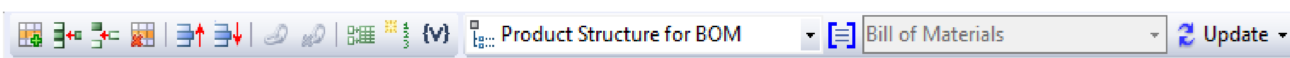
By default the new product structure has “General BOM” type.

Any product can be presented as the hierarchical structure (tree) in the **Product structure** window. The product itself is a root and its constituent components are presented as hierarchical records.

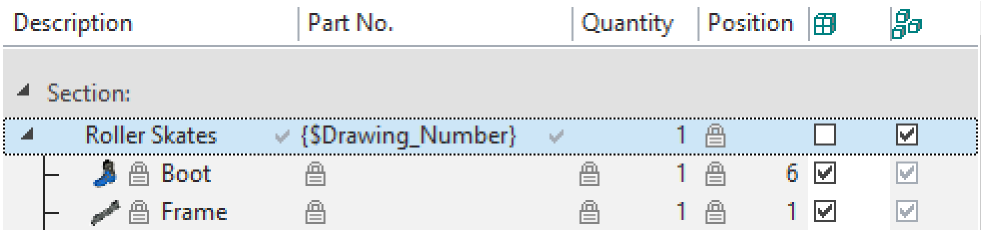
The product structure record is presented as a row in table with columns. It contains information about parts, quantities, materials, documents, etc. All data is inserted into data cells corresponding to the columns.



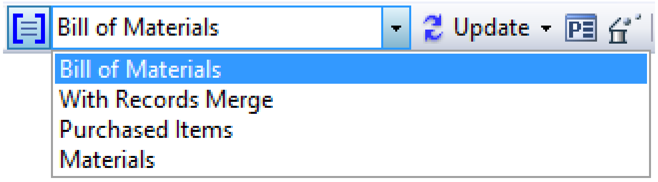
Product structure window has control toolbar that is located in the upper part of the window. It is used for various operations with the product structure.



Under the toolbar you may see the main product structure window that contains list of records, which are included into the current product structure. The following columns are displayed by default: “Description”, “Part No”, “Quantity”, “Position” and two columns for specifying inclusion of the records into reports and assemblies. If necessary, you can add/remove columns to be displayed in window manually.

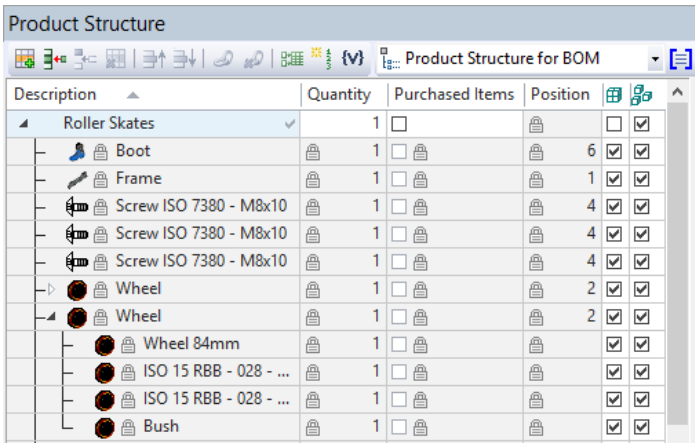


Every product structure may have various “representations”. Representations allow to display the same product structure data according to the preset rules. For example, similar parts of assembly can be merged and their quantity can be displayed in the corresponding column. Or some data may be filtered out. To use representations press **Apply product structure representation** button on the toolbar. Necessary representation can be then selected from the drop-down list.

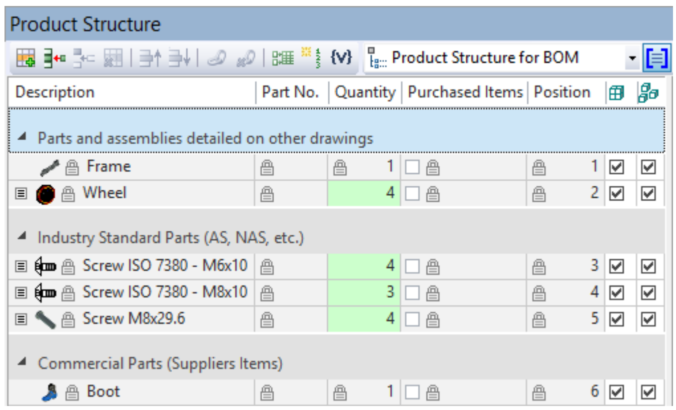


Each representation has its own properties set: displayed columns, sorting and grouping rules, filters, etc.

More information about representations can be found in “Representations tab” section.

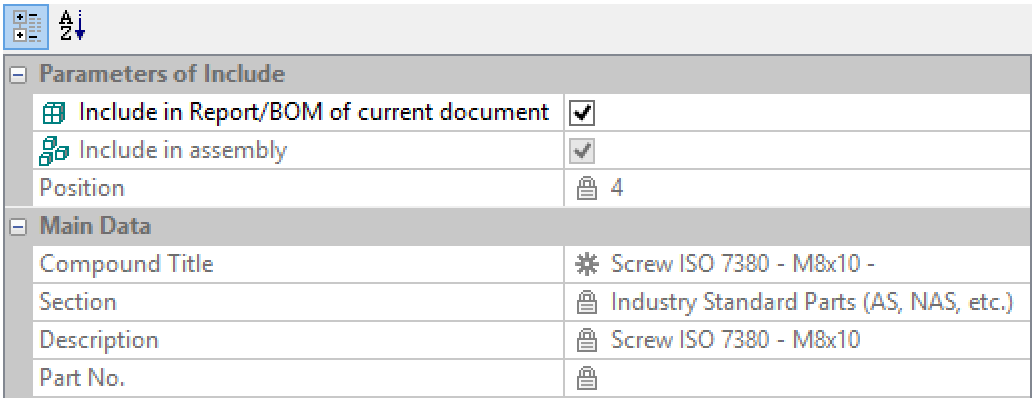


Without representation



With representation “With Records Merge”

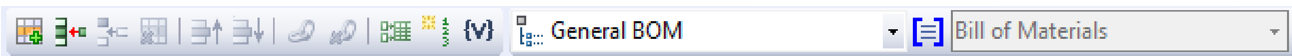
On the right side of the **Product structure** window you may open Parameters window via toolbar option **Show record parameters window.** All product structure columns and their content for the selected record are displayed here. Here you have access to all fields of the product structure record even those that are not displayed in the main window.



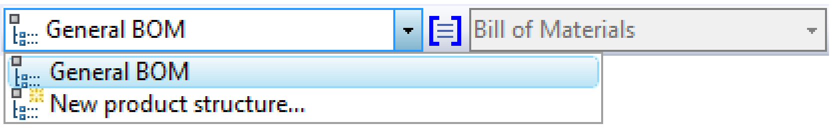
Toolbar

Product structure window toolbar provides options for:

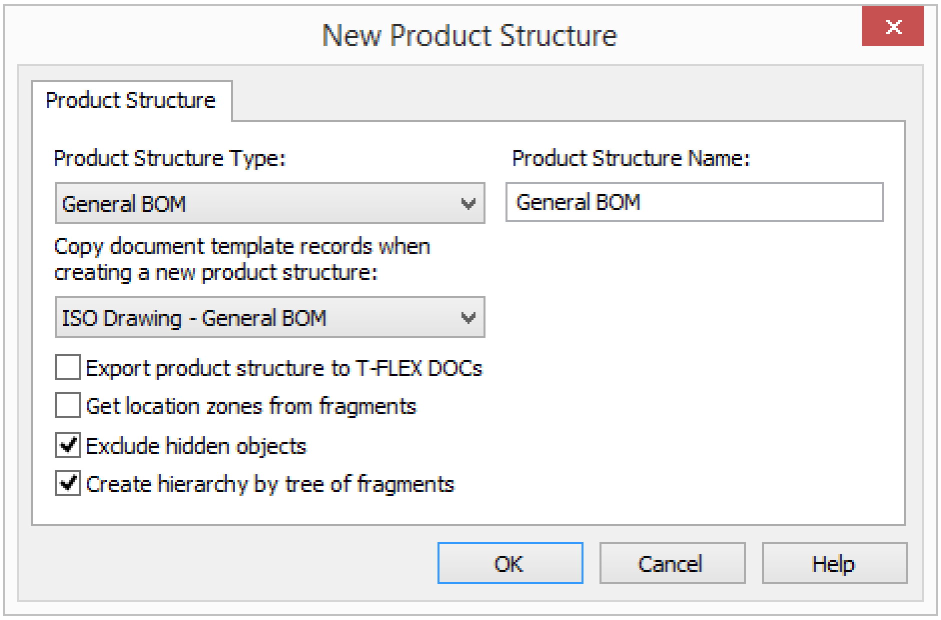
- creating and deleting the product structures;
- editing the product structure properties;
- data adding, grouping and sorting;
- setting relations between product structure records and objects in the current document;
- reports generation and updating;
- callouts creation;
- export to Excel.



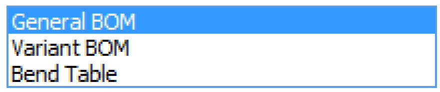
Product structure name is displayed in the field:



Every new document already contains the product structure called "General BOM" created on the base of "General BOM" product structure type.
If you want to create another product structure you need to select "New product structure" item.
After that in **New product structure** dialog box you can specify **Product structure name** and other parameters of the product structure.



Product structure type. The drop-down list contains all available product structure types.



Copy document template records when creating a new product structure. Product structure records will be copied from the selected template to the new product structure. User can select any document as a prototype if he select **User-defined** item in the drop-down list.

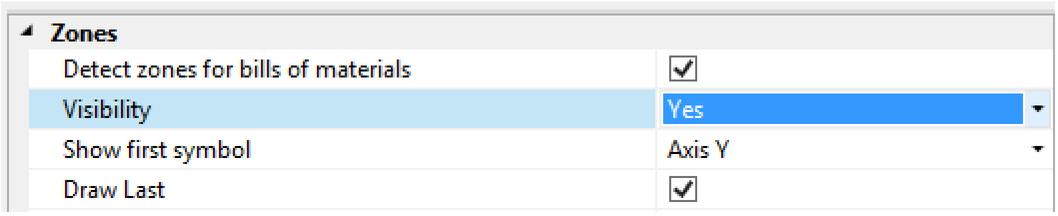
> Product structure type is a set of properties for product structures used as a template for new product structure creation. Properties include columns composition, grouping rules, sorting and records filtering. Each product structure type is stored in the separate file.
> New product structure types can be created with **BY: Product structure types** command.

You can set the following parameters in **New Product Structure** dialog box:

Export product structure to T-FLEX DOCs. If the flag is set, the current product structure will be saved when the file is exported to T-FLEX DOCs application. If there are several product structures, but no one was set for export, then the first one structure from the list is exported.

Getting location zones from fragments. If the flag is set, zones values are defined by the fragment location. If the flag is not set, the zone is defined by the callout location.

> Zones are used for simplifying search of parts in big assemblies. You need to open **Paper > Zones** tab in the **ST: Set Document Parameters** command. Set the corresponding flags.



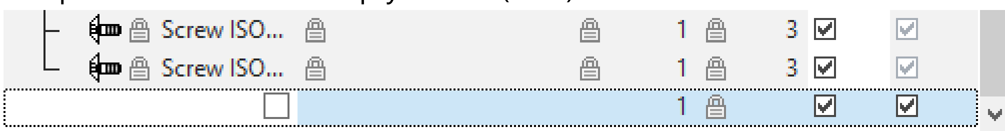
Exclude hidden objects. When the flag is set, only visible objects will be included in the product structure. Object no visible on the drawing/model, for example, hidden using levels mechanism or suppressed, will not be included.

Create hierarchy by tree of fragments. When the flag is set, the hierarchy, that exists in the fragment file, will be added to the assembly product structure. When the flag is not set, all items from the fragment file will be added to the same level in the assembly product structure.

When you create a new product structure based on the selected type, all properties will be copied and saved in the current file.

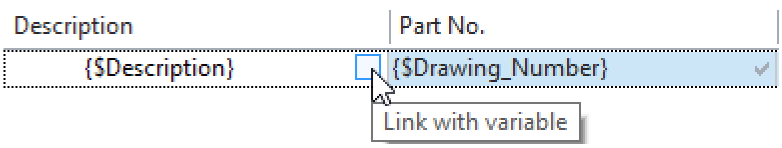
More information about the properties can be found in "Properties Tab" section.

Add record option creates a new empty record (item) in the **Product structure** window.

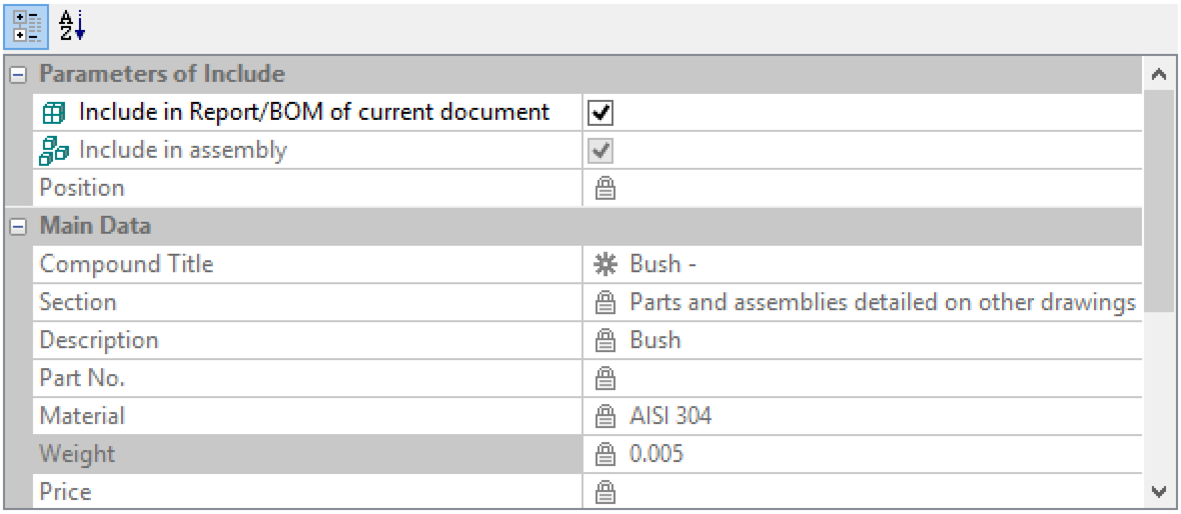


All new records are added to the zero hierarchical level. By default only records from first and upper levels are displayed in **Apply product structure representation** mode. That is why the new records will not be displayed in this mode.

If there are several sections in the product structure, the new record will be created in the section that has input focus. Data is entered manually. Each data cell can be associated with a variable by setting the flag in the field.



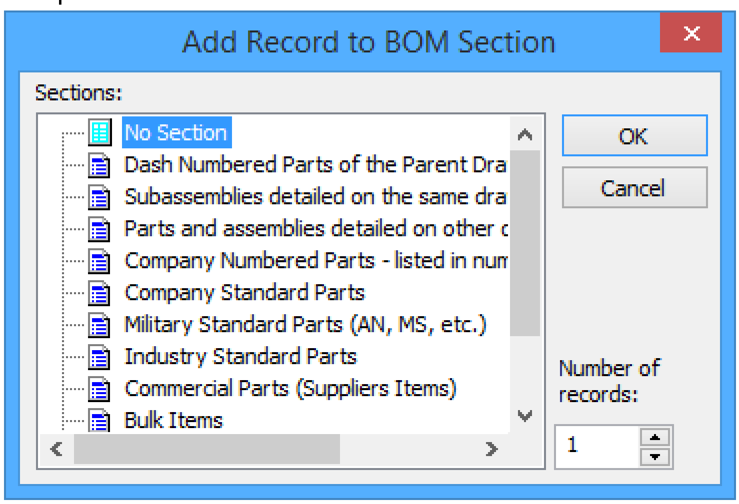
To manage all existing data cells of the current record use **Show record parameters window** option.



List of columns (cells) can be sorted using **Categorized** and **Alphabetical** options in the top of the window.

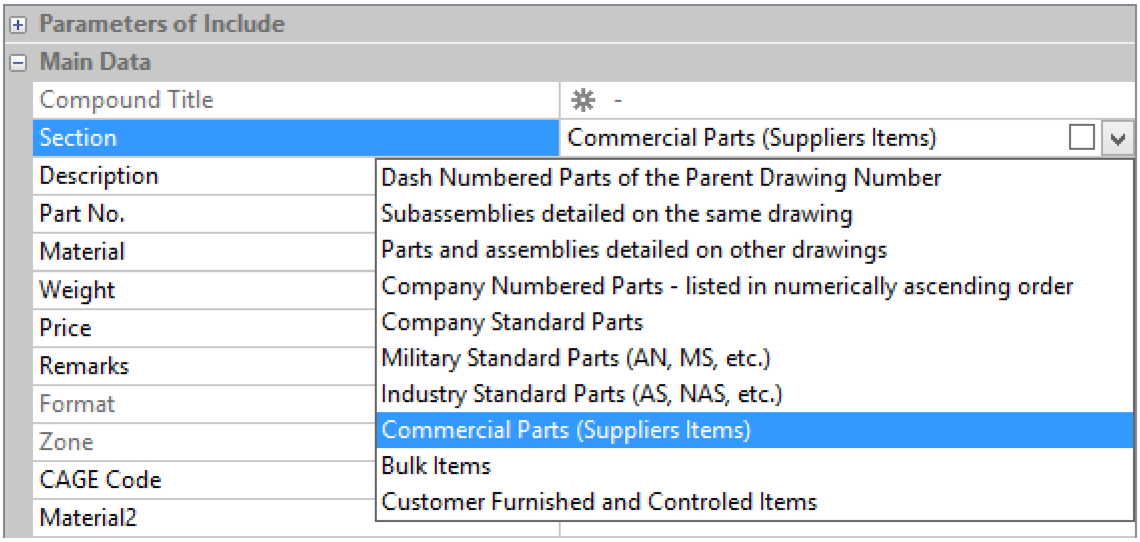
Add child record. Option is used to form the hierarchical structure of a product. It creates an embedded record relative to the current. The new record is indicated with icon.

Add record to section option is used to add a new record to the section.



If there are any records existing in the required section, you can point the cursor on one of them and add a new record with option.

Also you can specify the record section in the parameters window, by pressing on the corresponding field in the drop-down list.



Delete record option deletes the current record from the product structure.

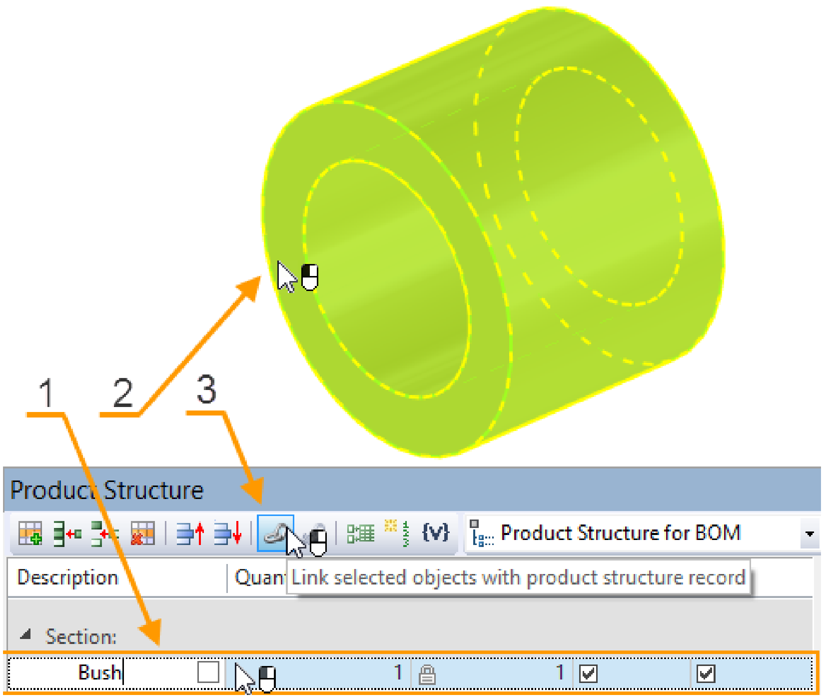
Move records up/down. This options allow to move selected records in the product structure. Moving is allowed only inside one section and with inactive sorting.

Link selected objects with product structure record option allows to set links between product structure records and document objects (3D operations, 3D construction elements, 2D drawing elements).

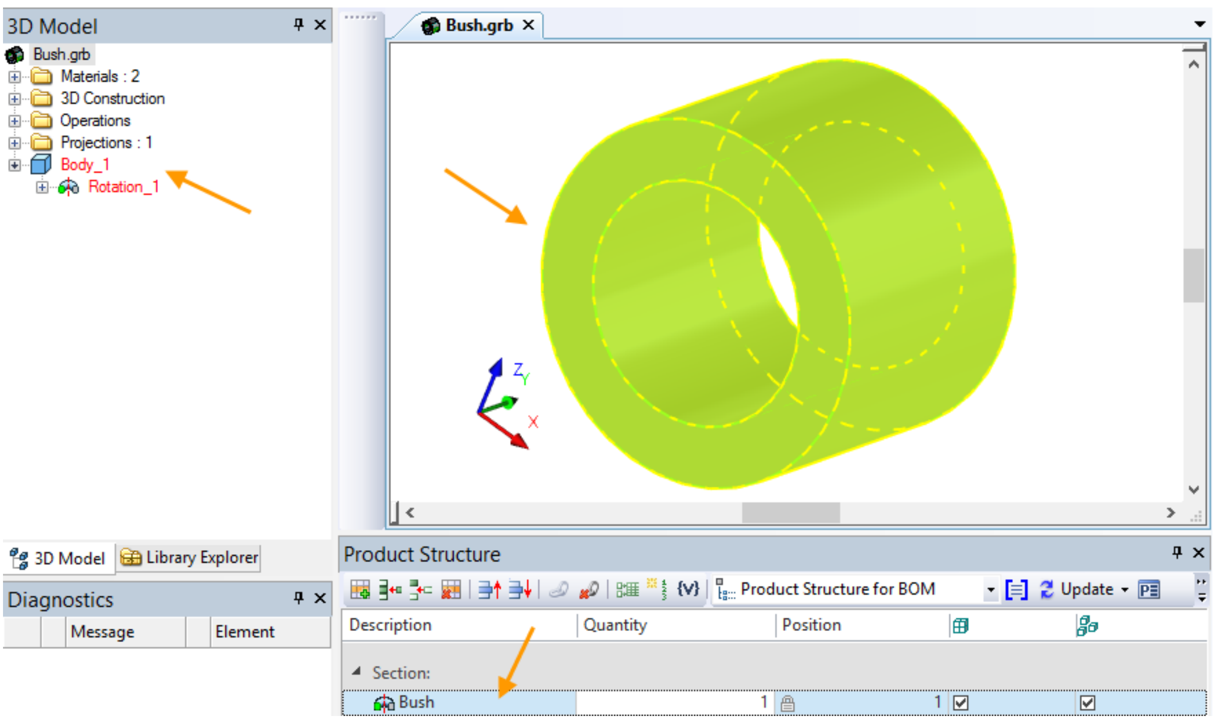
> The option is available only for manually created records. Such link cannot be set for records from fragments or any other data sources.

To set the link, perform the following actions:

1. Select record in the **Product structure** window. The string will be marked.
2. Select object in the scene with mouse click (it will be highlighted).
3. Press icon.



When the record is selected in the **Product structure** window, the linked objects are highlighted in the scene and in **3D Model** window.



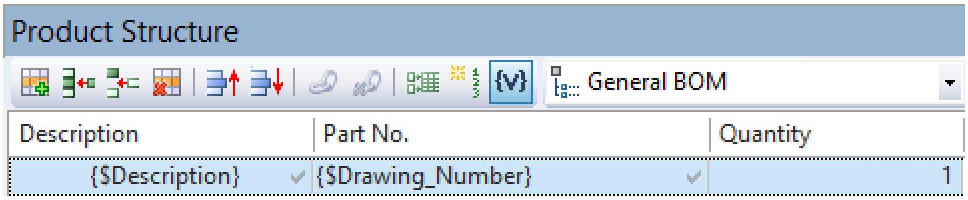
It is possible to link one record with several objects.

You can delete the link using **Delete link of product structure record with objects** option.

Use **BL: Create BOM Callout** command to create callouts. For quick access to the command use icon .

> More information about the command can be found in section "Callout Creation".

Show variable names. When option is active, the variables names are showed in cells that have link with those variables. When the option is inactive, the variables values are shown.

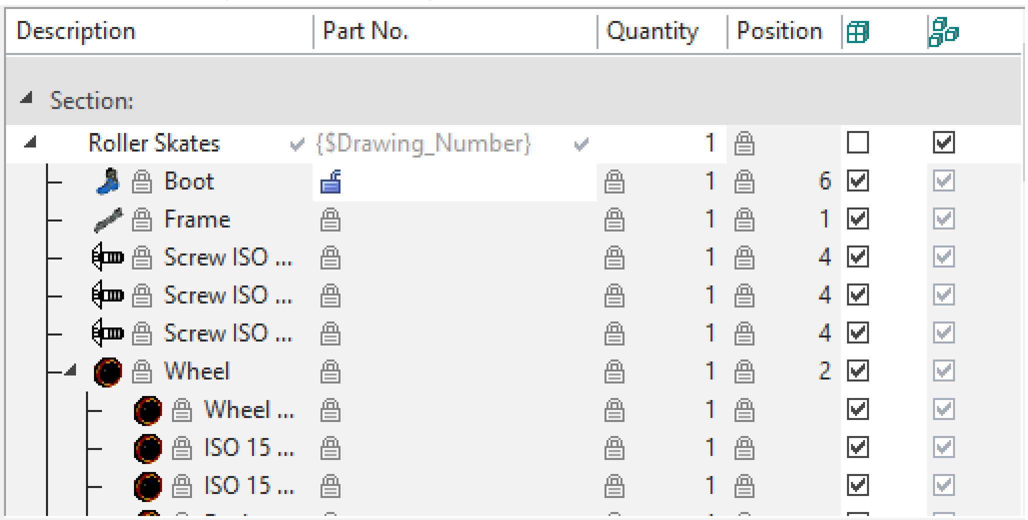


Use **BI: Include in product structure** command to manage parameters of the fragments inclusion into the product structure. You can use icon on the toolbar to activate the command.

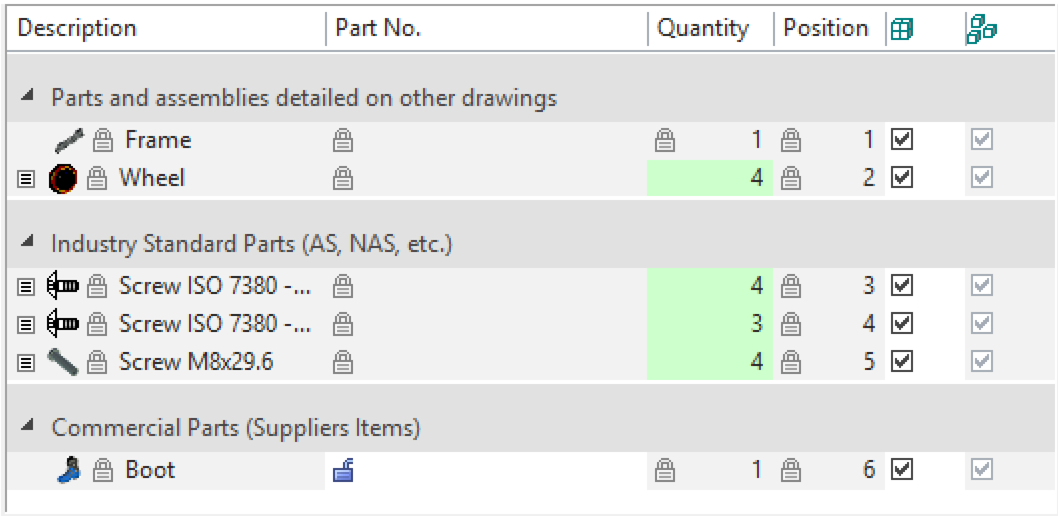
> More information about fragments inclusion can be found in section "Include in product structure".

Apply product structure representation mode shows list of records according to the grouping and sorting rules set in the representation. When the mode is active, group names are displayed in the list and representations selection becomes available.

The mode allows to display data according to the preset rules of representations.



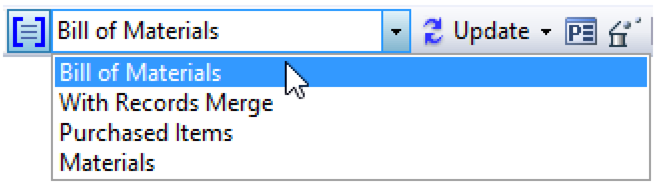
Product structure without representation



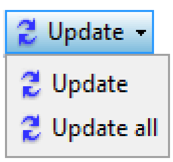
Product structure with representation that corresponds to standard BOM table. In this case, the group names are the names of the BOM sections

Fields are indicated with color if their values are summed.

Representation field. If there are several representations in the product structure properties, then you can select one of them in the **Apply product structure representation** field.



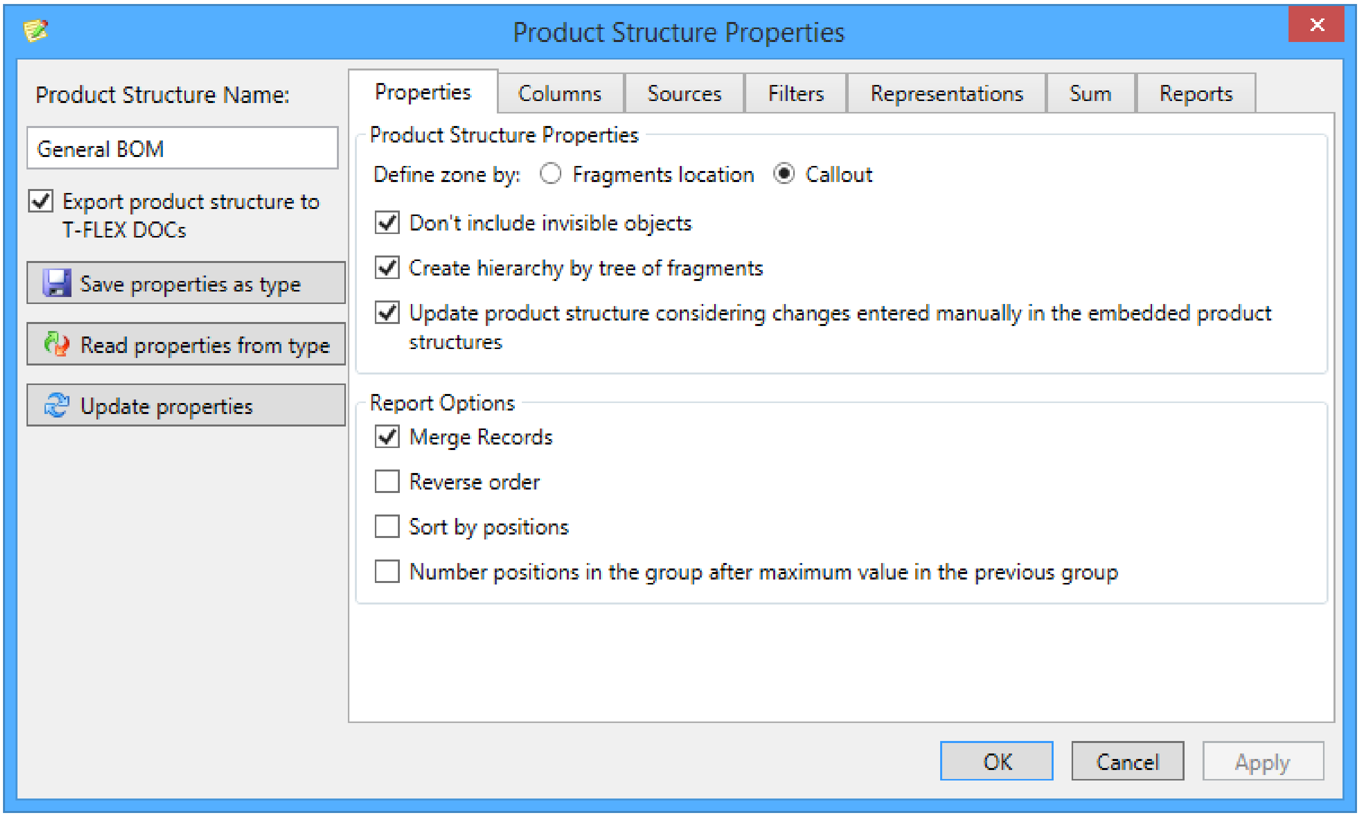
More information about representations creation can be found in section "Product structure types".



Update. The command updates current product structure after changes in the assembly document or fragments.

Update all. The command is updating all existing product structures. First, existing positions are updated, then – linked records.

Product structure properties option opens dialog of the current product structure properties. Modifications are saved in the current assembly file.



You can change name of the current product structure in appropriate field. With a special flag you can mark product structure for be used when exporting assembly to T-FLEX DOCs.

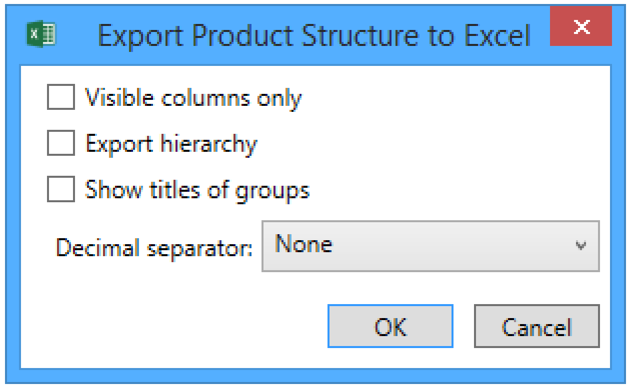
Button **Save properties as type** will save the current product structure properties as a product structure type in *.xml format.

When a new product structure is created, it uses one of the product structure types that defines its properties and behavior. The original product structure type may change later due to various system improvements or customization. If this file with product structure type changes, system reflects this case with a warning message “Properties of product structure differ from properties of type used for its creation. You may update properties from the modified type or ignore this warning”.

You may either ignore this message and leave the original behavior of the product structure or update product structure properties from the modified type using **Update properties** button.

If necessary, you may also reassign product structure properties by reading them from any arbitrary product structure type by using **Read properties from type** button.

> The information about product structure properties can be found in “Product structure types” section.



Use option **Export product structure to Excel** to export the current product structure to Excel format.

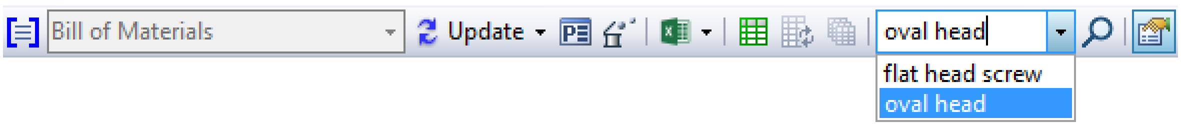
You can specify exporting of only columns visible in the product structure window by setting appropriate flag in the dialog that appears. If the flag is inactive, all product structure columns will be exported.
Export hierarchy allows to export hierarchy of product structure record into Excel.
You can set decimal separator for the exported values.

> If export is used in **Apply product structure representation** mode, the **Show titles of sections** flag will appear. These titles will be added to the additional column of the resulting excel table.

After that you can specify file name, folder and necessary Excel format - XLSX or XLS.
From a product structure you may generate various reports, like for example BOM tables on the drawing page.
To show all reports and BOMs created in the current document use **BM: Report/Bills of material** command. Icon is used to obtain quick access to the command.

Delete product structure. The option allows to delete the current product structure.

The search bar is located in the right part of the toolbar. It simplifies data search inside the product structure.



Input search string or a substring in the field or select from one of the previously used search strings.
The corresponding record will be selected after pressing <Enter> or button. Repeated pressing on <Enter> or button gives next search result.

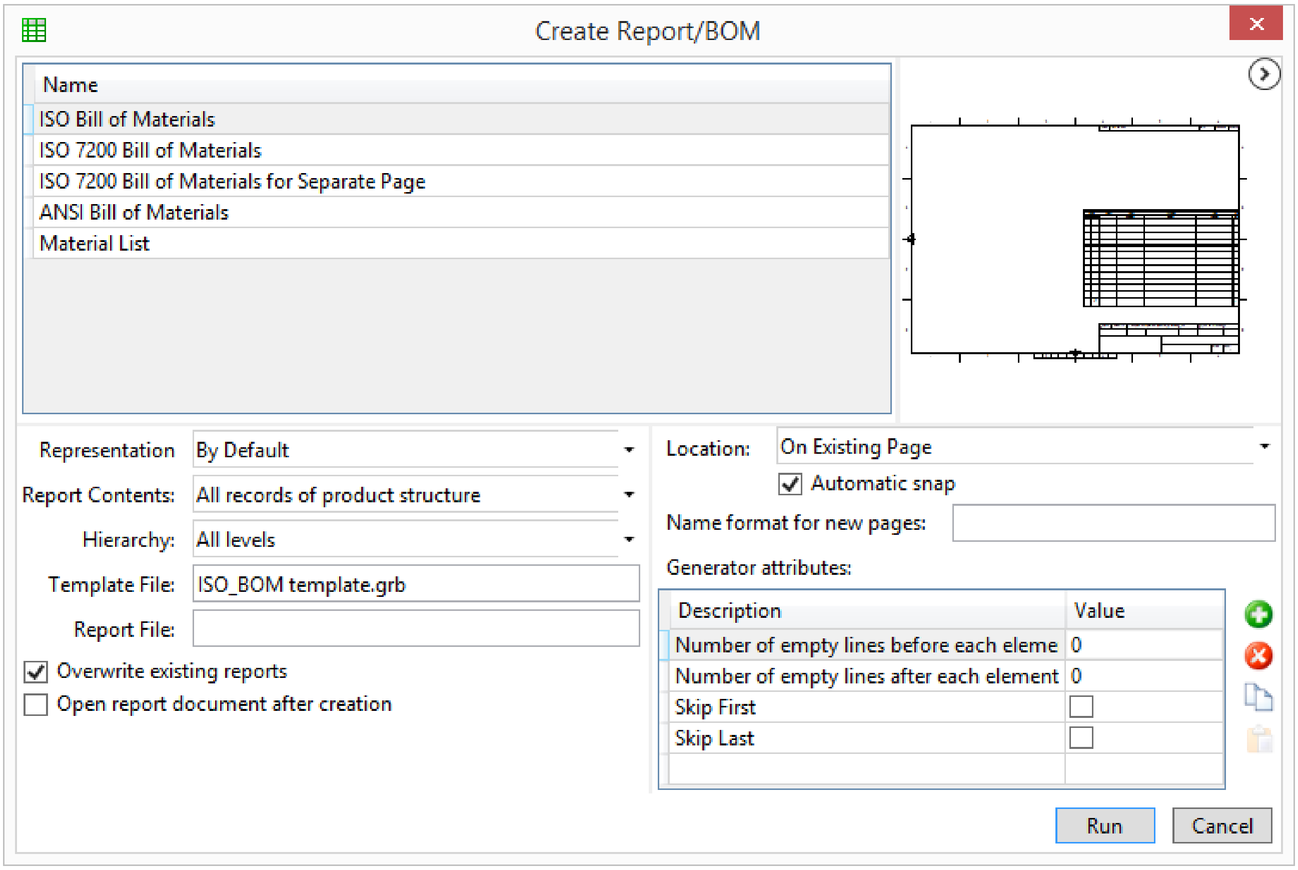
The data will be searched in both visible and invisible columns. When found, the corresponding record will be marked.

Reports

To create a report use option **Create report based on Product Structure.** Window with the list of reports available for the current product structure will appear. This contents of this list is unique for each product structure and originally is taken from the product structure type used for product structure creation.

Preview is located to the left from the list. It can be hidden using button .

Properties of the product structure reports are displayed in the lower part of the window. You can change them for each created report if necessary. Lists of reports and their properties are stored in the product structure properties.



Report creation:

1. Select template for the report with preset rules for filling its fields;
2. Set properties of the selected template (optional);
3. Press [Run] button.

Use option **Update product structure reports**, located on the toolbar, to update the already existing reports.

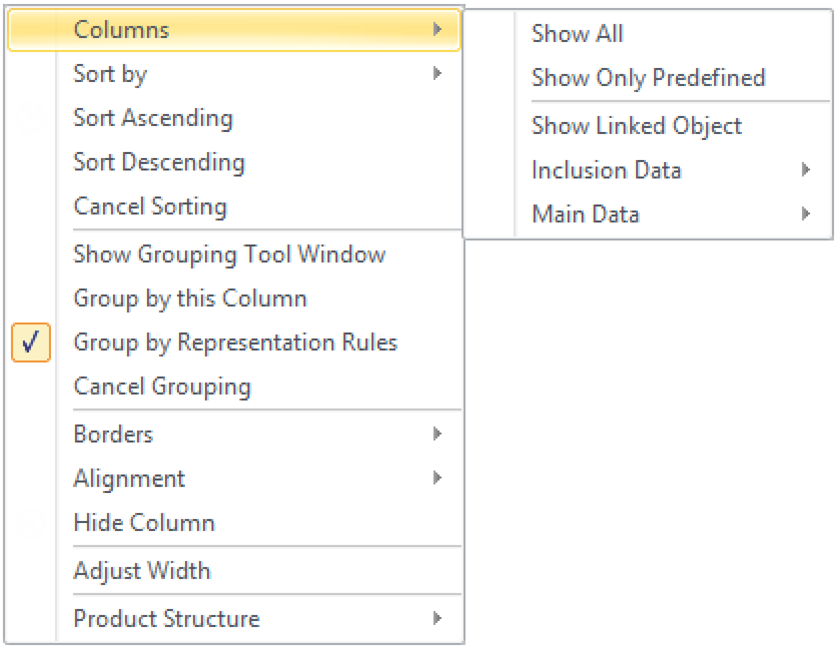
More information about report creation can be found in “Create Report/BOM” section.
More information about report generator properties can be found in “Reports Tab” section.

Commands of the Product Structure Window Header Context Menu

You can find some commands for managing product structure in context menu of header. The context menu is called by right mouse click in the title of the product structure table. It includes the following commands:

Columns. The drop-down list of this command allows you to select columns to be displayed in the **Product structure** window.

A column is displayed in **Product structure** window when the flag is set near the column’s name.



Show all. When option is active all columns existing in the product structure will be shown.

Show Only Predefined. When option is active only columns with the preset flag **Show column in Product Structure window** will be shown. The option can be set on **Columns** tab in **Product structure properties** window.

Show Linked object. The option adds “Linked object” column to the table. The column shows names and icons of the data sources (for example, data of fragments).

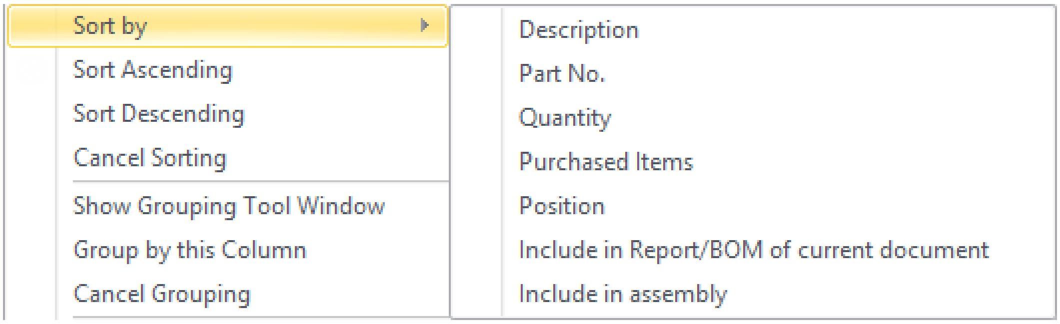
Main Data. Includes main and manually created columns.

You can create a new category for columns. All manually created columns can be added to any category, except “Parameters of include”.

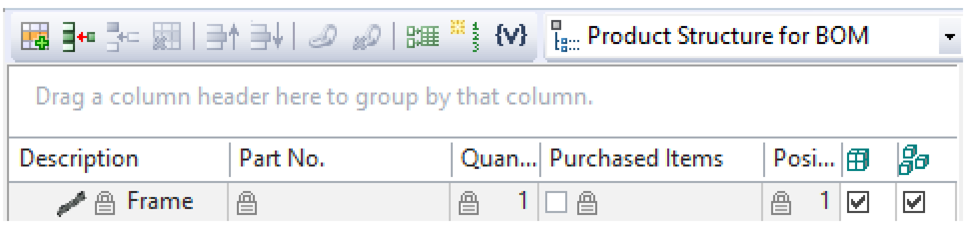
More information about columns can be found in the “Columns tab” section

Inclusion Data. Presents the following columns: **Position, Include in report/BOM of current document, include in assembly.**

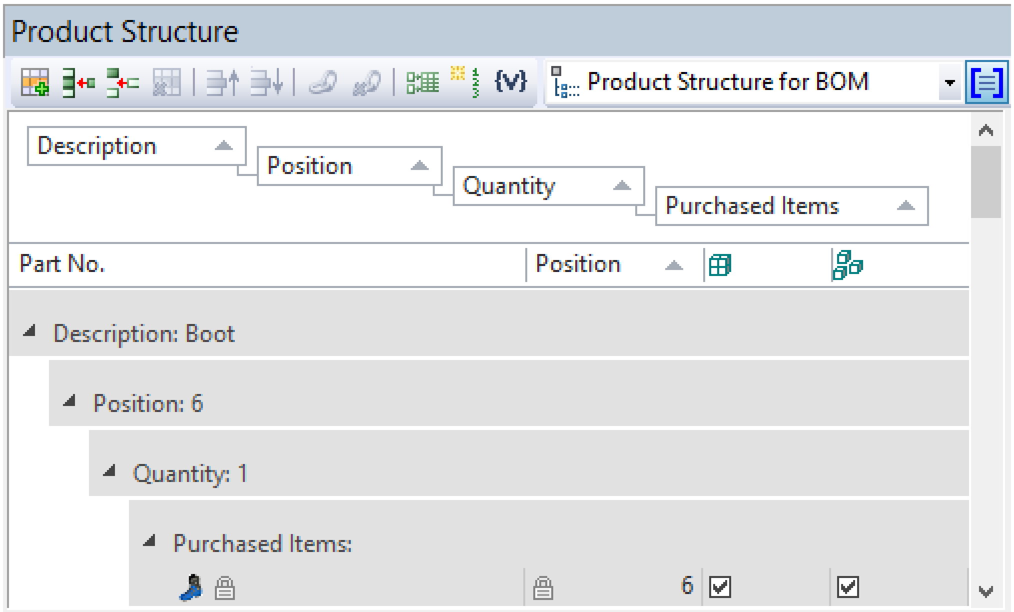
Include in report/BOM of current document and **Include in assembly** columns displays flags that allow to include records into the current document reports and product structure of the upper level assembly.
Sort by. The drop-down list contains the most commonly used columns for sorting records.



Sort Ascending, **Sort Descending**. Sets sorting order for the current column of the product structure.
Cancel Sorting. Cancels sorting rules.
Show Grouping Tool Window. Activates special tool area in the upper part of the window to simplify grouping. To set grouping by column you need to drag its header to this area.



Group by This Column. When this mode is enabled, all records in the product structure will be organized as groups with the same value in the current column. Sequence of grouping is displayed in the upper part of the window. Marker near the header allows to change sorting sequence.

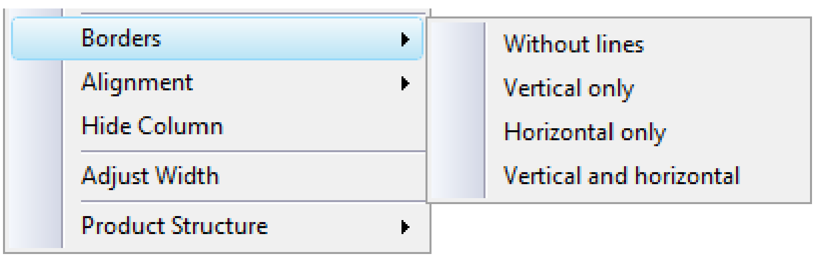


Group by Representation Rules. The grouping mode is set from the product structure properties. Each representation can have its own grouping rules set. Data is not grouped, when the flag is disabled.

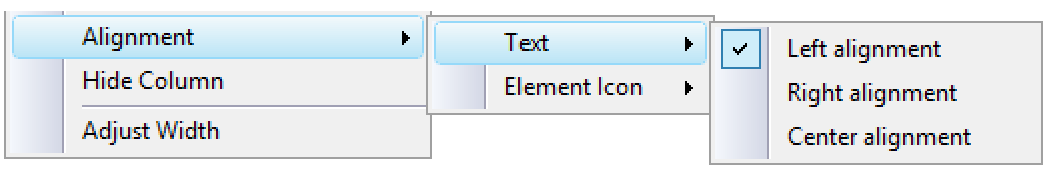
> The option is available only in **Apply product structure representation** mode.

Cancel Grouping cancels usage of all groupings rules set in the context menu or in the representation.

Borders. Allows to manage display of table borders in the **Product structure** window.



Alignment specifies text or icons alignment in the fields of the product structure for the column.



Hide Column. The selected column will be hided in the main window of the product structure.

Adjust Width. The option is adjusting width of columns to show their full names in the **Product structure** window. In case of insufficient space the priority goes to the column which header was used for command activation.

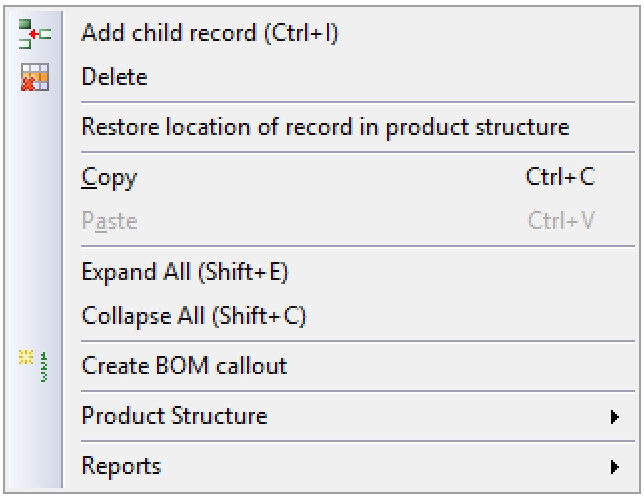
The **Product structure** item provides quick access to several commands that also exist in the toolbar.

Summation results command allows to assign sums of values in columns to variables. Summation of values is specified on the **Sum** tab in the product structure properties.

More information about the summation can be found in section “Sum tab”.

Context Menu of Product Structure Records

Context menu of a product structure record includes options for the product structure and the record itself.



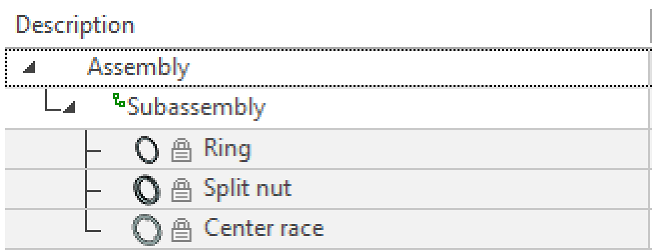
You can create a child record for the selected record using option **Add child record.**

To restore the original position of a record in the **Product structure** window, you need to select item **Restore location of record in product structure.**

Option **Expand All/Collapse All** allows to show/hide grouped records.

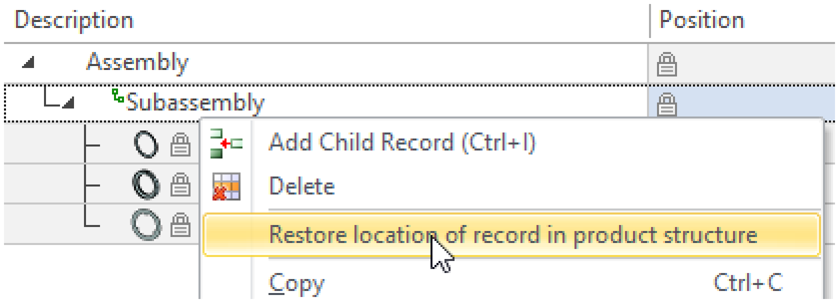
Create BOM Callout. Activates command for creating callout in the current document. After callout creation, you can select another record and continue working with the command.

Restore location of record in product structure. You can restore locations of child records.



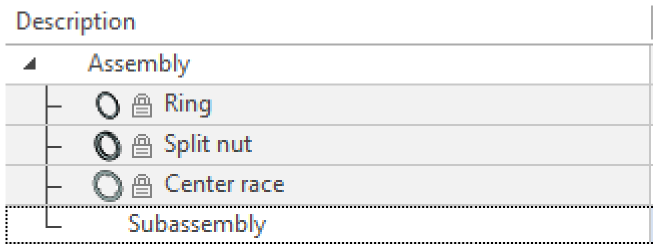
You need to select **Restore location of record in product structure** option to restore the source position of a record. The record appears on the source position after the product structure update.

1



2

3



Copy/paste options:

- You can copy the selected records using the clipboard;
- All records pasted from the clipboard will be added as “created manually”;
- If there were hierarchical relations between records before copying, they are retained after pasting.
- Links between records and variables persist while copying within one document. When you paste records into another document or text, only values will be copied;
- Hotkeys <Ctrl+C>/<Ctrl+V> are available for coping;
- When you paste records in the 2D window, they are inserted as text. Only columns visible in **Product structure** window are copied as text.

Product structure item in the context menu is similar to the same item from the headers context menu.
3D Fragment item appears in the context menu of each fragment. Its drop-down list contains all standard options for working with a fragment.

Reports item allows you to select report template from the drop-down list. The report will be generated after selection. Its properties are preset in the product structure properties.

PRODUCT STRUCTURE TYPES

The product structure type is a file that contains the product structure properties. The product structure type is a prototype for creating new product structures in documents.

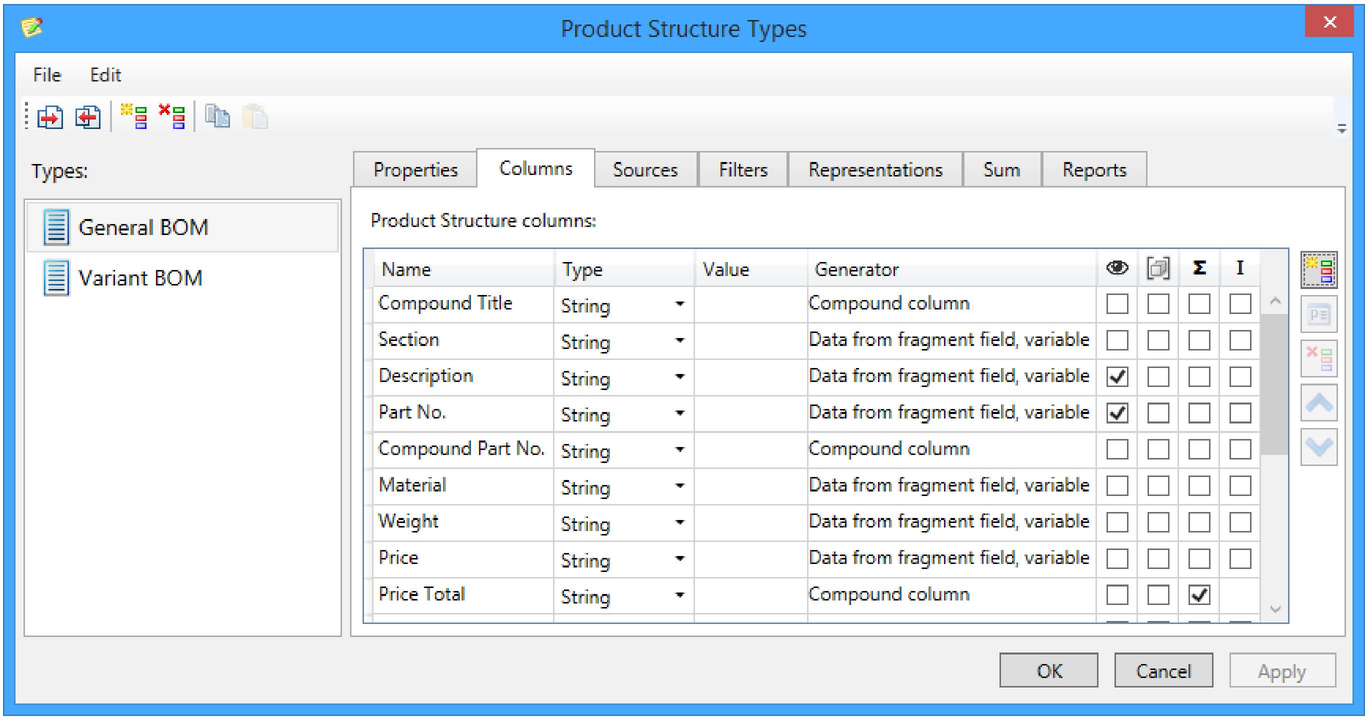
Product structure type command is used to create new product structures types and edit the existing ones.

To call the command:

Icon	Ribbon
	Bill of materials → Options→ Product structure type
Keyboard	Textual Menu
<BY>	Tools > Report/Bill of materials > Product structure type

You can also open the same window for the current product structure using option on the toolbar.

The list of existing product structure types is located in the left part of the window. Tabs with their properties are located on the right side.



Each product structure type is stored in XML file. Settings from such type will be copied to the current document when type is used for creating a new product structure or for an update of product structure. Path folder with the product structure types is set in **SO: Set system options** command on the **Bill of Materials** tab in **Product Structure Types** field.

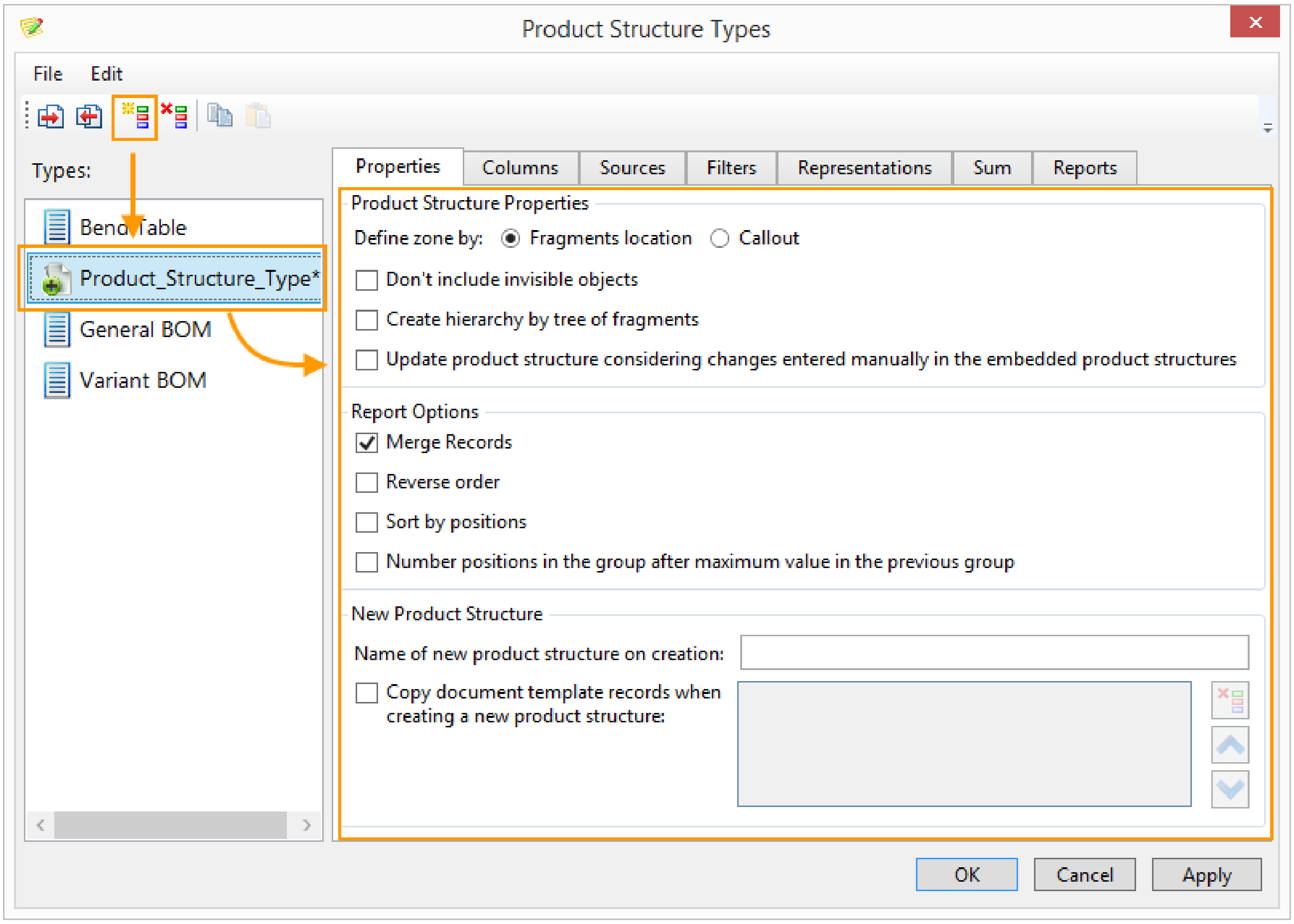
New documents already contain "General BOM" product structure created on the base of "Default product structure.xml" file.

The following icons are located at the top of the **Product Structure Types** window:

 **Open type from file** loads description of the product structure type from the *.XML file.

 **Save type as** saves description of the product structure type into the XML file.

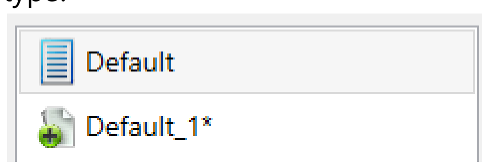
 **New type** creates a new type based on the "empty type". Properties of such a type are not set.



 **Delete type** deletes the current record from the list of product structure types.

 **Copy type** copies properties of the current product structure type.

 **Paste type** creates a new product structure type based on the copied properties – columns, groupings, sorting, representations, etc.

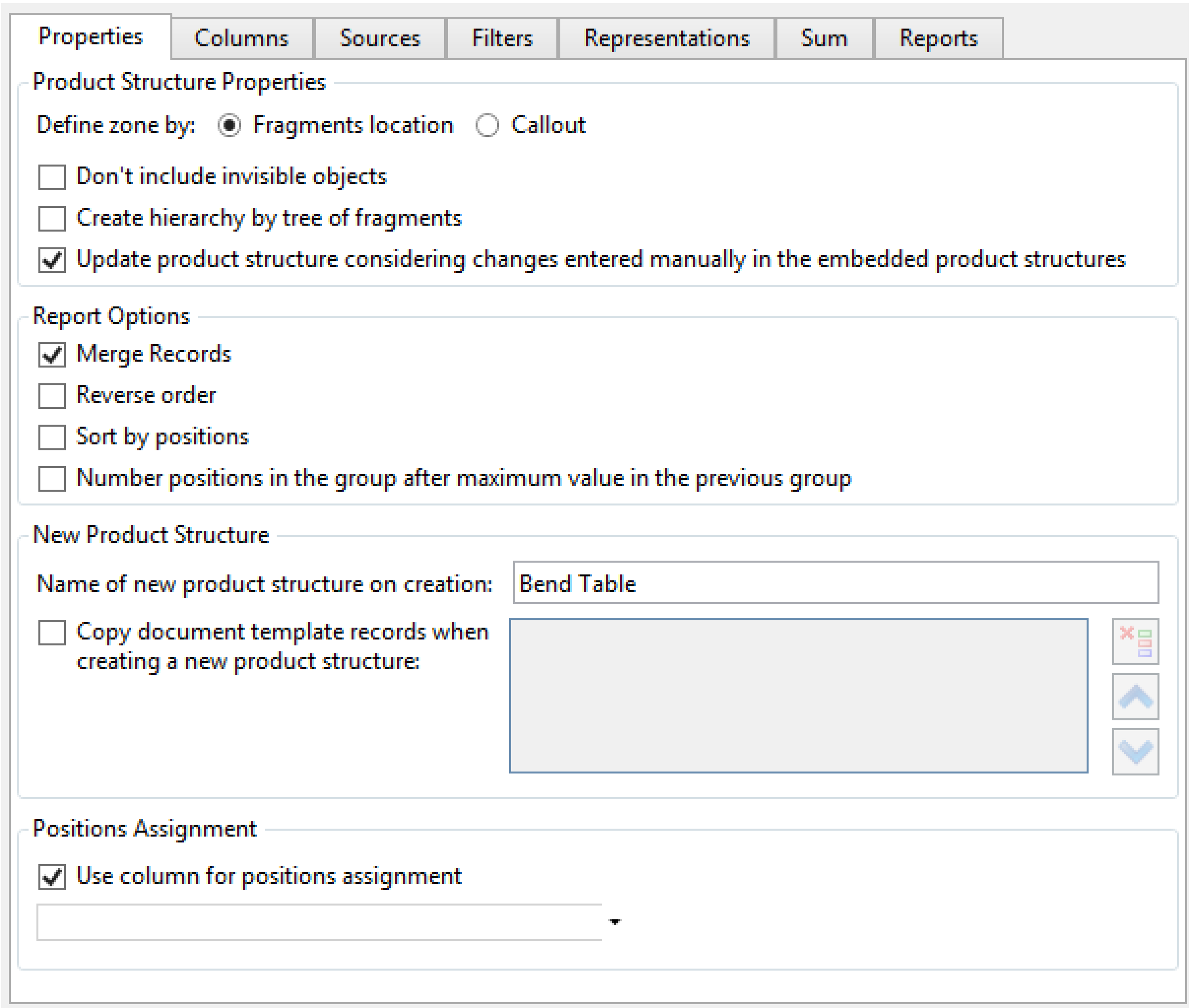


To rename the product structure type, click on its name and fill in text.

Note that if you create a BOM table based not on the product structure report template but on the legacy mechanism based on the BOM prototype file, the prototype properties will be used along with the product structure properties. The prototype properties can differ from the product structure properties. In this case, the created BOM may differ from what you see in the **Product structure** window, because the prototype properties like grouping and sorting will be applied to it after the product structure properties.

Properties Tab

The following properties of the product structure type can be specified on **Properties** tab:



Define zone by. When you work with large assemblies, they are divided into zones to simplify fragments searching. If the **Fragments location** flag is set, the zone value is determined by position of the fragment's bounding box that encloses elements belonging to the fragment.
When the **Callout** flag is set, the zone is defined according to the callout location.
Don't include invisible objects. If the flag is set, only visible fragments will be included in the product structure. Fragments that are invisible on the drawings or 3D models, i.e. hidden using various mechanisms, will be ignored.
Create hierarchy by tree of fragments. When the flag is set, the hierarchy that exists in the product structure of fragment will be retained in the assembly product structure. If the flag is not set, all the records from the fragment's product structure will be placed on the single level in the assembly product structure.

> This option does not affect records that were automatically generated based on fragments or other sources and then manually moved inside the assembly product structure.

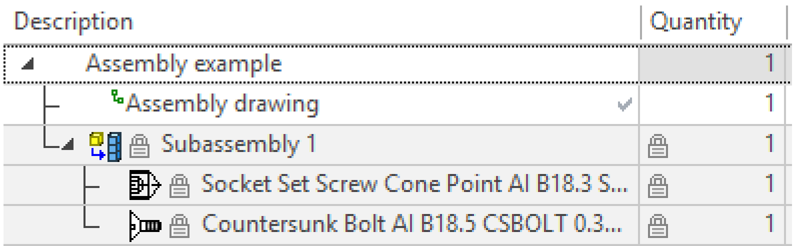
Example:
Product structure "Subassembly 1" contains two item-records for two fragments – Bolt and Screw.

Description	Quantity	Position		
Subassembly 1	1		☐	☑
Assembly drawing	1		☑	☐
Socket Set Screw Cone P...	1	2	☑	☑
Countersunk Bolt Al B18...	1	1	☑	☑

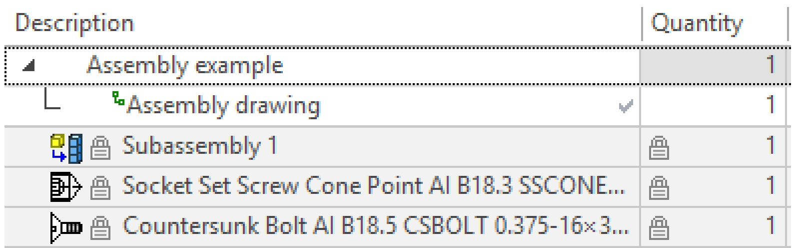
The flag set in the column means that records will be included when the Subassembly 1 is inserted into the upper level assembly.
Next, the assembly "Assembly example" with its own product structure was created using **With embedded elements** mode in option .
The "Assembly example" has the following product structure:

Description	Quantity
Assembly example	1
Assembly drawing	1

After inserting "Subassembly 1" into file "Assembly example" as a fragment and updating the product structure you will receive the following results.
If the **Create hierarchy by tree of fragments** flag is set in the current product structure, then all the records from the fragment becomes "child" for the assembly record of the product structure. Both the hierarchy of assembly example and the hierarchy of Subassembly 1 are taken into account.

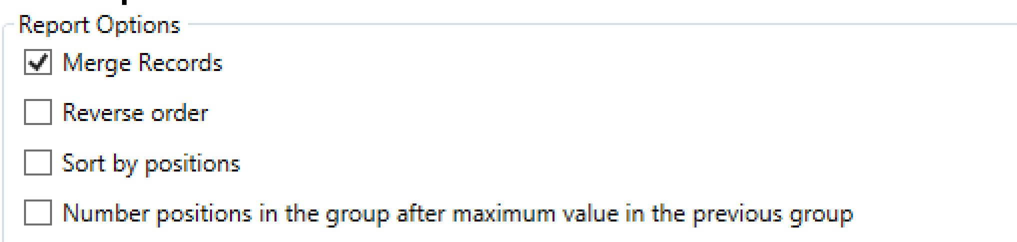


If the flag is not set, all the records from the fragment file will be placed on the same level of the product structure.



Update product structure considering changes entered manually in the embedded product structures. If the flag is not set, records added manually inside items constituting the product structure, will not be included in the current product structure.

Report Options Group



Merge Records. When the flag is set, all records with the same content will be merged. Flag **Ignore parameter when comparing records to merge** for particular columns will be also taken into account.

More information about the parameter can be found in “Columns tab” section.

For example, records with the same descriptions and different quantities can be merged.

Merging is performed in the following way:

A record in the product structure is compared with the following record in the list. If their columns' content is the same, they are merged and comparison with the following record in the list is performed.

For correct merging list of records should be first sorted prior to merging. Sorting is performed by specifying sorting rules.

More information about the sorting rules can be found in “Representations Tab” section.

Option disabled:

Description	Part No.	Quantity	Position		
No group					
Part 1	1	1	1	☑	☑
Part 1	1	1	1	☑	☑
Part 1	1	1	1	☑	☑

Option enabled:

Description	Part No.	Quantity	Position		
No group					
Part 1	1	3	1	☑	☑

Reverse order. When the flag is set, records in the **Product structure** window will be organized in reverse order, starting with the highest position number.

Description	Position
Boot	6
Screw M8x29.6	5
Screw ISO 7380 - M8x10	4
Screw ISO 7380 - M6x10	3
Wheel	2
Frame	1

Sort by positions. If there are manually set values for positions in the product structure, they will be organized by order after setting this flag.

Center race	3
Ring	2
Split nut	4

Without sorting by order

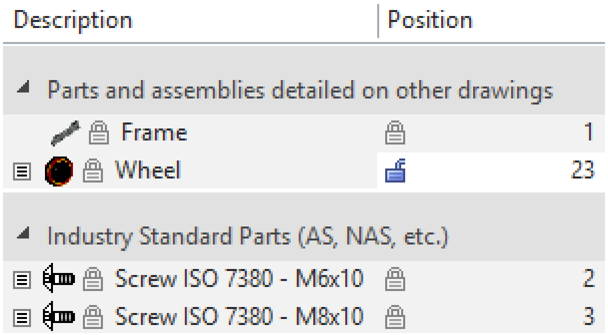
Ring	2
Center race	3
Split nut	4

With sorting by order

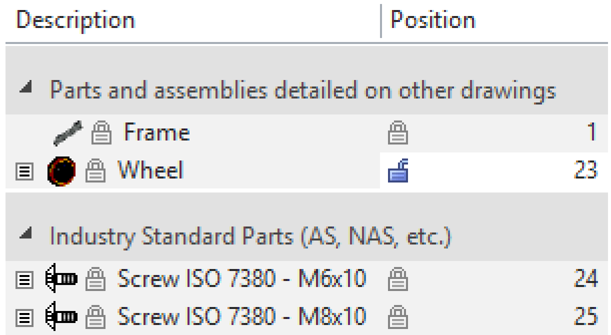
Number positions in the group after maximum value in the previous group.

The option is used to correctly support reservation of strings and positions when some positions are omitted for possible further usage. If there are manually created positions in the sections and their value is higher than position values in the following group, automatic numeration in the following group will begin according to the highest value of manually created positions.

If the flag is not set, the “missed” positions values from the precious section will be used.

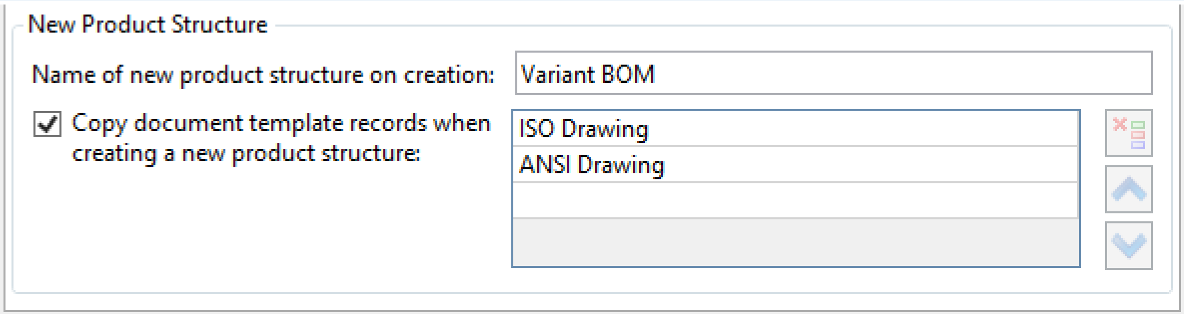


Inactive option



Active option

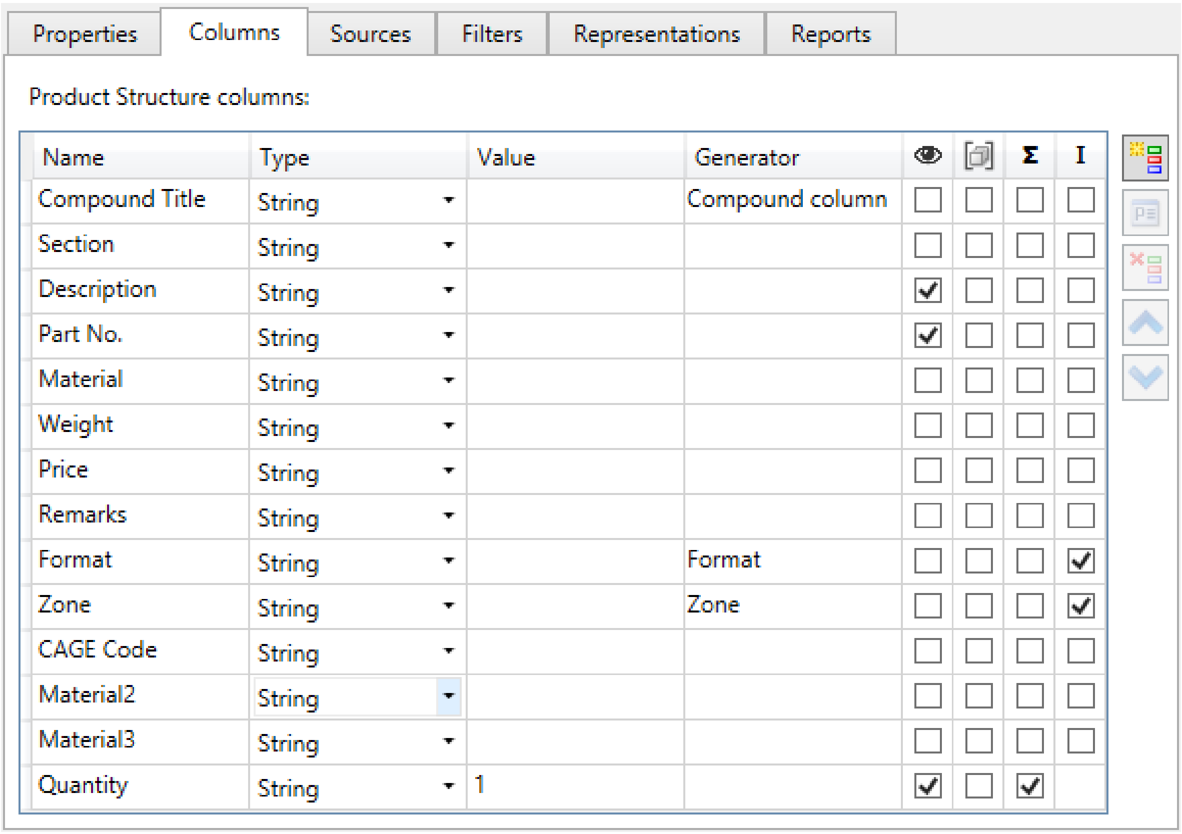
All options are used only when **Apply product structure representation** mode is active.



New Product Structure group.
Name of new product structure on creation sets name to be used when creating new product structures of the current type.
Copy document template records when creating a new product structure. You can enter filenames of prototypes from “Program/Template” folder in the fields. Records from the templates will be added to the new product structure created using the current type.
Use column for positions assignment on the drawing. You can select a column from the drop-down list. The column will be used upon callouts creation instead of the “Position” column.

Columns Tab

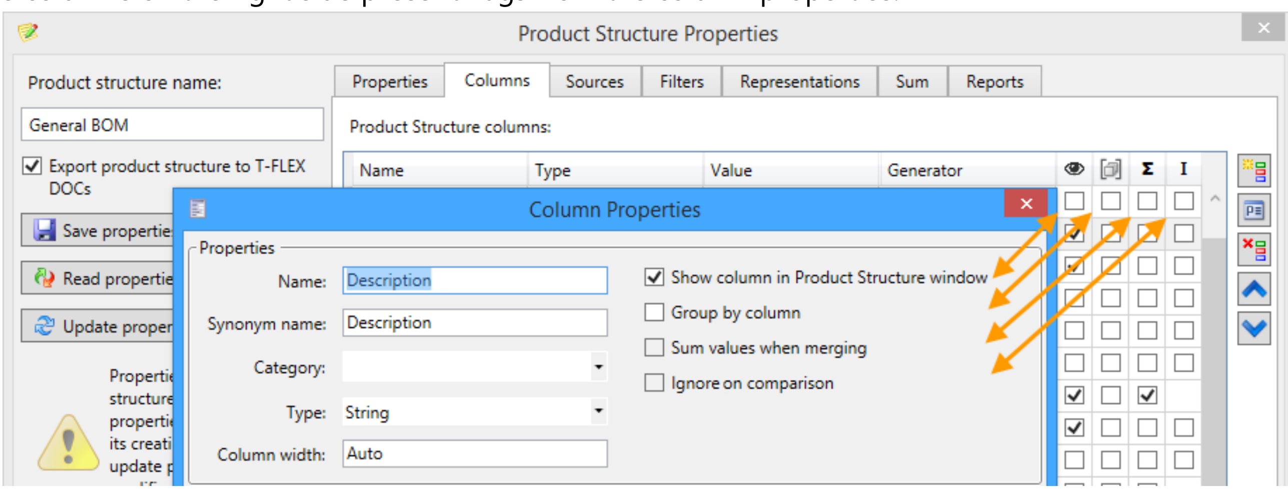
You can add new columns for the product structure type and modify properties for already existing columns on **Columns** tab.



There is a the set of standard BOM columns and their properties in the product structure by default. Buttons

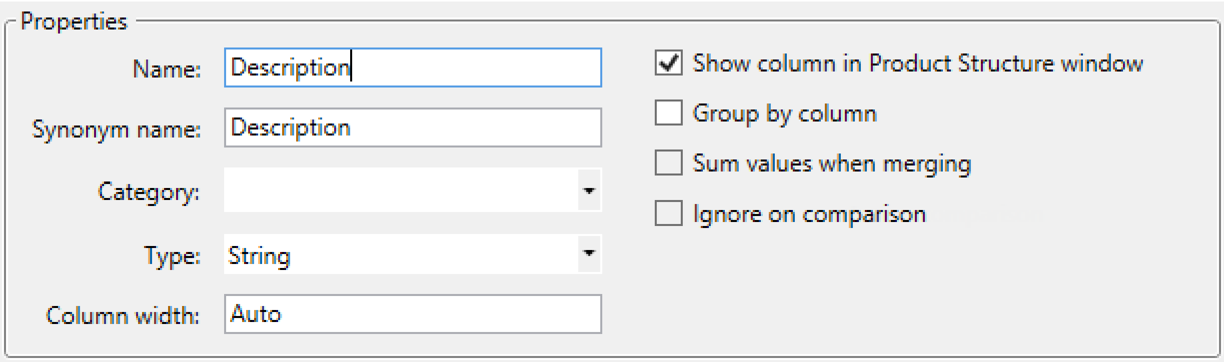
Add , **Properties** , **Delete** , **Up** and **Down** are located on the right side.

The columns on the right side present flags from the column properties.



When you create a new column the **Column properties** window appears:

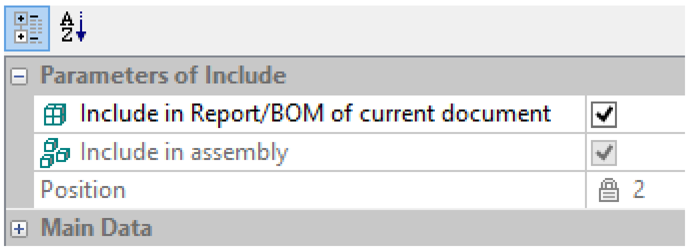
Properties group

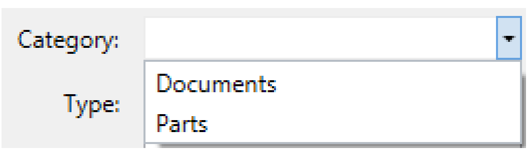


Name. Sets the column name.

Synonym name is used to simplify the entry of the element in the report template. It is recommended to specify a synonym without spaces. For example, instead of the {param name = "Part No."/} element, you can write the {PartNo/} element.

Category. Columns can be grouped into categories. Categories are displayed in the product structure parameters window and in the header's context menu.
The "Inclusion parameters" category is reserved for the system columns "Include in Report/BOM of current document", "Include in assembly" and "Position".

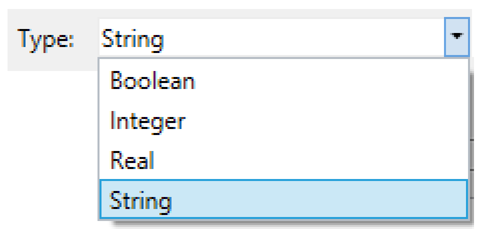




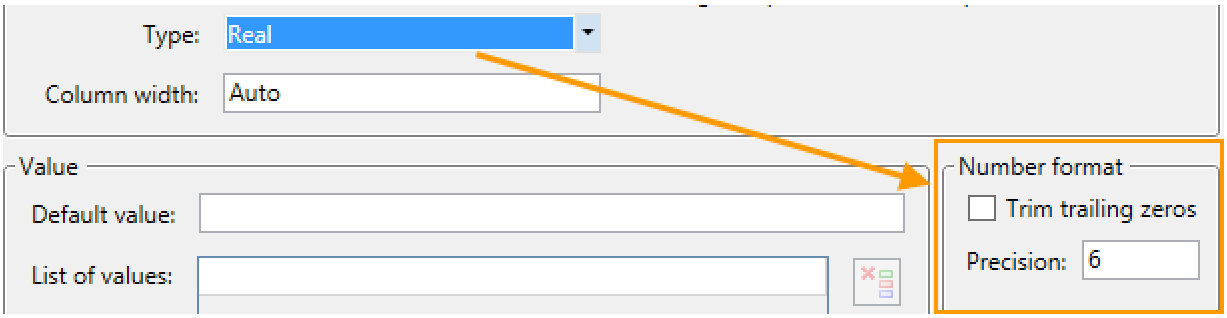
If the column has no category, it will be placed to the "Main Data" category. You can select any other category from the drop-down list. When you enter a new category name, it is added to the list.

Type. Parameter sets type of data in the column.

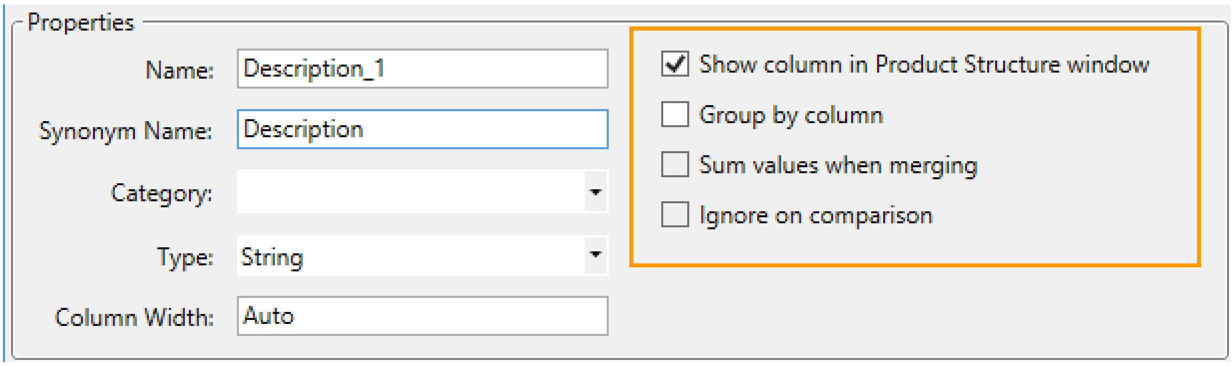
String. The column contains text string.
Real. The column contains real values.
Integer. The column contains integer numbers. When the real number is entered into the column, its fractional part will be omitted.
Boolean. The value is used as a toggle. If the column field is linked with a variable, then the "0" variable value switches the field off (value "False"), other values switch the field on (value "True")



You can specify accuracy of real values for the **Real** type.



Column width. Parameter sets column width in pixels in the **Product structure** window by default for the new product structures.



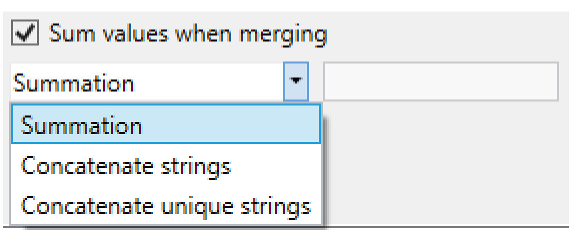
Show column in Product Structure window. When the flag is set, the column is shown in the **Product structure** window. This flag is taken into account only in **Product structure** window.

Group by column. When flag is set, all records with the same value in this column will be merged into groups. This flag is taken into account only in **Product structure** window.

Sum values when merging. When this flag is set, the values in this column will be summed for the merged records, for example the values in “Quantity” column will be summed, if there are similar parts in the assembly.

The flag is taken into account when the column is selected on the **Sum** tab.

Several algorithms for the summation are available:



Summation – the result is the arithmetic sum.

Concatenate strings – the result is the sequential combining of values with separator if necessary.

Concatenate unique strings – the result is the same as above but without duplicated values.

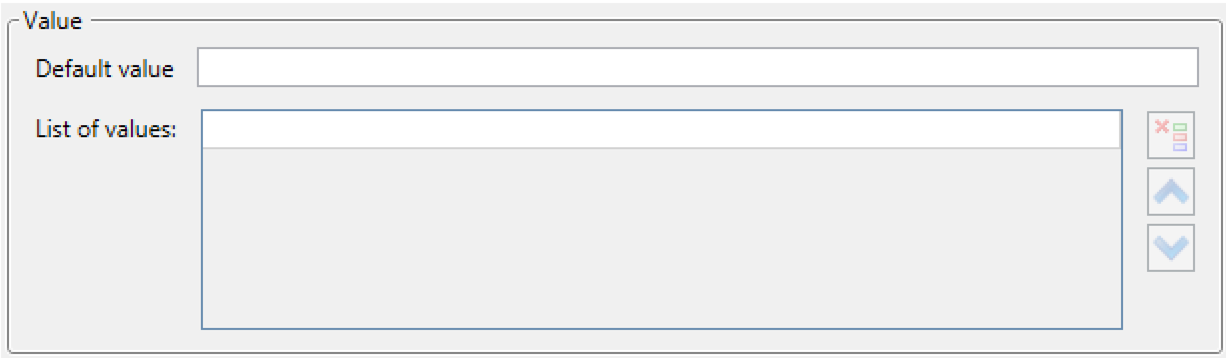
In the last two cases, the result presents one combined string.

In the right field, you can specify any symbol for values separation. E.g. you may generate single string enumerating positions of merged items.

Concatenate strings and **Concatenate unique strings** options only for text strings.

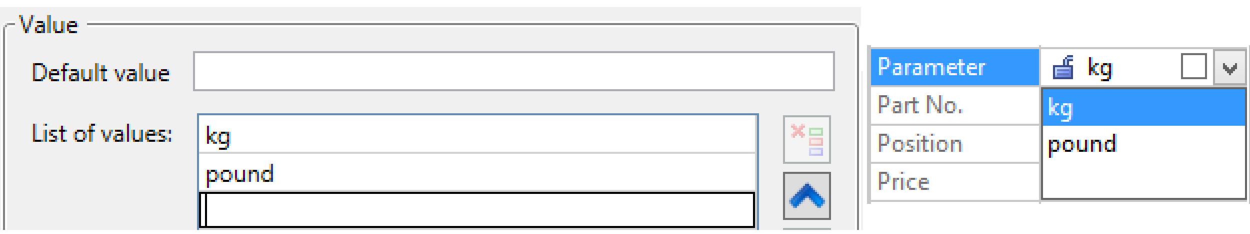
Ignore on comparison. When the flag is set, the content of the column will be not taken into account when the product structure records are compared. I.e. if there are several records, which parameters differ only in this column, these records can be merged - the column will not affect merging.

Value group



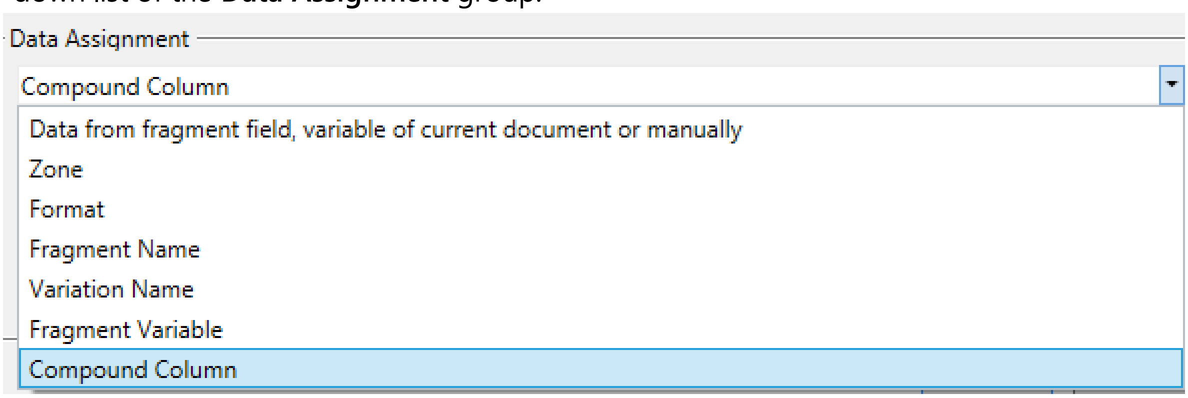
Default value. The entered value will be automatically set in the column data cell for each new manually created record.

List of values. Allows to create preset list of values for a column. When you enter text into the string it will be added to the list after pressing <Enter>. After that you may enter another string to the list. If list exists for a column it will be available for selection in the **Product structure** window.



Data Assignment group

Data can be entered in the column cell manually or automatically. You can select data source from the drop-down list of the **Data Assignment** group.



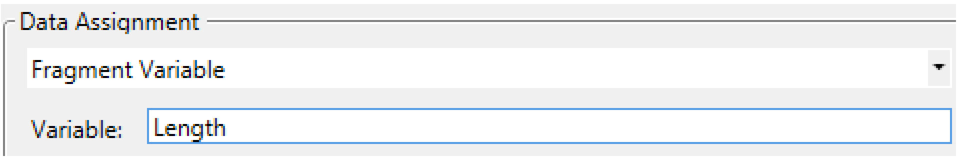
Data from fragment field, variable of current document or variable – data assignment method set by default.

Zone. **Format**. The column is automatically set with zone or format values for fragment-based records.

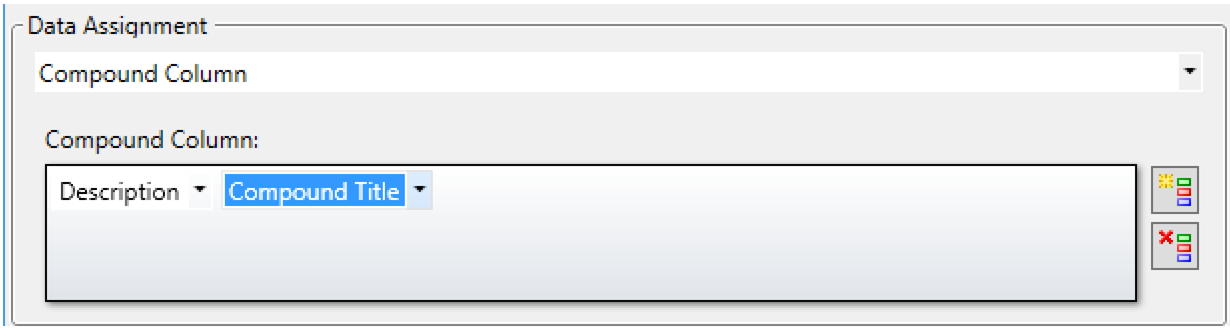
Zone is taken from the assembly drawing and depends on **Getting location zones from fragment** flag.

Format. The drawing format is used for the parts. For subassemblies, BOM format used in subassemblies is assigned.

Fragment Name. Name and folder of the fragment file of the product structure item.
Variation Name. Name of the fragment variation that is used in assembly. You can specify the variation name in **FCE: Edit model Configurations and Variations** command.
Fragment Variable. Fragment variable value will be used as value. You need to specify the fragment variable name. If there is no such a variable in the fragment file, the field will stay empty.



Compound Column. Allows you to compose the value using values of other columns. Data can be taken from any other columns. Mathematical and logical operators can be applied to the columns.

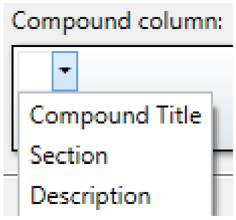


Compound Column Creation

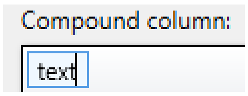
Option adds a new element to the compound column sequence.

The new element will be added after the selected element. The element type can be selected from the drop-down list.

Column. Element-column will be added to the string. You can select the column name from its drop-down list.



Text. Text constant element will be added to the string.

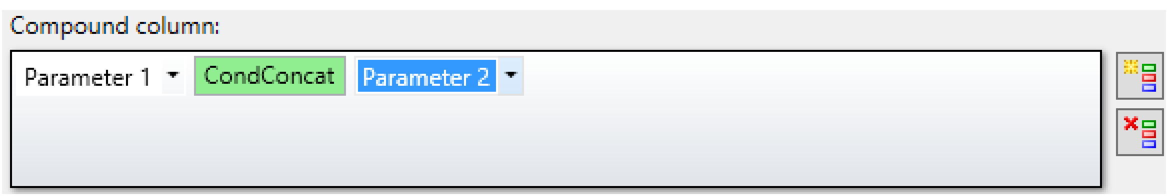


Parentheses. Parentheses allow to set sequence of mathematical and logical operations.

Mathematical operations: «+» - addition, «-» - subtraction, «x» - multiplication, «/» - division, «%» - division remainder.

Operations allow to apply mathematical formulas/relations for the neighboring elements of the string.

Concatenation «Concat»/ conditional concatenation «CondConcat» - merge element/ merge conditional (the operation is performed if both elements are not empty). The result of CondConcat operation for the "Compound column" is shown below.



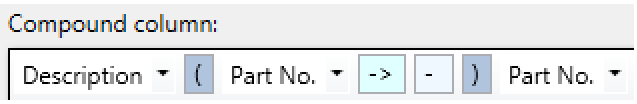
Column	Ctrl + P
Text	Ctrl + T
(	Ctrl + 9
)	Ctrl + 0
+	Ctrl + '+'
-	Ctrl + '-'
x	Ctrl + 8
/	Ctrl + /
%	
Concat	Ctrl + D
CondConcat	Ctrl + S
->	Ctrl + I
&&	Ctrl + 6
||	Ctrl + 7
!	Ctrl + Q
StrSubstr	Ctrl + Y
StrNotSubstr	Ctrl + U
StrStartWith	Ctrl + L
StrEndsWith	Ctrl + O
StrEq	Ctrl + 1
StrNotEq	Ctrl + 2
==	Ctrl + 3
!=	Ctrl + 4
>	Ctrl + >
<	Ctrl + <
Round	Ctrl + R
Floor	Ctrl + W
Ceil	Ctrl + E
Format	Ctrl + F

Parameter 1	Parameter 2	Compound column
abc	def	abcdef
	def	

Conditional operator «->» - if A, then B. The B will be included in the column if A is not empty, not equal "0", is "True". In particular it is useful to use this operator to control separation symbols, e.g. usage of separation symbol "-" in various situations.

Description	Part No.	Name
Part		Part-

We may create condition to not display separations symbol when the column "Part No" is empty.



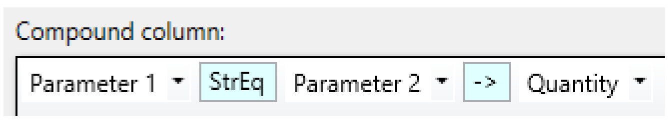
Description	Part No.	Name
Part		Part

Logical operators: «&&» - "And"; «||» - "Or"; «!» - "Not". These operators allow to perform logical operations on the neighboring elements.

For example, with logical operation «&&» you can output the separation symbol only when both "Description" and "Material" columns are not empty.

Operators of string comparison. «StrEq», «StrNotEq» - A equal B, A is not equal B. This operators perform comparison of text values. The result is "True" or "False".

For example, you can set the condition: display value of "Quantity" if the field "Parameter 1" is equal to "Parameter 2".



If the condition is satisfied, the composite column contains data. If not - it will be empty.

Quantity	Parameter 1	Parameter 2	Compound column_1
1	a	a	1
1	a	b	

Operators of string comparison. «StrSubstr», «StrSubstr», «StrStartWith», «StrEndsWith» work in the same way. The only difference is that the selected column is compared to some text constant.

Example: If the "Company Standard Parts" section is set for the record, the "Quantity" column data is displayed in the column.

Operators of numerical comparison. «==» - A is equal B; «!=» - A is not equal B; «>» - A is greater than B;

«<» – A is less than B. The operators perform comparison of number values. The result is “True” or “False”.

Round. Rounds decimal number the nearest integer.

Floor. Largest integer that is not greater than the argument.

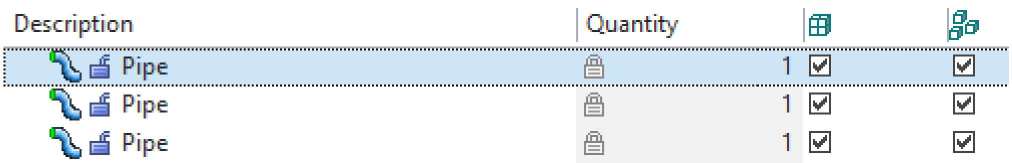
Ceil. Smallest integer that is greater than or equel to the argument.

Format. Converts real number to string. Allows to output values with the specified number of decimal places.

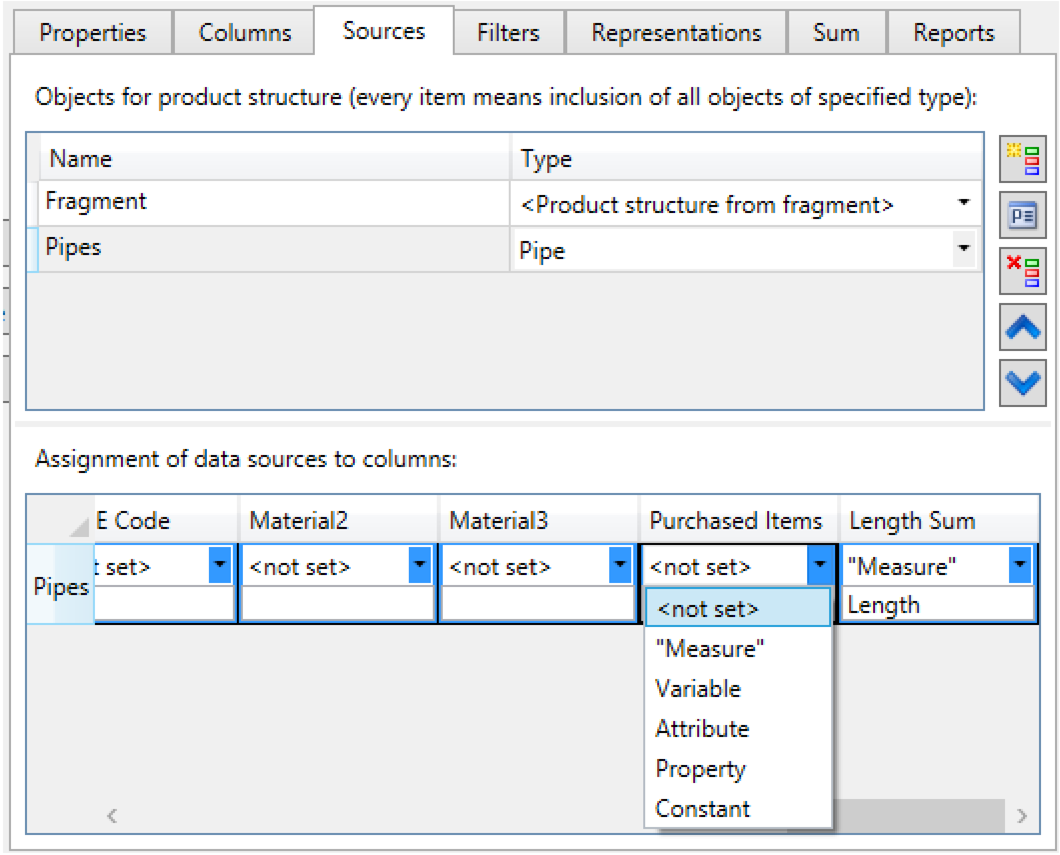
Sources Tab

Along with fragments you can use any objects of drawings or 3D models as items-records of the product structure. You can specify types of objects, included in the product structure, and set their selection conditions in **Product structure properties** window. The selected objects can be used as data sources.

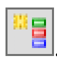
For example, all created pipe operations can be automatically included into the product structure.

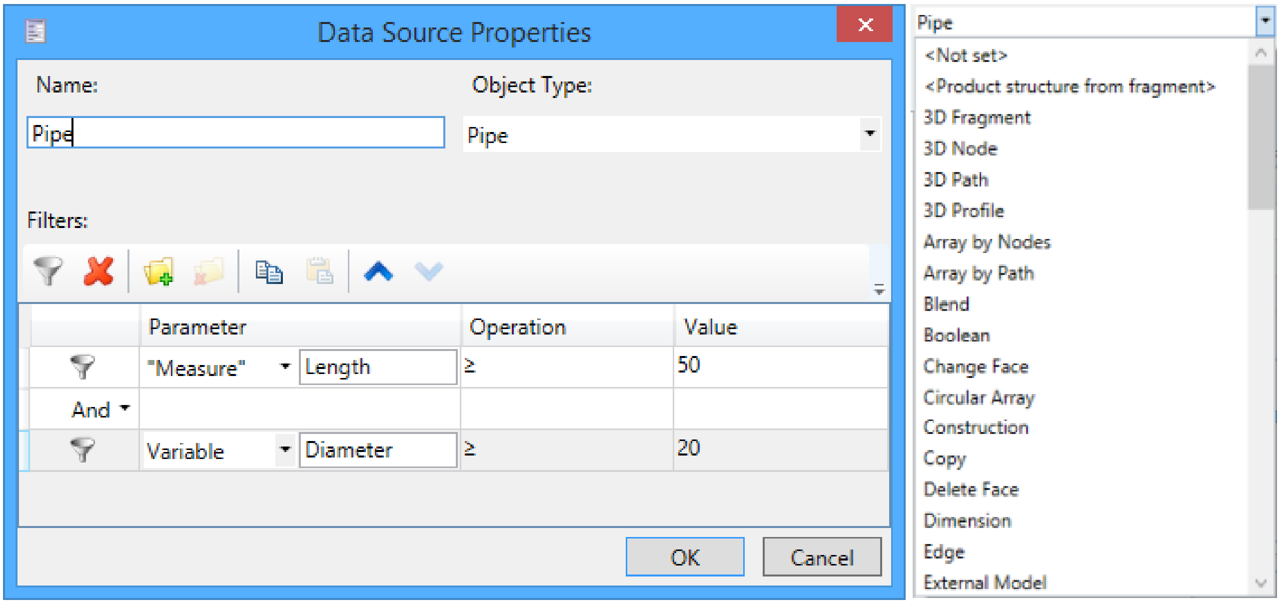


Source tab is used for this purpose. Here you can set sources and data, which must be set for various columns.



<Product structure from fragment> source exists in the product structure by default. It provides receiving data from files of fragments inserted into the current document. All data that is included in the product structure of fragment will be included in the product structure of assembly as well.

To create a new data source use option . The **Data source properties** window will appear. (This window is called with option  for existing strings).



Name. Sets the data source name.

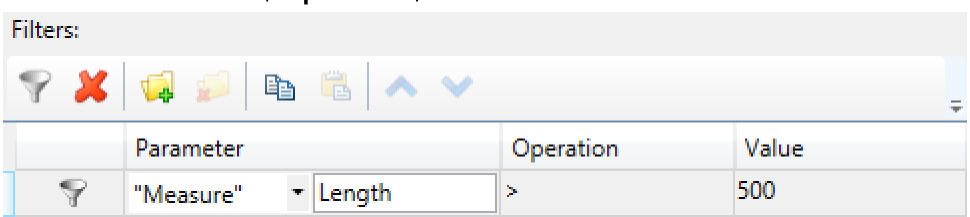
Object type. The drop-down list contains available types of source object.

Filters. Filters are used to set conditions for selecting objects of the specified type. Filter may consist of unlimited number of conditions combined with logical operations "And" and "Or", which will be used in combination. One of the two logical operations may be assigned when you add the second and other next condition to the list.

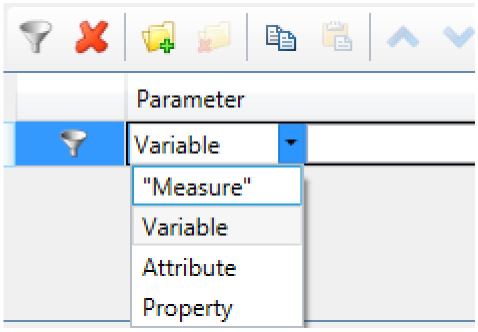
Filters section options:

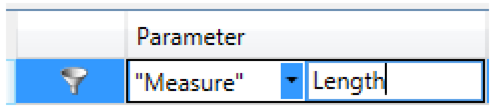
 **Add condition** option creates a new condition in the properties list. Objects included in the product structure will be selected according to these rules.

There are three columns: **Parameter**, **Operation**, and **Value**.



- **Parameter**. The parameter can be set as:
  - measured characteristic of object;
  - value of variable (only for fragments);
  - attribute – internal object property, usually assigned by add-on applications;
  - object property.





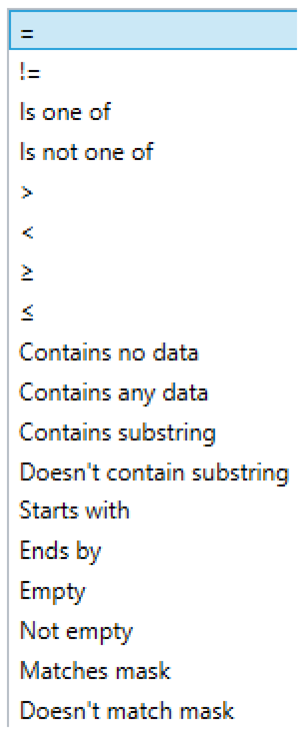
Next to parameter type there is parameter name field.
For **Measure** parameter you need to set the name of one of the object's measured properties. Name of the property must coincide with the name in **PM: Measure** command.
To specify **Variable** parameter, you need to enter name of variable that exists in the fragment document.

Attribute name is set for **Attribute** parameter. The object attribute can be found using **Information** command in its context menu.

For **Property** you to specify the property name from the **Properties** window.

> You can use only common properties of objects for filter. They are displayed in the **Properties** window when several objects of the same type are selected.

- **Operation.** You need to select an operation from the drop-down list. It will be performed for the **Parameter** and **Value** values.



Some notes on them:
Contains no data, **Contains any data** are used for numeric parameters.
Empty, **Not empty** are used for string parameters.
To set additional conditions for adding objects to the product structure is possible to use the mechanism of **Masks**. Mask - is a specialized pattern that when used in filter is applied to the string values of objects parameters. It allows to include or vice versa do not include objects into the product structure depending on the string content. To set a mask it is necessary first to choose one of the Operation option **Matches mask** or **Doesn't match mask**, and in the **Value** data field set the mask itself by a combination of symbols.

The following mask symbols are allowed:

Symbol	Description	Example
%	Any string with zero or more symbols	«A%» - returns all values that starts with A
_	Any single symbol	«A_» - returns all values that has two symbols length and starts with (A1, A2, Aa and so on)
[ ]	Any single symbol within the specified range ([a-f]) or set ([abcdef])	«[12][0-9]» - returns all values from 10 to 29
[^]	Any single symbol not within the specified range ([a-f]) or set ([abcdef)	«[^0-9]%» - returns all values that do not start with a number.

You can manage order of logical operations in filters using brackets.

- **Value**. In this field, you can enter value that will be compared with the selected parameter.

Delete condition. Option deletes the current condition from the list.

To create a group of conditions you must select several filters in the list using <Shift> or <Ctrl> and press button **Group conditions**. Grouped conditions will be considered as a single operand in the list.

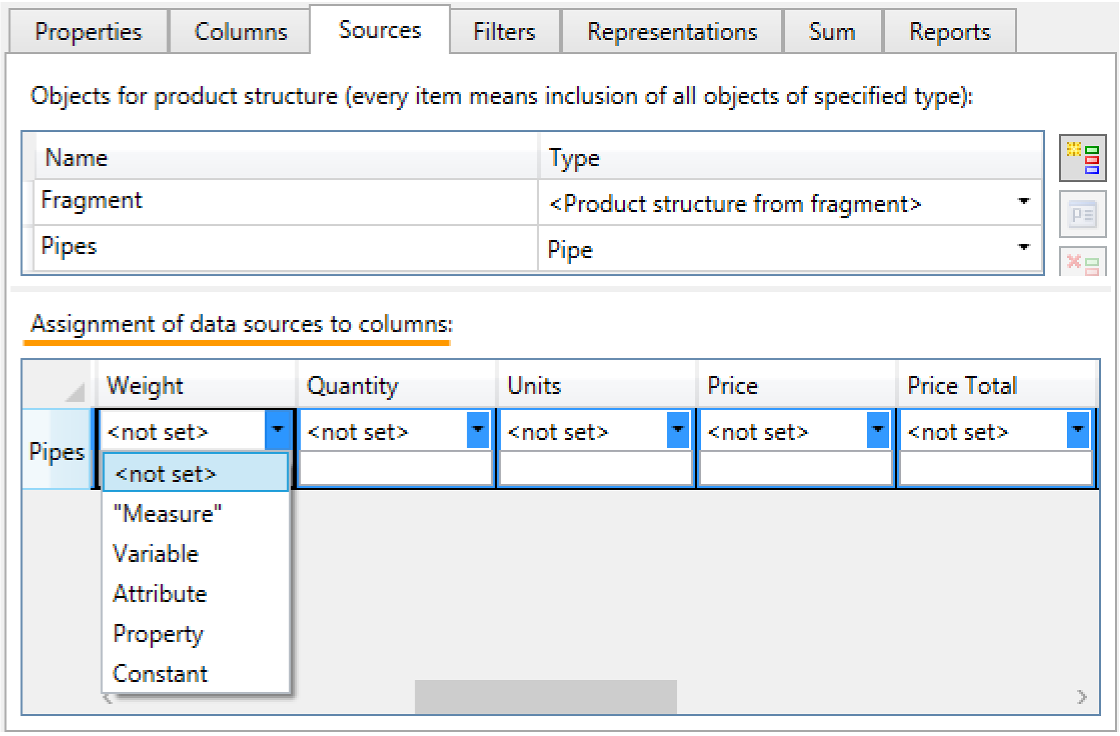
To cancel grouping, you need to select its header and press **Ungroup conditions**.

Copy and **Paste** options use the clipboard for conditions creation.

Up and **Down** options allow to change order of the created conditions.

Data Source Assignment

You can set data that will be set to column data cells of product structure for each data source. **Assignment of data sources to columns** section is used for this purpose.



A full list of columns is available in the section. You can specify assignment condition for each of them.

There are two strings available. You can use them to set the conditions. The data receiving method is specified in the upper string. It is similar to the **Parameter** field from sources filters.

The data can be:

- result of object measuring;
- variable value;
- attribute;
- object property;
- constant.

You can specify the name in the second string.

To specify **Measure** parameter you need to set the name of one of the object measured properties (name of the property must coincide with the name in **PM: Measure** command).

To specify **Variable** parameter, you need to enter the name of variable that exists in the fragment document.

Attribute name is set for **Attribute** parameter. Object attribute can be found using **Information** command of its context menu.

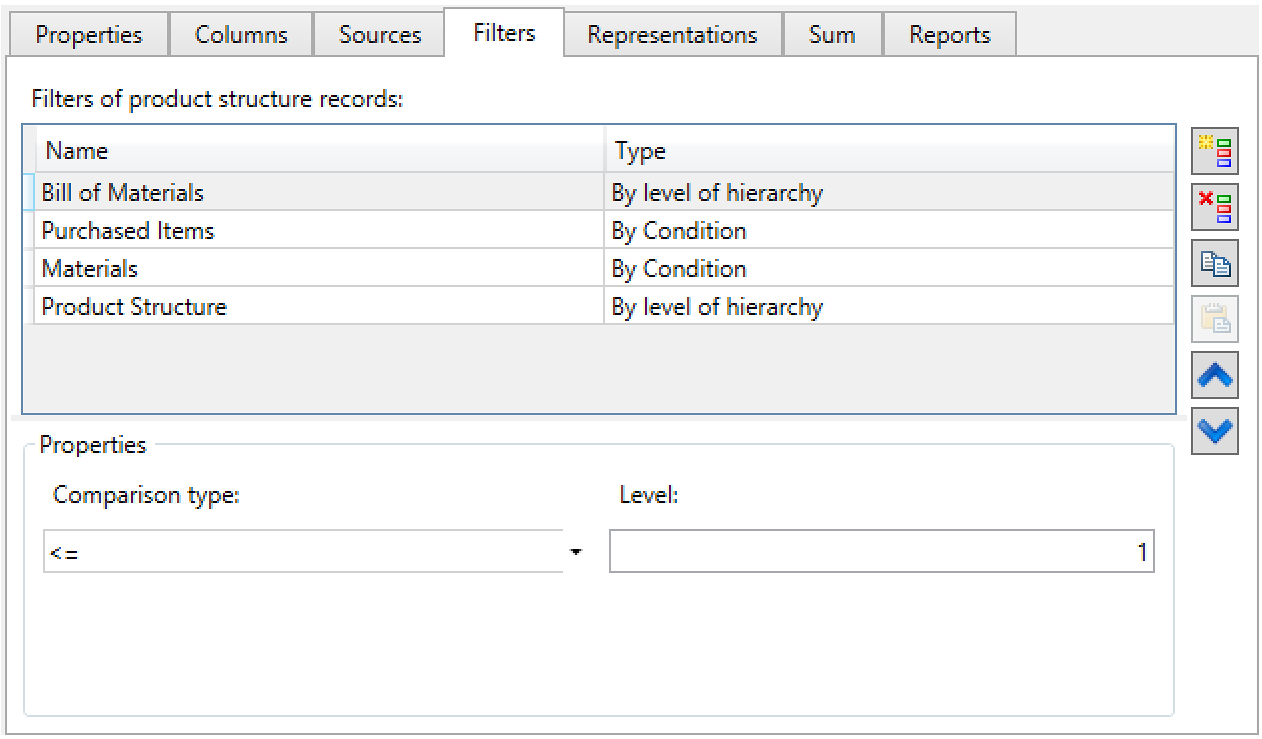
To specify **Property** parameter, you need to set property name from the **Properties** dialog box of corresponding object.

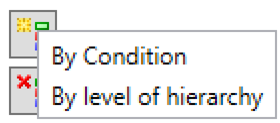
Constant parameter defines a constant value that will be used for all records in the selected column.

Filters Tab

Filters specify data arranging conditions for representations. You can specify various filters that can be used in different representations. For example, from the whole set of product structure records you can select only records about materials to receive the corresponding report.

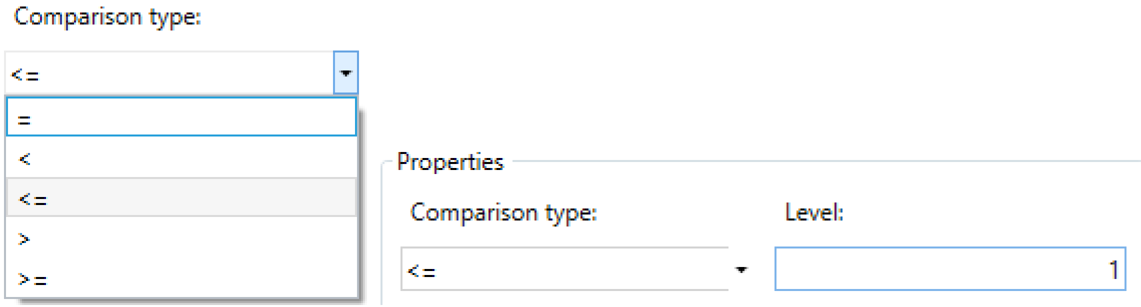
You can read about representations in "Representations tab" section.





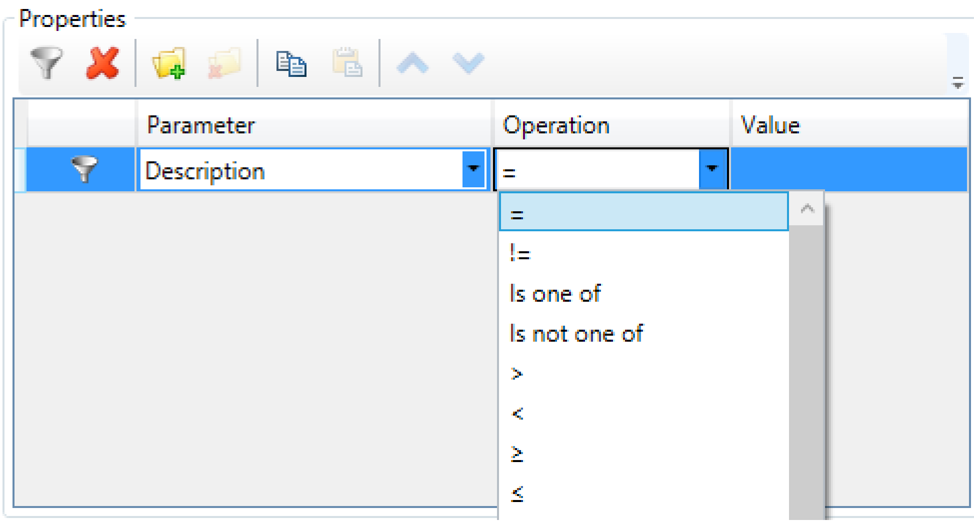
When creating a new filter you can select one of two types: **By Condition** or **By level of hierarchy**. You can change filter name by pressing on its name.

- **By level of hierarchy.** This filter helps to define level in assembly hierarchy for which to gather product structure data.
  You can specify **Comparison type** and **Level** for this filter. The levels values begin from "0" (the product is on a zero level). For example, filter "<= 1" will gather data in the assembly file and the fragment files of the first level.



- By Condition. This type allows to specify conditions using content of product structure columns. Filter creation is similar to the previously described data source filter creation. The only difference is that the **Parameter** drop-down list contains columns of the product structure.

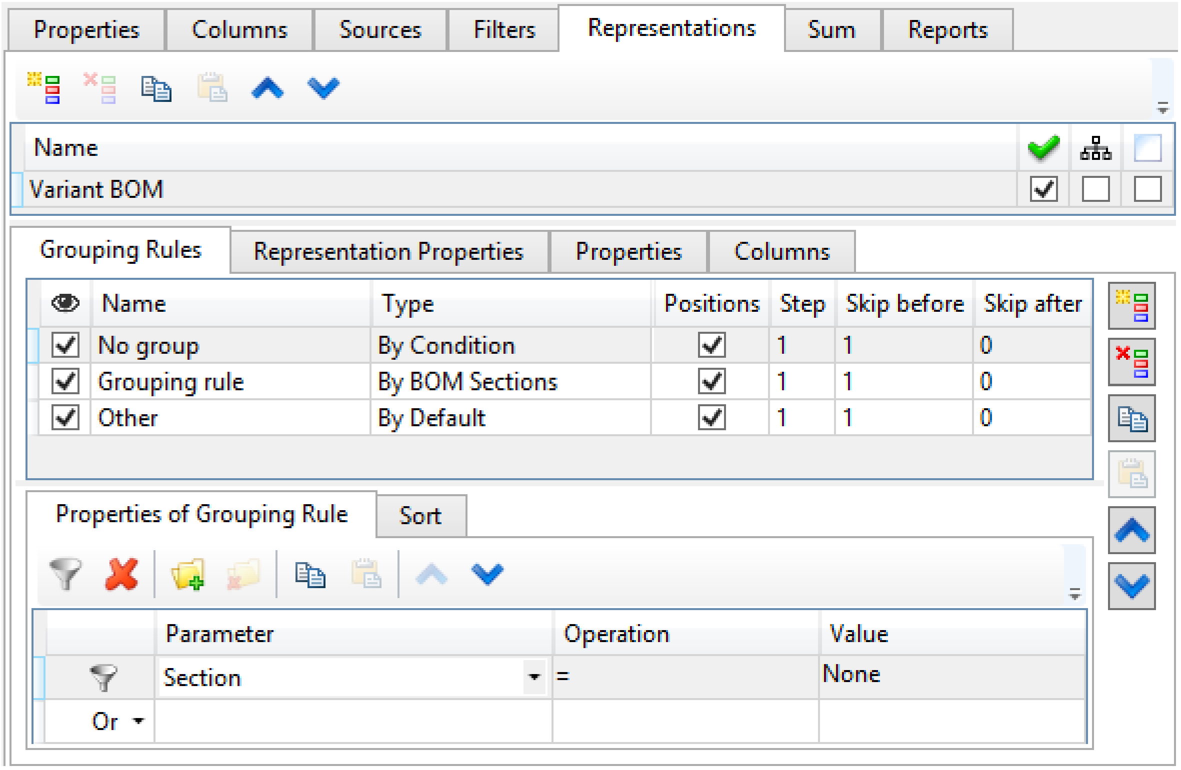
You need to select a column and specify condition for the values selection.



Representations Tab

Representation is a set of rules (for example, grouping and sorting) for displaying **Product structure** data in various forms and further usage in generating report tables. One product structure may have several representations that correspond to different report tables.

> All the conditions specified on **Representations** tab work only in  **Apply product structure representation** mode.



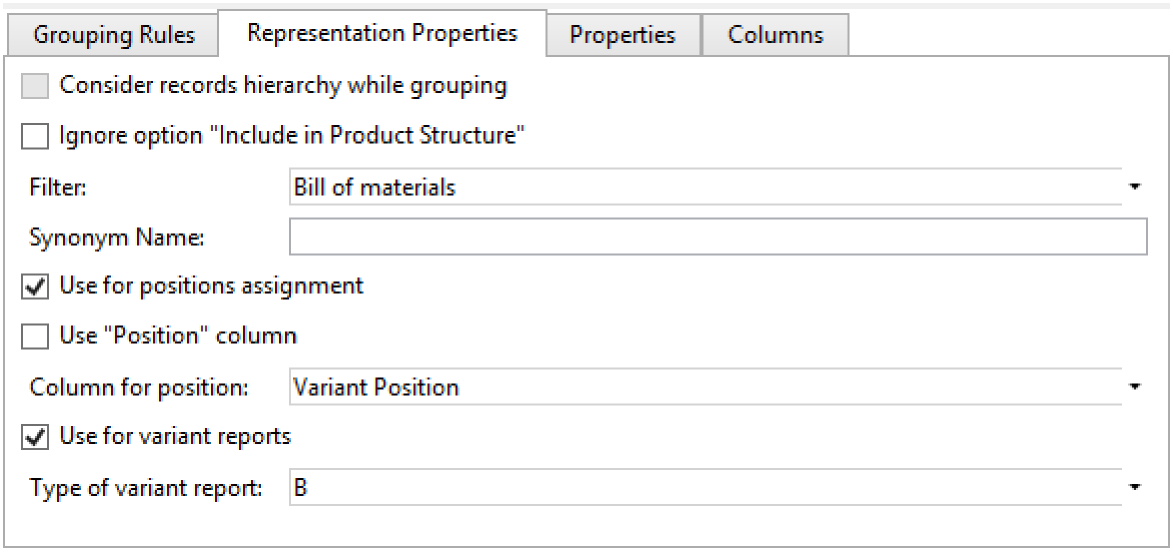
All existing representations are displayed in the corresponding field.

Properties of each representation are managed on **Grouping rules** and **Representation properties** tabs. The **Grouping rules** are managed on **Properties of grouping rule** and **Sort** tabs.

The following flags are located near representations names: **Use for positions assignment** , **Consider records hierarchy while grouping** , **Use position column** .

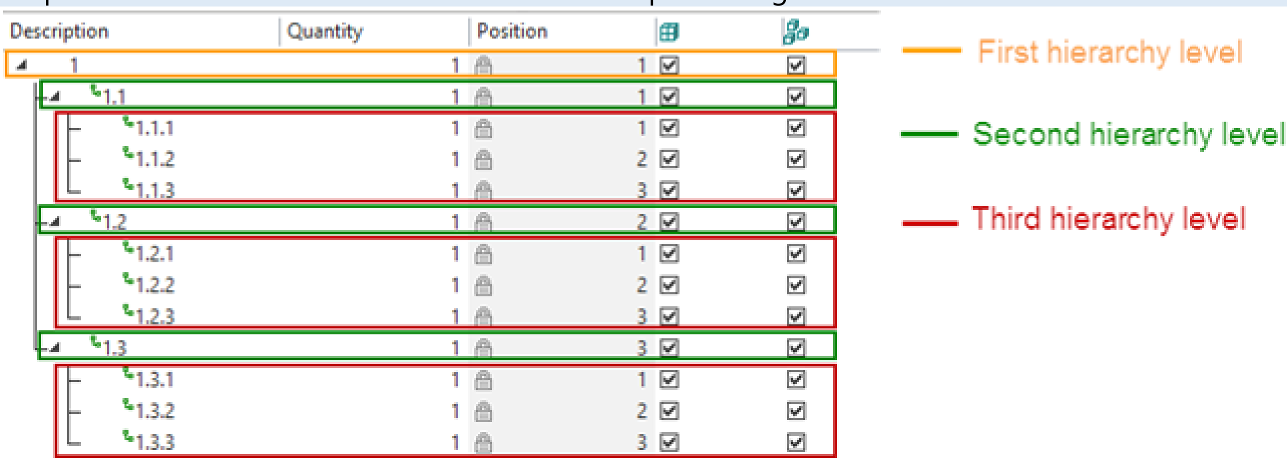
Representation Properties

You can specify common representation properties on **Representation Properties** tab.



Consider records hierarchy while grouping. If the flag is set, the grouping will be performed separately on each level of hierarchy. It allows to assign positions and receive reports for different “branches” of the product structure tree if necessary. The flag is not available when **Use for variant reports** flag is active.

> Important! It is required the topmost record of the product structure tree to be included into reports of the current document for correct data processing in this mode.



For example, you want to create BOM table only for the “1.1” subassembly. For this purpose, you need to select it in the product structure and create report with **Only selected elements** and **Only first level** parameters.

> More information about reports creation can be found in “Reports Tab” section.

Ignore option “**Include in product structure**”. If the flag is set the representation will show records with disabled **Include in product structure** option.

Filter. Specifies the filter set on the **Filters** tab that will be used for the current representation. The drop-down list contains filters from **Filters** tab.

Synonym name. You can specify a synonym name for the representation. The synonym name may be helpful if you plan to output summation results in **Summation results** window or link them with variable.

More information about the window can be found in "Summation result" section.

Use for positions assignment. If the flag is set, positions for representation are assigned automatically. Otherwise, positions will not be assigned for this representation Note, that this flag will be used in combination with analogous flags for automatic position assignment set for each grouping rule if any exists.

Use "**Position**" **column.** If the flag is set, the specified position is entered into "Position" column. If the flag is not set, the position is entered in the column specified in **Column for position** field.

Positions will not be displayed in the product structure if the flag is not set and the column name is not specified.

If flag **Assign positions automatically** is not set, this flag will be ignored.

Use for variant reports. When the flag is set, the representation data is processed in a specific way to generate a variant report. Report type can be selected from the drop-down list of **Type of variant report** field.

More information about variant reports can be found in "Variant report creation" chapter.

Grouping Rules

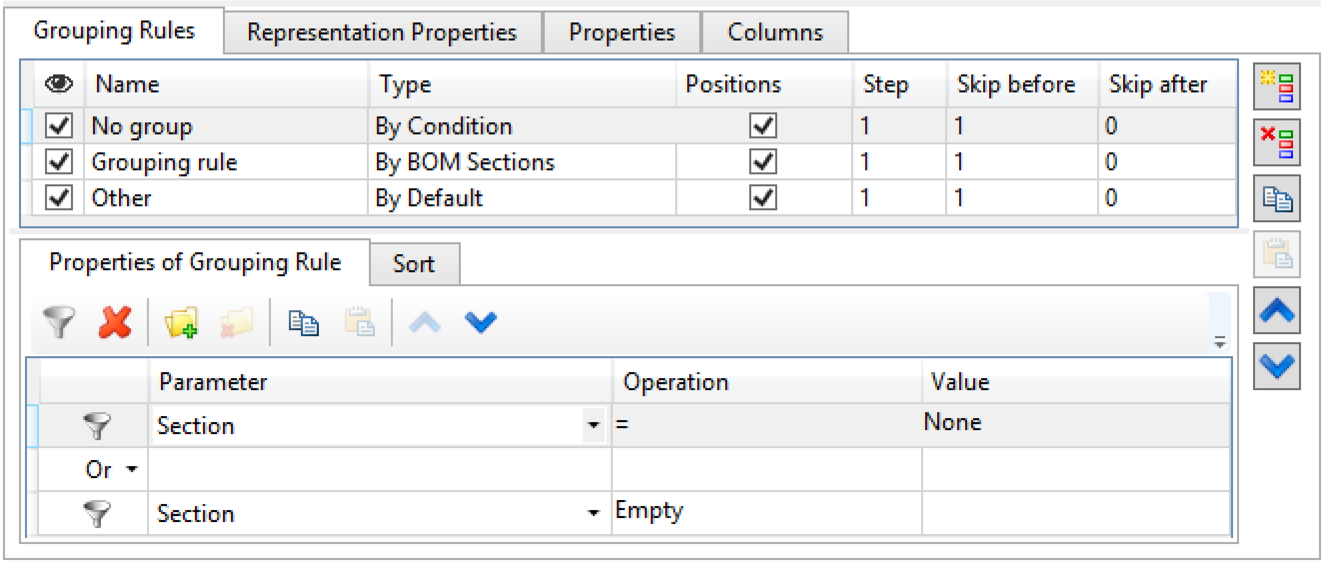
Rules for grouping records in the product structure window are specified on **Grouping rules** tab.

You can set name for a grouping rule. For each grouping rule there is also possible a set of positioning options: **create positions for group**, **positions increment**, **number of skipped positions at the beginning** and **number of skipped positions at the end of the group**.

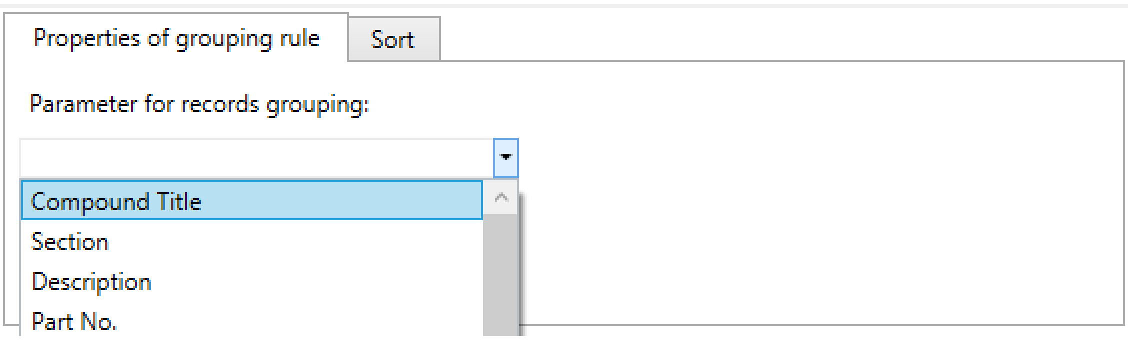
Every grouping rule has its type. According to the grouping rule type, you may set additional options on **Properties of grouping rule** tab.

The following **types** are available:

By condition. You may specify one or several conditions for the rule. Conditions are similar to the filters from the **Filters** tab.



By parameter. When you create this type of grouping rule, you can select column from the drop-down list. Records will be grouped according to the values in this column.



> Values from the column will be used as groups names if the flag **Consider records hierarchy while grouping** is not set.

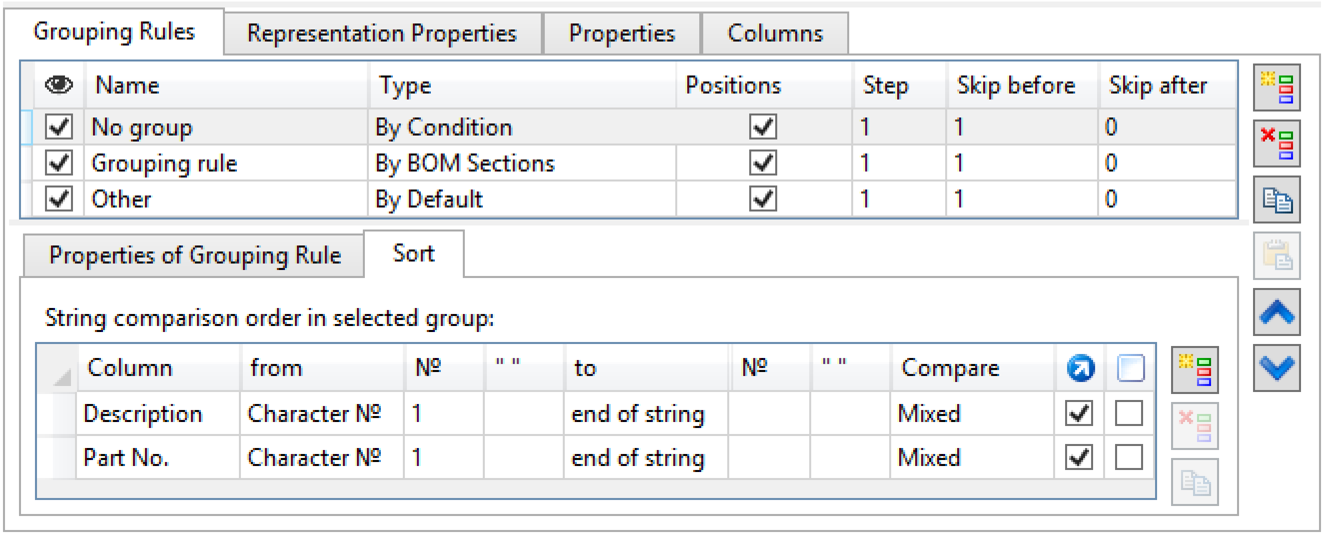
By BOM sections. Product structure records will be grouped by BOM sections. Each section is included into a separate group. BOM sections are defined in **BG: Edit BOM sections** command.

By default. This type merges all records into one group.

> If there are several grouping rules in one representation, they are applied sequentially. Initially the first rule that distributes records to the groups is applied. Some of the records may be not included in any group after that. The second rule is used for these records. The following rule is used for the remaining “Free” records and so on.

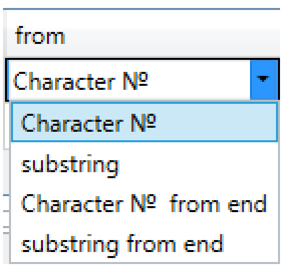
Sorting

Sorting can be specified for each grouping rule (group) on the **Sorting** tab.



To create a new sorting rule at first you need to select a grouping rule. Then, you need to create a sorting rule using icon  **add**. You can select column name in the **Column** field from the drop-down list. The data will be sorted by this column. After that, you need to select limits of comparison region for the columns.

From – defines the beginning of the comparison region.



You can select one of the following values from the drop-down list:

character № - the character serial number. For example, three will mean that comparison region will begin from the third character of the selected column string.

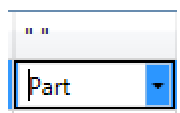
substring – the serial number of the selected character sequence occurrence, for example, from the first substring “Part” occurrence.

character № from end – the character serial number from the end of the string, for example, the third char from the end.

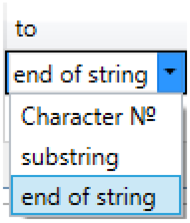
substring from end – the serial number of the selected character sequence occurrence from the end of the string, for example, the first appearance of “ISO” substring from the end.

№ - the serial number of the string or substring.

“ ” – the sequence of substring characters here.



To – defines limits of comparison region.

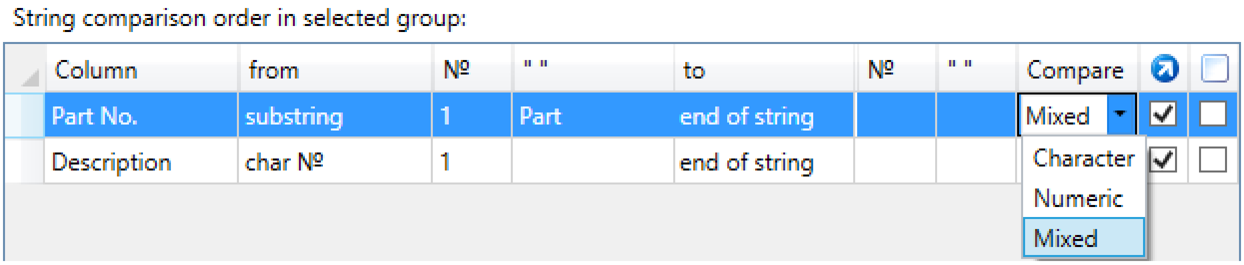


character № - the character serial number.

substring - The serial number of the selected character sequence occurrence, for example, to the first appearance of the "-" substring.

end of string –all characters to the end of string will be compared.

Compare. The field specifies comparison type.



Character –content of two data cells of the table will be compared as two character strings.

Numeric – content of two data cells of the table will be compared as two numeric values. If string contains non-numeric characters they will be omitted on comparison.

Mixed – The strings will divided into substrings that contain character and numeric values. After that, same-type substrings will be compared.

> If sorting rules for the group are not set, the records of the product structure will be placed in an arbitrary order.

> Important! The order of applying sorting rules depends on their order in the list.

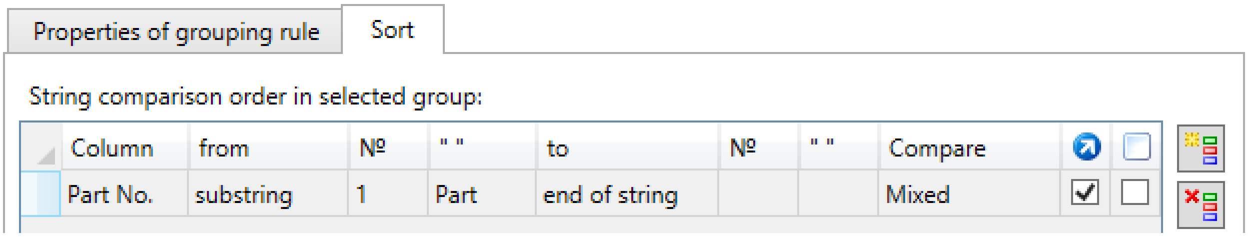
Position priorities

- If both numbers and characters are located on the same position in the data cells, the priority goes to numbers;

- If there are character sets "AA" and "AAA" on the same position, priority goes to the shortest record.

Example of sorting creation

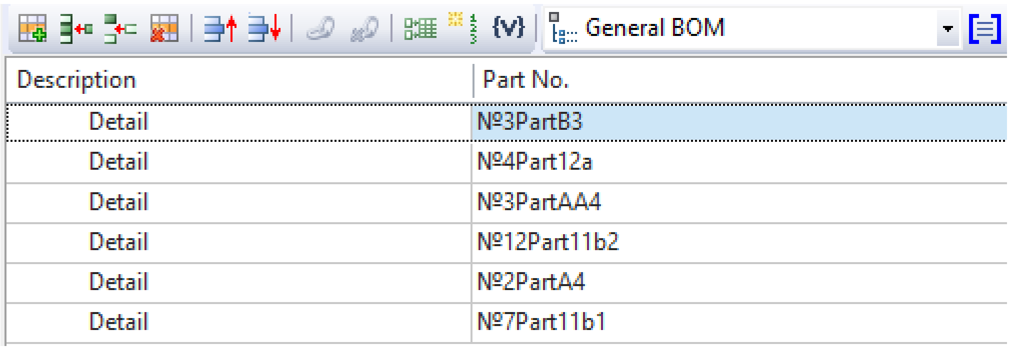
Sorting will be set for the "Part No." column for all records in the "Grouping rule_1" section.



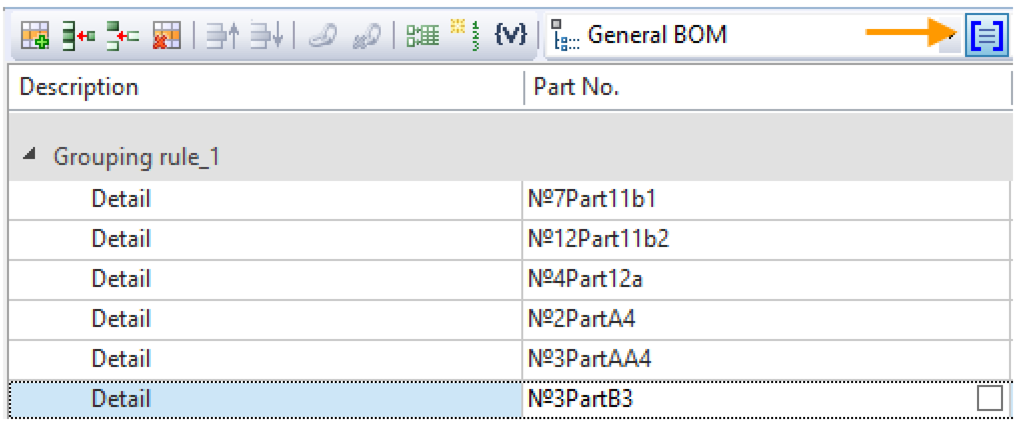
Each data cell of “Part No.” column will be searched for the **first** “Part” **substring** from the beginning.

Searching is performed to the **end of string**. Type of comparison is set to **Mixed**, i.e. all characters and numbers after the specified substring will be taken into account.

The data was inserted into the table after specifying the sorting conditions. They are sorted by order of creation.



After activation of **Apply product structure representation** mode, the sorting rules were applied to the records of the “Part No.” column.



The characters and numbers before **Part** substring were not taken into account.

Initially the comparison was made by the first position after the **Part** substring.

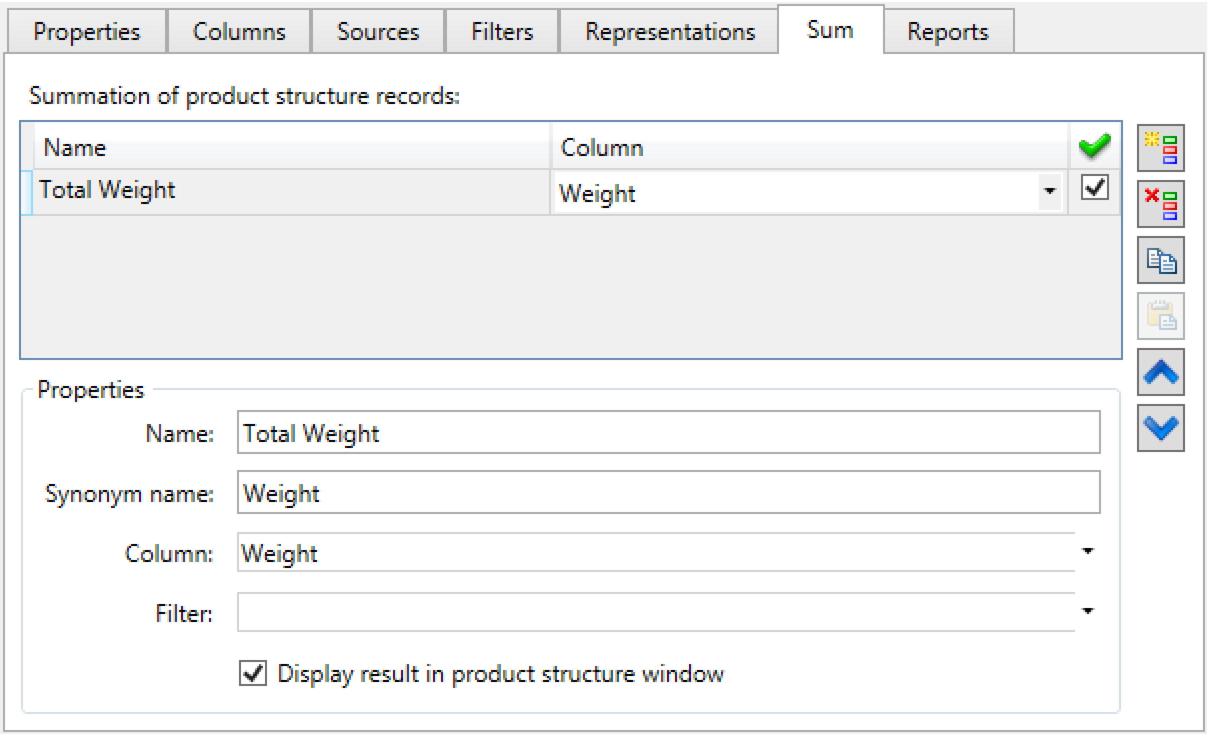
Numbers on the first position were selected from each cell (1). They were combined into substrings with nearby numbers (11, 12). After that, they were compared.

Then all chars on the first position were selected (A, B). They were combined into substrings with nearby characters, if available (AA). They were compared.

Then symbols on the second position were compared and so on until the end of the string.

Sum Tab

You can specify rules for summation in the product structure columns on **Sum** tab.



Summation rule will be displayed in **Summation of product structure records** list after creation.

There is the following set of specified properties for each summation rule in **Properties** group:

Name. Specifies name for the summation rule.

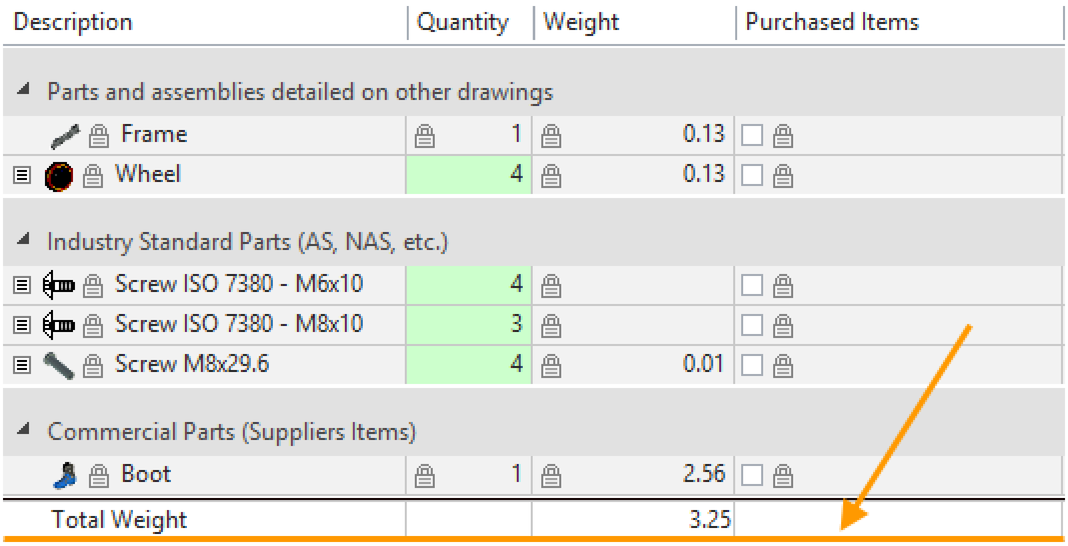
Synonym name. You can specify synonym name in the field. The synonym name is displayed in **Summation results** window. The window is described above.

Column. You need to select column from the drop-down list. Its data will be summed.

> If **Sum values when merging** parameter is specified for the selected column it will be taken into account.

Filter. You can select a filter from the drop-down list. The filter is created on the **Filters** tab. The filter will be applied for the values summation. I.e. only records that satisfy conditions of the filter will be summed.

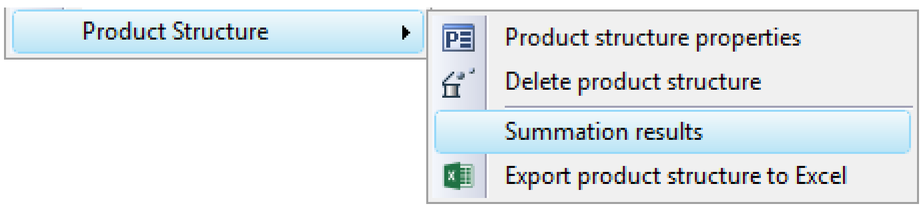
Display results in product structure window. When the flag is set, the summation result is displayed in the bottom part of **Product structure** window when **Apply product structure representation** mode is on.



Summation results

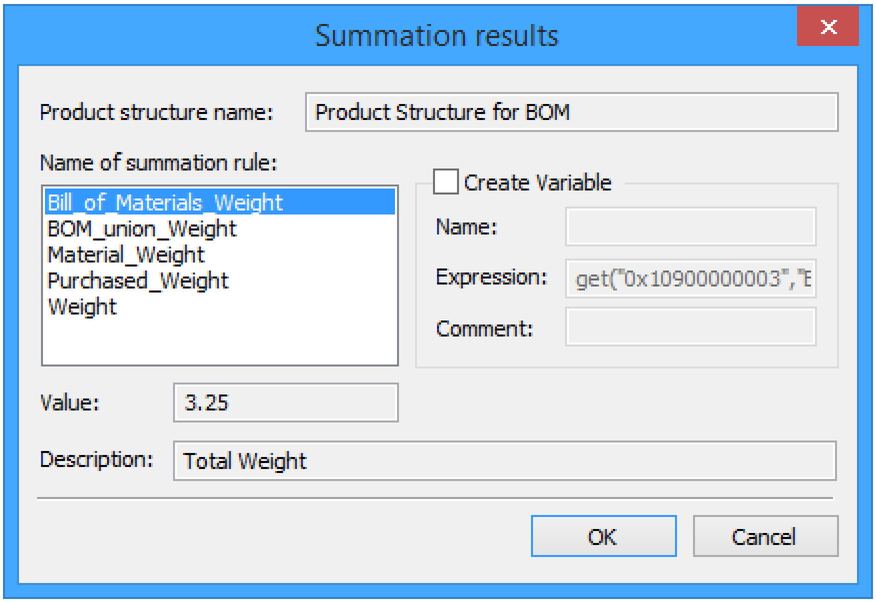
You can assign value of summation result to a variable. You need to perform the following sequence of actions to do that:

First, you need to select the **Summation results** item in the context menu of the product structure.



Summation results window will appear.

The product structure name is displayed in **Product structure name** field.



You need to activate **Create variable** flag and enter the variable **Name**. You can also enter a **Comment** to the variable.

The expression for getting variable value is displayed in **Expression** field.

You need to select necessary item from the list of summation rules. Each item name is a combination of representation and summation rule.

Press [OK] button to create the variable.

> Summation results are created separately for each representation.

Example: There are four representations in the product structure. Five records will appear in **Name of summation rule** field if you create one summation rule for the product structure.

Four of them refer to representations. They calculate sum in the specified column for records included in the representation.

The fifth record displays total amount in the column, but the representations are not taken into account. I.e. all the existing records data is summed.

> Selected **Filter** is used for each of the five records.

The **Summation rule name** consists of two parts. The first part is the name/synonym name of the representation (Bill_of_Materials), the second – the name/synonym name of the summation rule (Weight).

> If synonym names are specified for representations or summation rules, they will be displayed in the field. Otherwise, the names will be displayed.

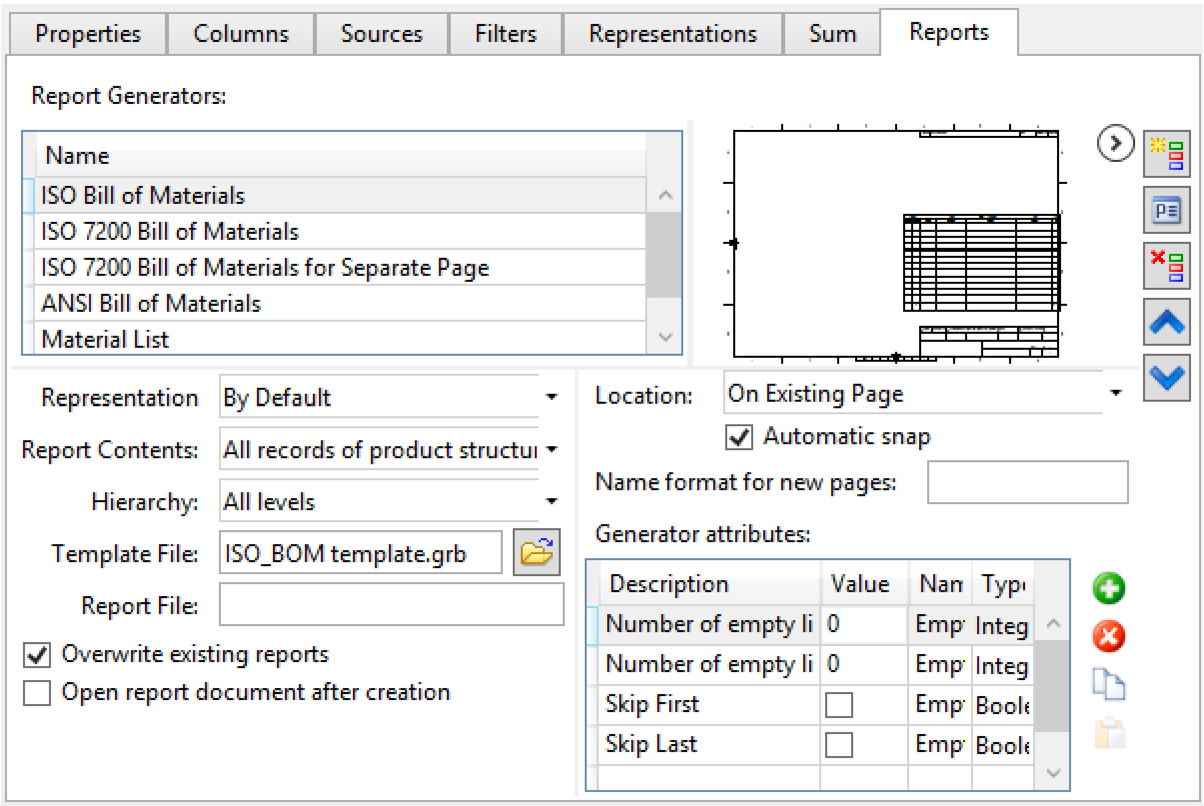
Information about the selected summation rule is displayed in **Value** and **Description** fields when you select any record in **Name of summation rule** list.

Reports Tab

Parameters of reports and appropriate template files can be set on **Reports** tab. The reports list is located on top of the window. Preview window is located to the right. You can change scale in the preview window using mouse wheel. You can hide the preview with button.

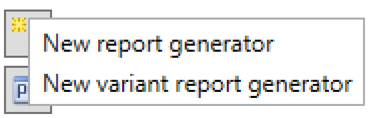
Buttons for creating, deleting and changing report properties are located on the right side. You can change reports order in the list using icons .

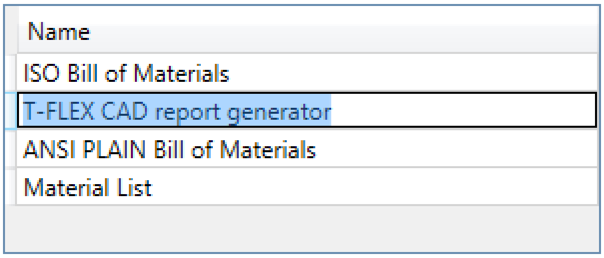
You can change properties of reports and their generators using **Report properties** and **Generator properties** tabs.



Add New Report

After pressing **Add** button you can select one of the available generator types. The generator processes existing data according to certain rules and forms reports.





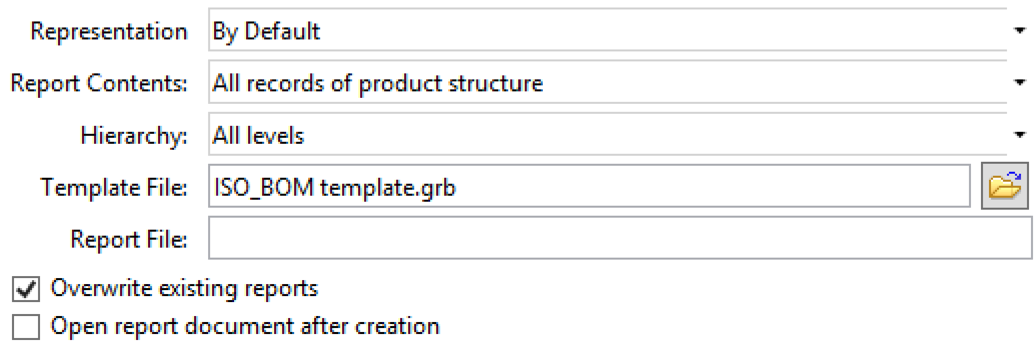
The selected generator name is shown in **Generator comment** field.

After selecting generator, a new report will appear in the list. You can change default name by pressing on its name string.

Report Properties

Report properties assigned for particular reports generator define values that will appear automatically in the corresponding data fields when a new report is created. This data can be then modified.

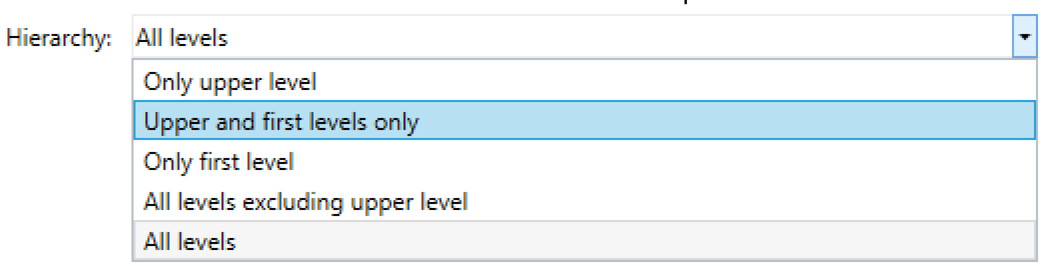
You can specify the following properties for the report:



Representation. Specifies representation which data and rules will be used in the report. If representation is not selected, the field contains value “By default” and the first representation will be used.

Report contents specifies mode of data inclusion in the report. If **All records of product structure** item is set, all its data will be included into the report. If **Only selected records** item is set, only selected product structure “branch” data will be included. It allows creating reports for subassemblies embedded in the main assembly file.

Hierarchy specifies levels of the product structure tree that will be included into the report. The “root” record of the tree is considered the zero level. There are several options:



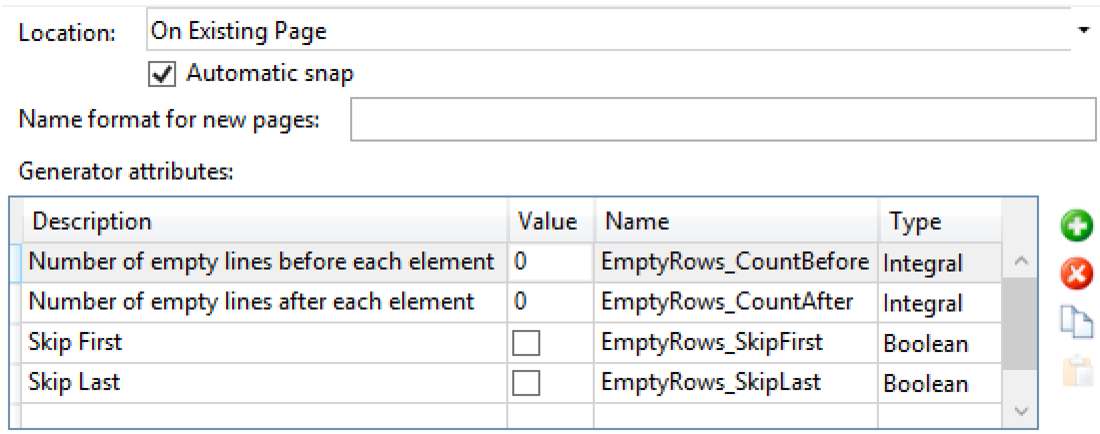
Template file. Specifies name of a report template file. Button is used for the file selection.

Template file creation is described in section “Report template”.

Report file specifies name of the file in which report, if created in a separate document, will be stored. The report file will be saved in the same folder that contains the product structure file.

Overwrite existing reports. If the flag is set, a new report created with the sane generator will replace the existing one. If the flag is not set, a new report will be created as a new table.

Open report document after creation. The flag is taken into account when you create report in a separate document. If the flag is set, report file will be opened after creation. Otherwise, the current document will stay active.

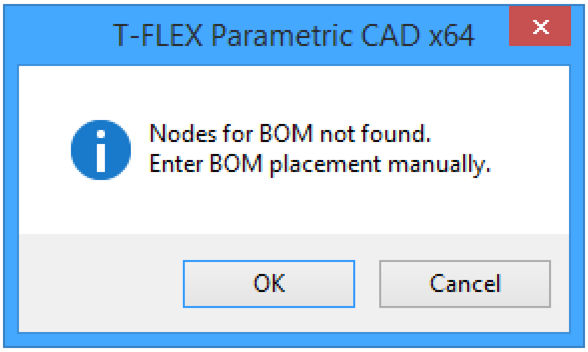


Location:

- **In new document.** After selection, a new document will be created. Its name is specified on the **Report** tab. The system creates a new document with the specified name and opens a new window with the report.
- **On new page.** A new drawing page with the report table is created in the current document. The table matches the selected template.
- **On existing page.**

Automatic snap flag activates mode of automatic snap of the report table to the two predefined nodes named “BOM1”, “BOM2” that define table position (upper left and bottom right points).

Names can be assigned to nodes in **EN: Edit node** command.



The snap requires that both named nodes exist on the same page. Standard title blocks already contain these nodes by default.

If the nodes are not found on the current page, system displays warning message and allows you to specify the table placement manually.

You need to specify placement of the top left corner of the table.

Name format for new pages.

In general case name format for new pages looks like: “BOM name” {#o} Page {#p}

For each page:

{#o} – will be replaced by the sequence number of the BOM of the current type.

{#p} – will be replaced by the sequence number of the BOM page.

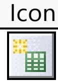
Generator Attributes

Generator attributes is the list of options that additionally control the result of report generation. These attribute are linked with macro program that is contained in the report template file, and that you can manage. The set of attributes may differ for various generators. Examples of such attributes:

- EmptyRows_CountBefore and EmptyRows_CountAfter – value sets number of empty strings before/after each record of the table.
- EmptyRows_SkipFirst and EmptyRows_SkipLast – when“0” value is set, empty strings that were set by attribute EmptyRows_CountBefore/EmptyRows_CountAfter will be displayed for all records. When another value is set, the empty strings will not be created for the first/last record in the section.

CREATE REPORT / BOM

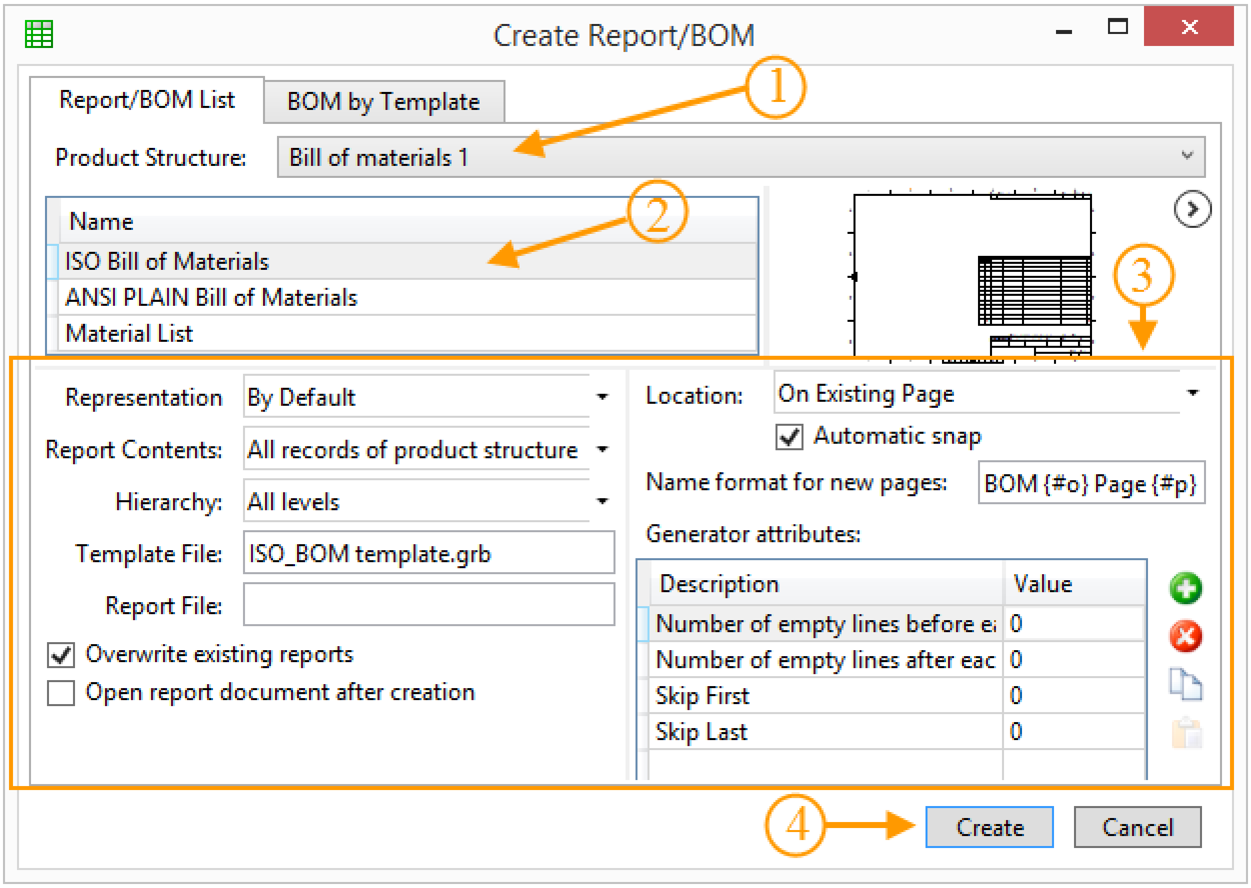
You can create a report/BOM via command:

Icon	Ribbon
	Bill of materials → Bill of materials → New…
Keyboard	Textual Menu
<BC>	Tools > Report/Bill of materials > New…

The command is used to create report or a BOM for the current assembly drawing. There are two tabs in the dialog box of this command: **Report/BOM List** and **BOM by Template**. The latter corresponds to the legacy mechanism and is not recommended for usage.

Report Creation

You can create report or BOM table using template mechanism on **Report/BOM List** tab.



1. Select product structure for the report;
2. Select report generator from the list.
3. Set report parameters.

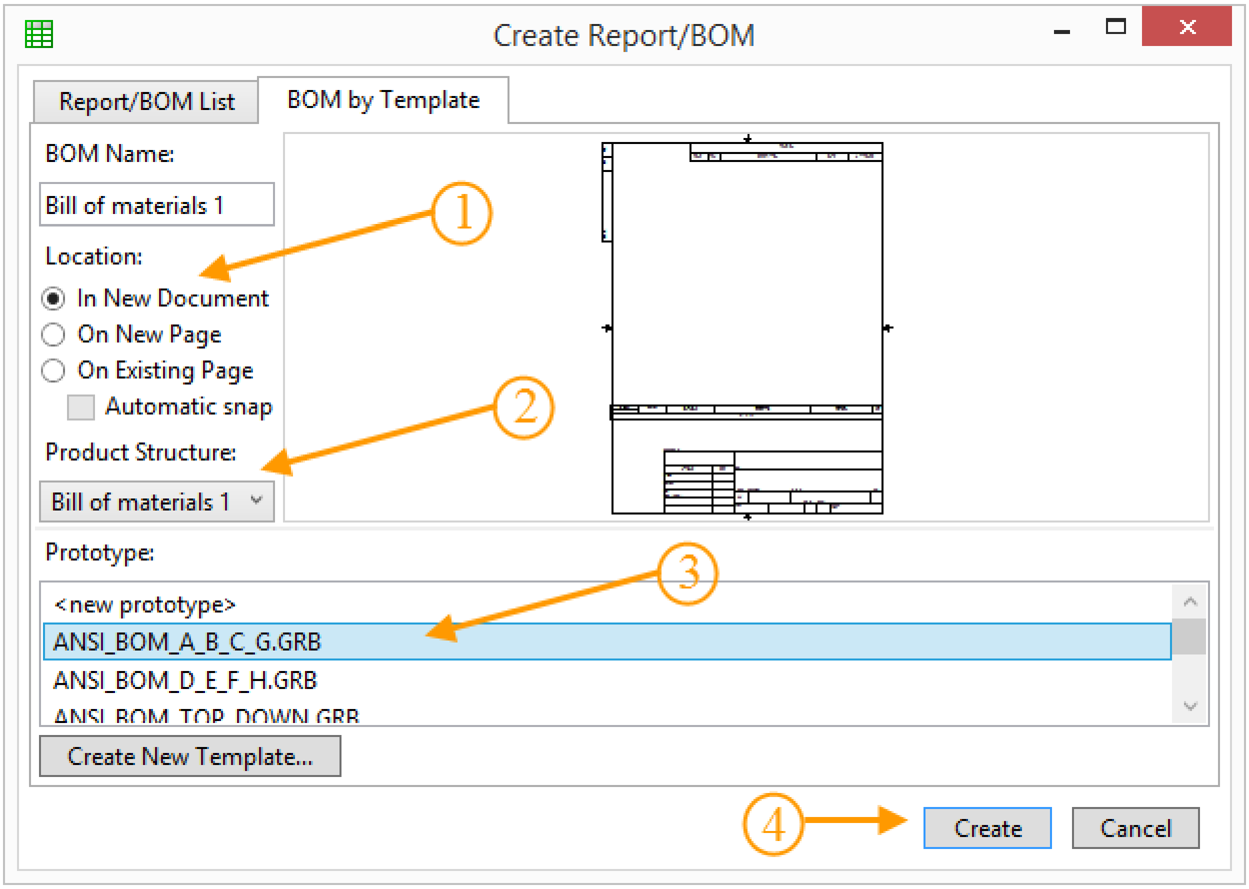
Parameters description can be found in “Product structure types” section.

4. Confirm creation using [Create] button.

You can also initiate new report creation with the toolbar in the **Product structure** window using option  **Create report based on product structure** or from the context menu of the selected record. In the second case, you can’t change the report properties before its creation. They are just copied from the product structure properties.

BOM Creation

You can also create a BOM using obsolete prototypes mechanism on the **BOM by template** tab.



1. Select location type:
   - In new document
   - On new page
   - On existing page

   Automatic snap flag is available only for **On existing page** type.
2. Select product structure for the BOM;
3. Select **Prototype** that will be used for BOM (The preview of the document will be shown in the preview window);
4. Confirm creation using [Create] button.

When you select the <new prototype> prototype, the BOM is created based on the "BOMStructure.mdb" database that contains only standard fields. I.e. you need to create all other parameters (column width, sorting rules, etc.) manually. It differs from the new prototype creation only in the fact that the specified parameters and properties of the BOM will be used only in the current document.

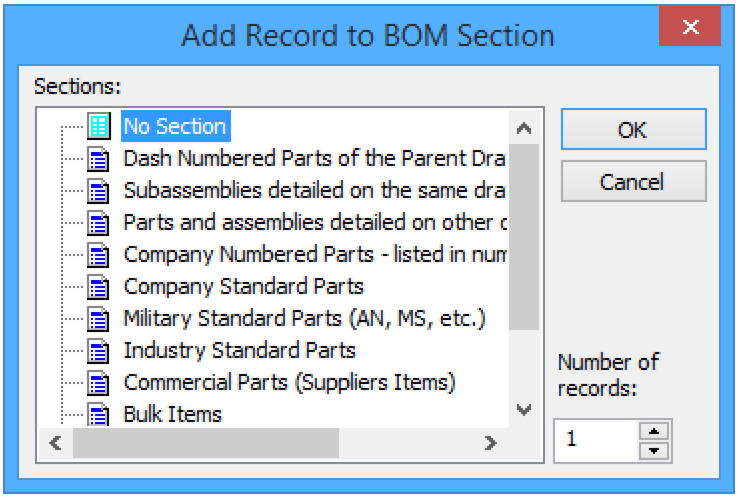
> If you want to use this prototype for any other BOMs, you need to save it in the "...\Program\Template\BOM" folder.

Variants of BOM location are described in “Product Structure Types” section.

If there are BOMs created in previous versions of the BOM creator, the **Convert old specification** flag will appear. When it is set, the old BOM data will be converted according to the selected type.

If parameter **Include in product structure** was enabled for fragments in assembly drawing file and any data was set for the fragment drawings, the created BOM will contain an appropriate records number. If the conditions were not met, the BOM will be empty.

After the BOM creation, the editing mode is automatically activated (except for BOM **on existing page**). If BOM is empty, the following window appears:



Using this window, you can manually enter a new record to any section. You can decline it by pressing [Cancel] button.

It is recommended to set “**Transparent**” **text editing** parameter on **Preferences** tab in **ST: Set document parameters** command. It allows to enter edit mode after pressing on any record.

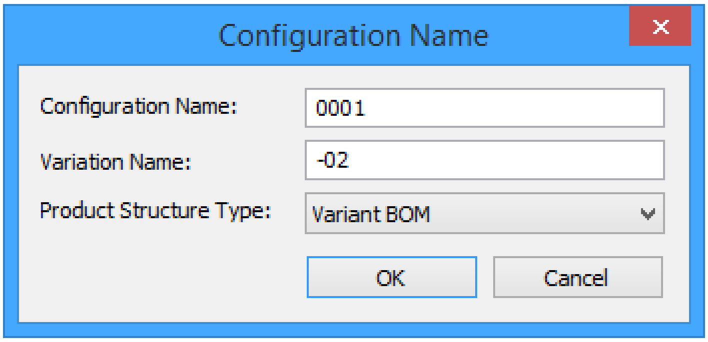
Variant Report / BOM Creation

To create a variant report/BOM you need to perform the following sequence of actions:

1. You need to create variations in the assembly file using  option in **FCE: Edit model configurations and variations** command.

More information about the command can be found in “Auxiliary Tools for 3D Assemblies Modeling” chapter.

You need to choose **Name** and "Variant BOM" **Product Structure Type** for the new variation in the following window:

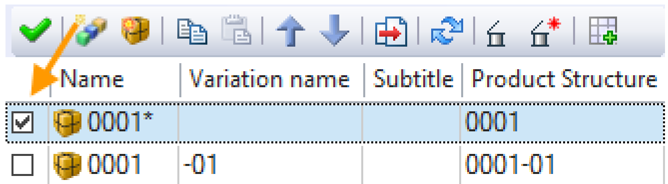


After saving the variation, the same product structure will be automatically created in the document.

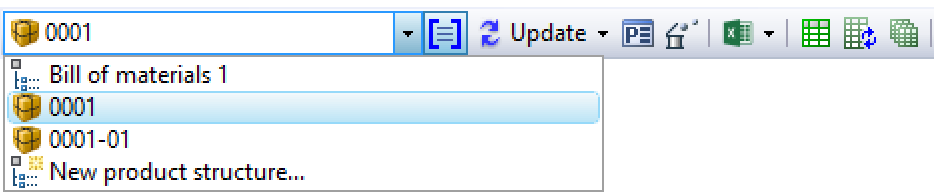
Note that variations creation is not available until the current document is not saved.

Optionally, you can specify subtitles of variations directly in the variations list.

2. Select base variation of variant BOM by setting the flag in the list of variations. If base variant is not specified, it will be set to the first variation in the list.



3. Select the variation in the **Product structure** window.

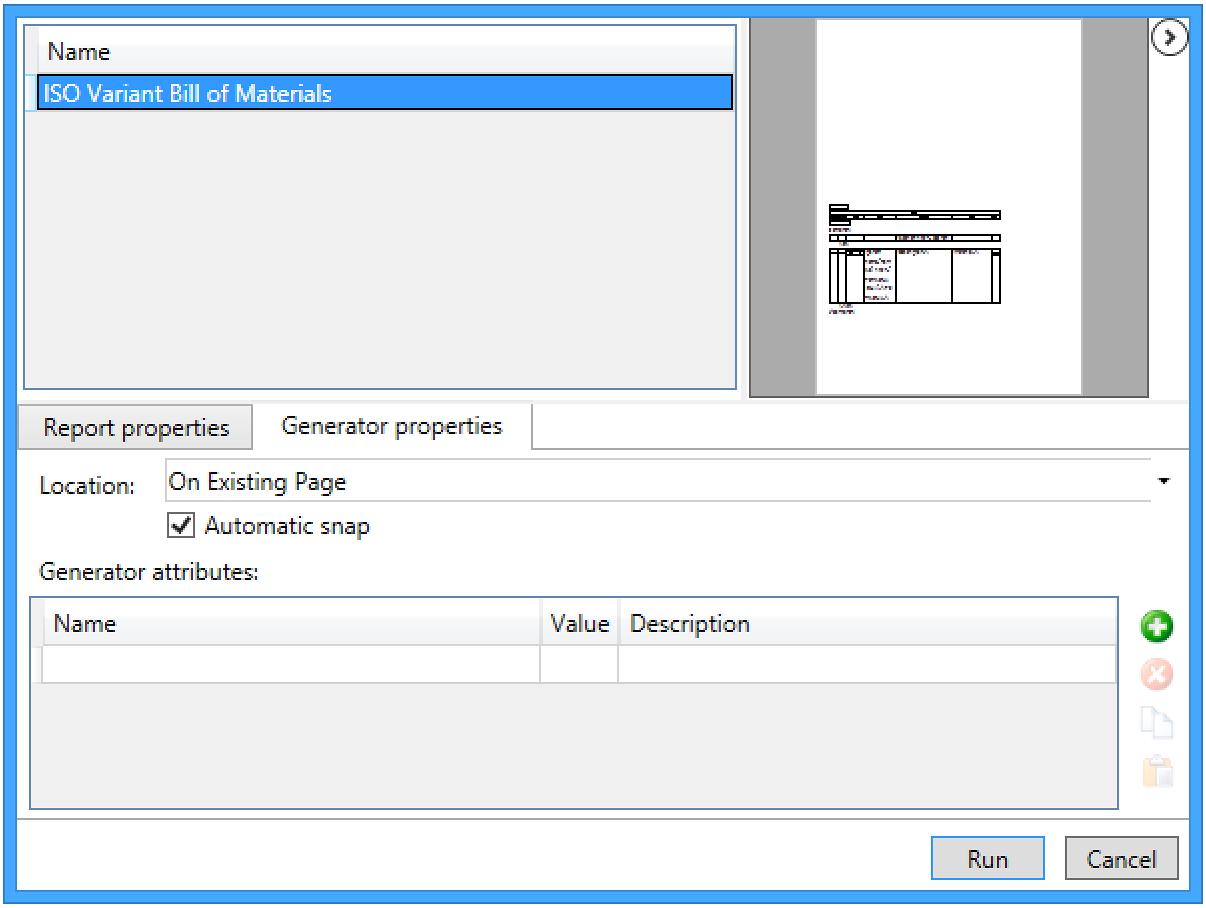


Its content can be edited if necessary.

4. Activate **Create report based on Product structure** command.

Report template corresponding to the selected variation will appear.

5. You can select the report location on **Generator properties** tab. The report will be created after [Run] button pressing.



Creation of Callouts on the Drawing

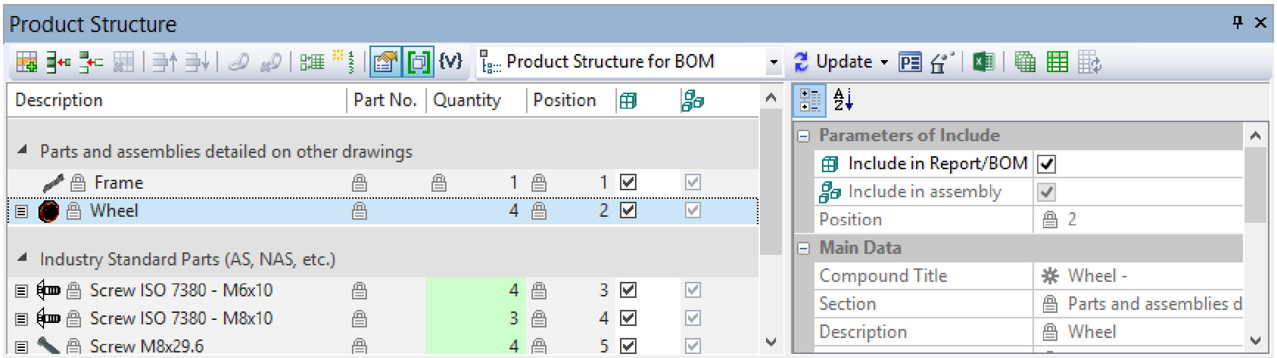
You can create callouts using command:

Icon	Ribbon
	Bill of materials → Positions→ Callouts
Keyboard	**Textual Menu**
<BL>	Tools > Report/Bill of materials > Callouts

There are three options in the automenu:

	<W>	Show product structure window
	<A>	Create All Callouts
	<Esc>	Exit command

If **Show product structure window** option is active, **Product structure** window will appear.



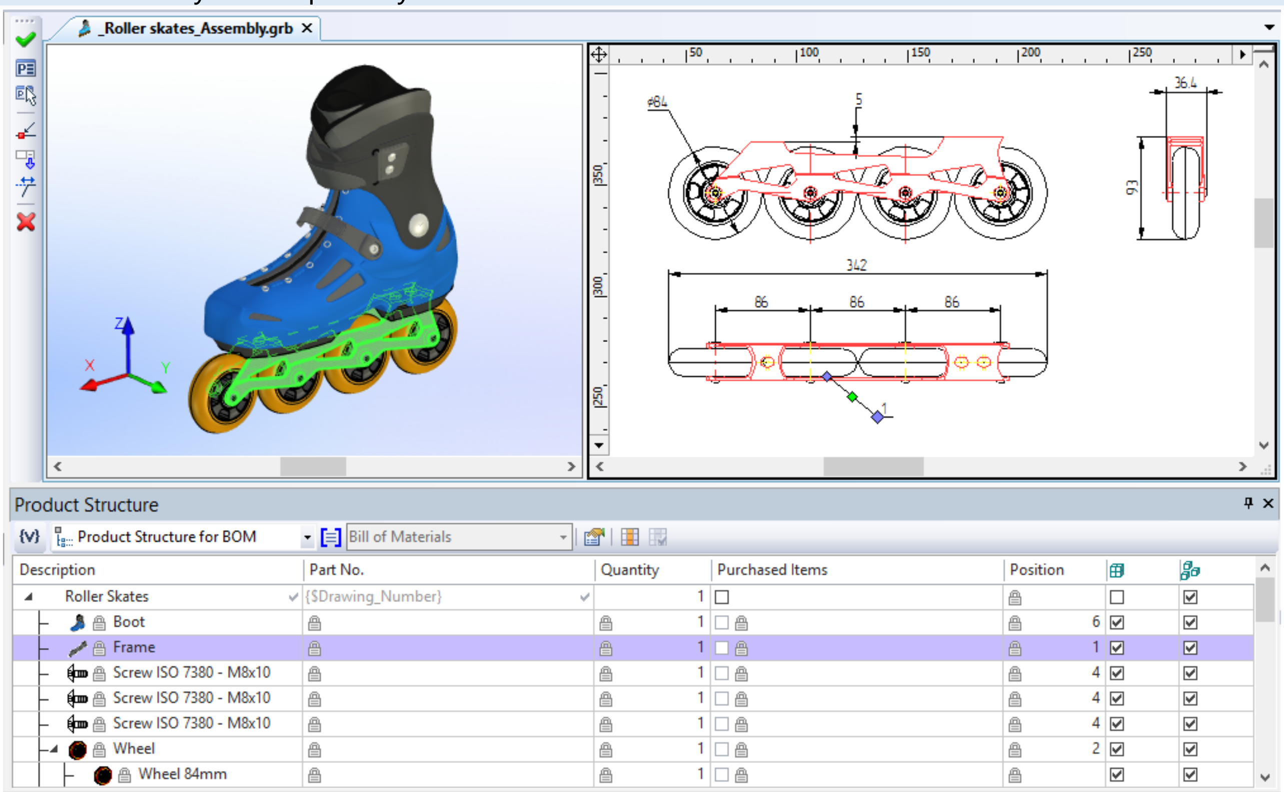
There are two ways to create callouts after calling the command:

1. You need to select a record in the Product structure window. A callout cursor for the record will appear. Callout is a specific type of leader note element. The corresponding fragment will be highlighted in 2D and in 3D windows. Then you need to specify fixing points of the callout in one of the existing windows.

   Position number will be assigned to Text field of inscription/callout Properties window automatically in “{{Position}}” format.

More information about leader notes creation can be found in “Leader Notes” chapter.

The “By default” parameters for leader notes, callouts of the product structure and bend notes are stored in the system separately.



2. You need to select an object on the drawing. A callout for the record, which corresponds to the object, will appear. You need to specify its location on the drawing. The **Product structure** window is not necessary in this case.

Create BOM callout command provides links between records of product structures and callouts. If there were any changes which lead to the position number change, callouts will update.

You can update positions using the following commands:

BRP: Update callouts. This option is used when you change sorting or perform any other action that means positions change in the product structure. The callouts will be updated according to the current values after the option activation.

BRA: Update all. This option updates product structures and callouts. It is used when content of product structure changes, e.g. when new fragments are included into assembly.

Automatic Callouts Creation

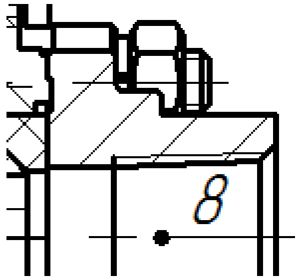
Create All Callouts command is used for multiple callouts creation.

The following options are available in the automenu of the command:

	<G>	Group Alignment of Callouts
	<N>	Callout without Leader
	<*>	Select All Elements
	<C>	Cancel Elements Selection

The **Group Alignment of Callouts** option allows to align all callouts after their creation.

If the option **Callout without Leader** is enabled, the callouts will be created without leaders. Numbers of positions will be snapped to the center of objects that linked with the record in the product structure.



You can select all records in the product structure and create callouts for them using the option **Select All Elements** .

> If there are several projections (2D views) created for a product structure item, callouts are created for each of them.

Use the **Cancel Elements Selection** to cancel selection of all records.

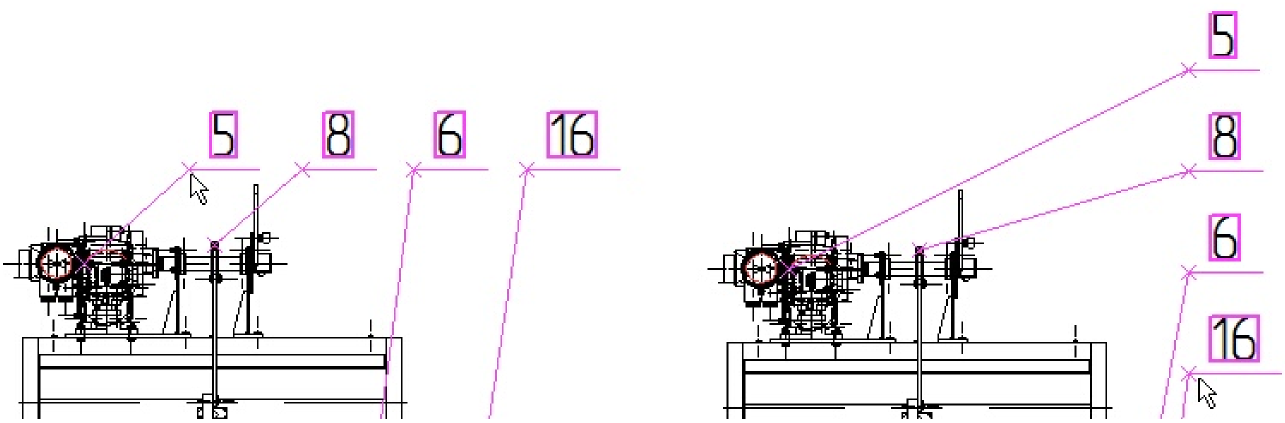
After all options are specified, you should press .

Callouts without leaders are created immediately.

If the option **Group Alignment of Callouts** is activated, the callouts will automatically follow the cursor. The following options will appear in automenu:

	<Tab>	Horizontal
	<Tab>	Vertical
	<Z>	Change leader line jog orientation
	<D>	Change placement of additional jogs

Options and allow to change direction of the created callouts row – horizontal or vertical.



The option changes jog orientation for all created callouts.

> When the callouts are assigned automatically, placement of leader note arrows is defined as the middle of the bounding rectangular for each 2D fragment or the middle of the selected projections lines of 3D fragments.

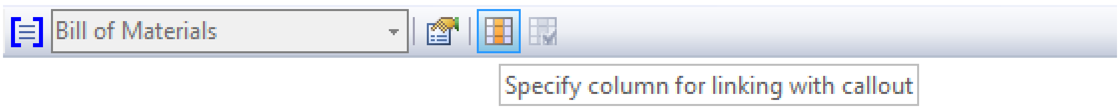
You need to press to finally set the callouts location.

The callouts use standard parameters of leader notes except for arrow ending type (the point is used) and priority (127 is used). If there are several positions corresponding to one fragment, the callout with several jogs is used.

You can change additional jogs placement using option. Created callouts can be edited.

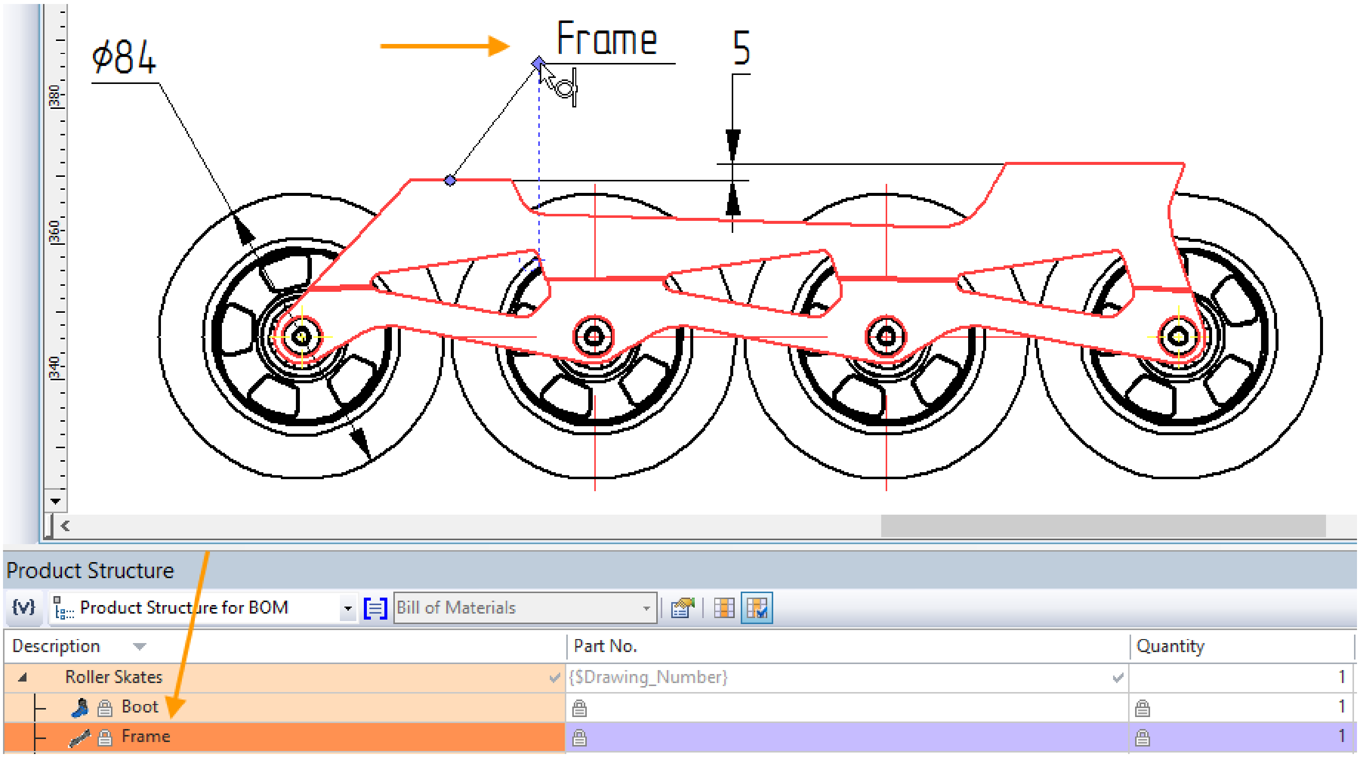
Arbitrary data cell of product structure selection

You can output content of an arbitrary product structure cell to the callout. Use the **Specify column for linking with callout** option for this purpose.



System waits until you select the product structure column after the option activation. The selected column will be highlighted and **Use specified column for linking with callout** mode will be activated.

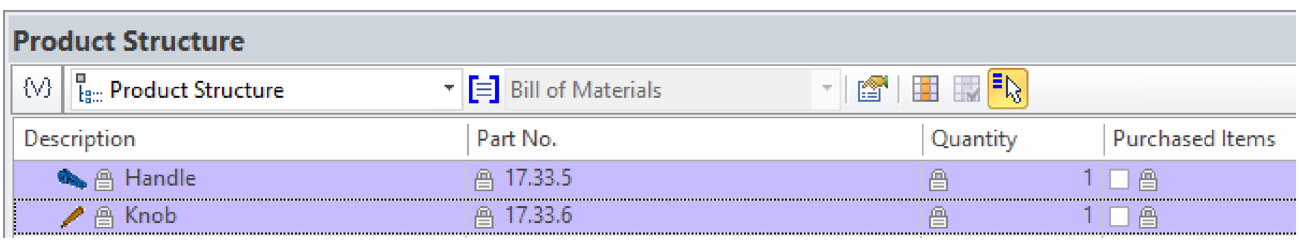
The next step is to select string of the product structure. Its content will be set to the callout string.



The selected column is stored. You can disable option (the column will not be highlighted) and continue callouts creation in an ordinary way. You can enable link with the selected column again, if necessary. The next callout will be linked with the selected cell of the column.

Additional Jogs

Use option to create several jogs. After the option activation, you need to select all records of the product structure that should be added to the callout.



You can use existing callouts to show additional positions on the drawing. Therefore, several records may be associated with one callout. For this purpose, point the callout using the mouse cursor, press and select **Edit** item from the appeared automenu.

You need to select the record of the product structure and create a new jog using button on the **Multiple jogs** tab in the **Properties** window.

Use and buttons to change the order of jogs. To delete jogs use button.

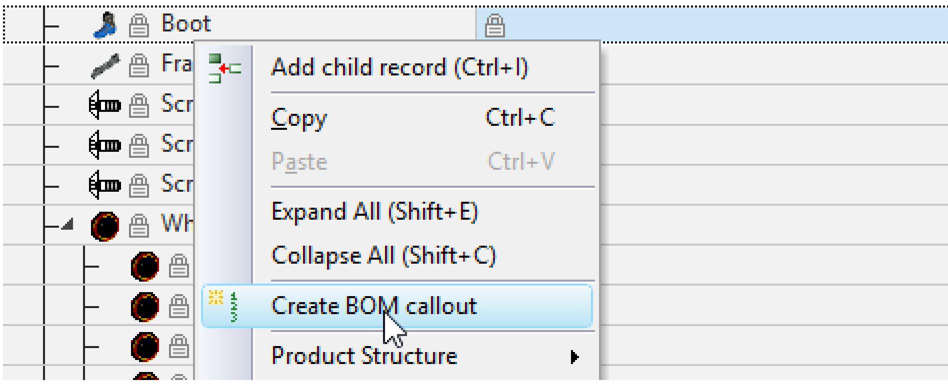
To change position on the existing jog, select it and specify any other record in the **Product structure** window.

> **Multiple jogs** list contains the main leader note jog. Do not delete it.

The symbols "{{ }}" means that the callout is linked to the product structure and is updated in accordance with it. If you remove these symbols, then the link will be broken.

Transparent Call of Create BOM Callout Command

Create BOM callout command is available in the context menu of the selected record.



The command also activates automatically when you drag a record into the scene using Drag'n'Drop mechanism. For this purpose select a record in the product structure, move it and release in the 2D or 3D window. The command of callout creation will be activated.

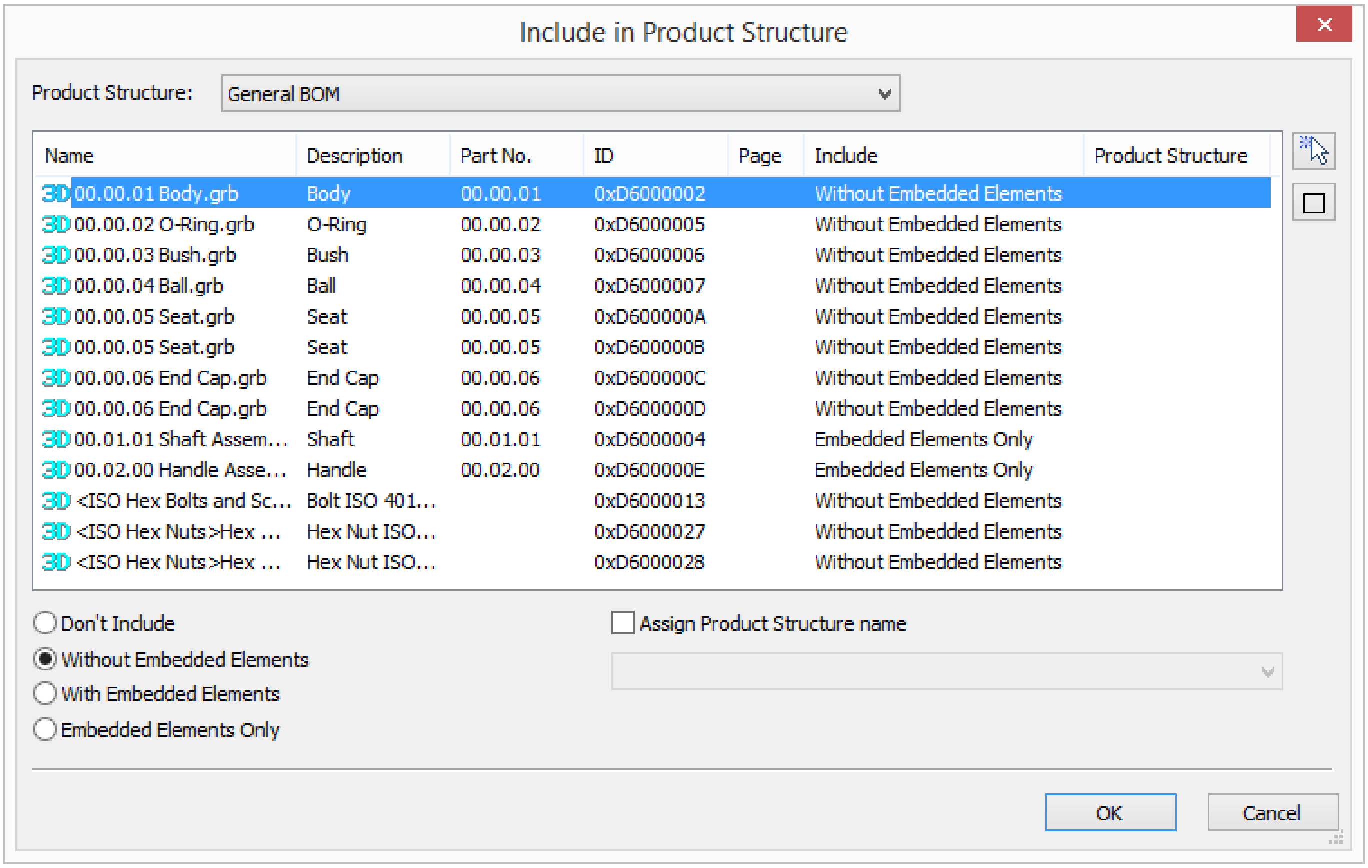
INCLUDE IN PRODUCT STRUCTURE

The command:

Icon	Ribbon
	Bill of materials → Product structure → Included fragments
Keyboard	Textual Menu
<BI>	Tools > Report/Bill of materials > Included fragments

Is used for managing inclusion of document fragments in a product structure.

After activation of the command the window that contains lists of fragments and product structures appears.



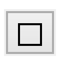
The **product structure** parameter specifies the name of product structure for which the specified fragments inclusion modes will be used: **don't include, without embedded elements, with embedded elements, embedded elements only.**

When you select <New Product Structures> from the drop-down list, the specified inclusion parameters will be used for new product structures.

When you select fragment its inclusion mode is specified in the lower part of the window.

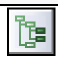
Using **Assign product structure name** parameter you can select a product structure from the fragment file, which data will be included in an assembly product structure.

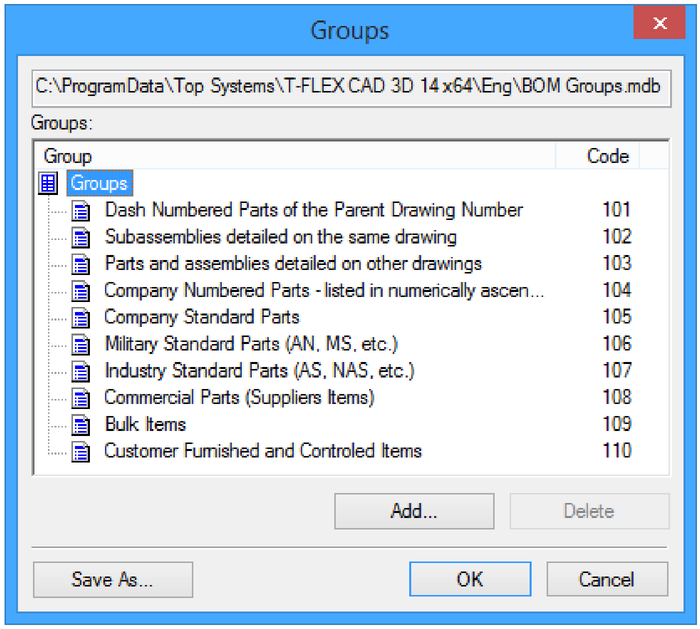
Multiselect. . You can select several elements from the fragments list using tapped button <Shift>. Tapped <Ctrl> button allows you to add/remove fragments from the selected list.

Button  **Select all** allows to select all fragments. Button  **Deselect all** allows to deselect all fragments. When you select fragments, each of them is highlighted in 2D and 3D windows. If there are more than one fragment in the list, selected parameters are applied to all of them.

EDITING BOM SECTIONS

BOM sections are stored in a special database. The system installation includes the database with the standard set of BOM sections (…`\Program\ BOM Groups.mdb`). The standard set of sections can be changed by the command **BG: Edit BOM Sections**:

Icon	Ribbon
	Bill of materials → Options → Sections…
Keyboard	Textual Menu
<BG>	Tools > Report/Bill of materials > Sections…

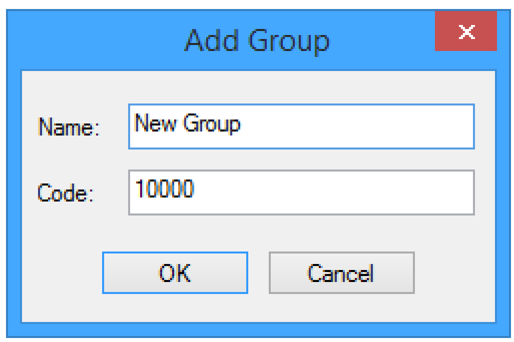


Upon calling the command, the **Sections** dialog box appears.

The data box in the upper portion of the dialog box displays the path to the currently used database. The current database can be set in the command **SO: Set system options** on the tab **Bill of Materials**.

The BOM sections are displayed as a tree, with each group being assigned its unique Id number (Code). To add a new group, select the parent group to add the desired one as a subgroup, and press the graphic button **[Add…]**. As a result, the **Add Group** dialog box will appear.

It allows defining the name for the group being created, and its ID number. The ID provides the relation between the group database and the BOM data. The default IDs are assigned subsequently, beginning with 10000.



You can manually assign an ID number. If the number you try to assign is already reserved, the system will prompt you about this. Upon the confirmation, the newly created move will be entered in the active database and will be displayed in the tree of sections, thus being incorporated in the BOM definition. To delete a group, select it in the list and press the graphic button **[Delete]**.

> Deleting listed sections is not recommended, since the relations may be lost between the sections and the earlier created BOMs.

To reorder the sections in a BOM, use the BOM properties dialog box and its tab **Sections**. To save the database under a different name, use the button **[Save As…]**.

REPORT / BOM TEMPLATE

The report/BOM template is a specially prepared GRB file that contains data and options for generating reports. The report format and data composition are described in this file according to certain rules.

This file contains table created using command **TE: Text** and various structured data in its cells. System installation has various predefined report templates. Path to the report templates folder is specified on the **Bill of Materials** tab in **SO: Set system options** command.

Report template structure

The report structure is similar to the XML language. The structure represents tree of elements. One or two tags specify an element. The element has embedded *attributes* and *contents*.

Allowed elements names (without register):

1.*group* is used only for table framing.

2.*container* is used for variant reports (more information can be found in a report generator description).

3.*list* – The list of elements. It is used for table framing.

4.*param* – The parameter is used only in the table. Content is ignored. An element is replaced by the value from the product structure column. Should not be embedded into each other.

5.*group_macro* – The macro for group processing. Should be used as embedded element for element *group*.

6.*summary* – The result. Used only for table framing.

7.*outcome* – The resulting value. Used in *summary* element.

8.*frag* – insertion into the fragment cell.

9.*sum_res* - The result of the summation described on the “Sum” tab in the product structure properties. Used only in the *summary* table.

10.*variable* – The value of the variable. The variable should exist in the file on the basis of which the report is created.

11.*table_hider* – The element to hide the table. Used with the filter.

Only *group* and *list* can be the top-level elements.

Allowed *attributes* values:

1) *name* is used only for *param* element.

2) *filter* is used in all elements.

3) *recursive* is used only for *group* and *list* elements. If there is hierarchy in the input data, it will influence the result of product structure records sampling.

4) *recursive_template* The attribute shows that the element is a template for all groups.

5) *str_process* is used in *param* and *outcome* elements for post-processing or text value retrieving from the record. This attribute is used if insignificant processing of text value is necessary. For example, to specify the formatting and adjust the number of decimal places.

6) *str_proc_macro* is used in *param* and *outcome* elements for post-processing or a text value received from the record. This attribute uses the specified macro for text processing.

7) *hide_table* is used in table framing elements. Specifies the rule according to which the table doesn't displayed.

8) *source* is used with *frag* element.

9) *sorce_macro* is used with *frag* element.

10) *from_item* is used with *frag* element.

11) regenerate 3d is used with frag.

12) index is used with table_hider.

The elements have two forms of recording:

1) *{list}*element content*{/list}* - 2 tags and the content.

2) *{param name="Description "/}* – 1 tag and no content.

When the report is generated the *group* and *list* elements copy the content (table) for each group/ product structure record to the report

Element group

Param elements are filled in the table of *group* element as follows: you take the first record from the group and its values are substituted.

All embedded *list* elements will operate merged data (those data that will be shown in **Product structure** window in **Apply product structure representation** mode).

Element list

If *list* element is not embedded into *group* element, it is operating all records of the product structure (as they look like in **Product structure** window in **Apply product structure representation** mode).

If the input data has the hierarchy:

- By default, *recursive* attribute is disabled, i.e. only the structure records of the top level will be displayed.
- If you set *recursive* as *true*, all records will be displayed.

It is allowed to create an embedded *list* element in the existing *list* element. In this case, the table from the embedded *list* element will be created and filled in for each product structure record with child records.

Element param

Element *param* is replaced with the corresponding value of the product structure record cell.

Full element record: *{param name="Description"/}*. A full element record should be used if the column name contains spaces.

The element has a simplified record:

1)*{param Description/}* – the *name* is an attribute by default, thus its specifying is not necessary; 2)*{Description/}* – a *param* is an element by default, , thus its specifying is not necessary.

For example, record *{Part No./}* is not recognized as correct. For names of two or more words, you need to use a full record.

Names *Pos, Position* are allowed to display positions.

Element summary

The element frames the table. It can be used as a top-level element. The table is just copied to report in this case.

When you use the element in *group* and *list* elements, information obtained by processing *group/list* records can be output into the table. The information is output using *outcome* element.

Optional element *before="true"* outputs table before *group/list* records.

Example of top-level *summary* element:

{summary}

Header		*EmptyGlobalHeader*		

{/summary}

Example with embedded **group** and **list** elements:

{summary}

Header		*EmptyGlobalHeader*		

{/summary}

{group}

	{Section *str_process="str = .SpecGroup(str)"*/}	

{list recursive="true"}

{summary before="true"}

EachGroupHeader:	*At all: {outcome name="Quantity" out_operation="numeric_sum"*

	argument="F4"/}

{/summary}

{summary}

EachGroupFooter:	*At all: {outcome name="Quantity" out_operation="numeric_sum" argument="F4"/}*

{/summary}

{/list}

{summary before="true"}

AllGroupsHeader:	*At all: {outcome name=" Quantity " out_operation="numeric_sum"/}*

{/summary}

{summary}

AllGroupsFooter:	*At all: {outcome name=" Quantity " out_operation="numeric_sum"/}*

{/summary}

{summary}

AllGroupsFooter:	*All names: {outcome name="Description" out_operation="str_concat" argument="-;"/}*

{/summary}

{summary}

AllGroupsFooter:	*Unique names: {outcome name="Description" out_operation="unique_strs" argument=";"/}*

{/summary}

{/group}

{summary}

Header		*EmptyGlobalFooter*		

{/summary}

Element outcome

The element is used to output resulting information inside framed table {summary}.

Main attributes:

1.*name* – a name of parameter which is used to gather resulting information.

2.*out_operation* – values processing type:

- numeric_sum – numerical sum of all values;

- str_concat – strings sum;
- unique_strs – list of unique values.

3.*argument* – additional string parameter conveyed to *out_operation*.

- numeric_sum – string output format;
- str_concat – delimiter between summed strings for values enumeration;
- unique_strs – delimiter between summed strings for unique values enumeration.

You can also specify attribute for sorting records that will be used in calculation using *source_filter* attribute (works similar to *filter* parameter). You can use *str_process* for the string post-processing.
Examples:
{outcome name="Description " out_operation="unique_strs" argument=";"/}
At all: *{outcome name="Quantity" out_operation="numeric_sum" argument="F4"/}*

Element sum_res

The *sum_res* element is used for summation result output. The summation is set on the "Sum" tab in the product structure properties.

Main attributes:

name – a summation name. The name can be found in the product structure window. To do so call **Product structure** > **Summation results** from the context menu.

The name can be omitted. Then it will be selected automatically (Representation name + Name of the first summation).

You can use *filter* attribute.

You can use *str_process* for string postprocessing.

Examples:

{sum_res/}

{sum_res name="Mass"/}

Total: {sum_res Mass/}

Element variable

The *variable* element is used to output document variables into report. It can be displayed in the *summary* header table or in *group* and *list* tables.

Main attributes:
name – a variable name.

If there is no variable in the document, an empty string will be outputted.

You can use *filter* attribute.

You can use *str_process* for a string postprocessing.

Examples:

{variable name="nCount"/}

{variable name="$Number"/}

Formatted output of valid values (1 decimal place):

{variable name="nCount" str_process="str = str.AsDouble().Format(1)"/}

Element group_macro

The *Group_macro* element is a group processing macro. This element should be used as an embedded element of *group* element. Can use attribute *filter* – the macro will be applied only for the groups, which satisfy the condition.

Name of the element must be specified. The name specifies macro that should be started before the table filling. The macro prototype: void func(MacroCallContext context, GroupItemInfo group);

The first parameter contains context for the macro call.

Example:

```
{group}
    {group_macro name="Gen.Gen.GroupMacro" filter="Section = Company standard parts"/}
...
{/group}
```

The macro example:

```
using System;
using System.Linq;
using TFlex.Model;
using System.Collections.Generic;
using TFlex.Model.Model2D;

using TFlex.CadReportGenerator;

namespace Gen
{
   public class Gen
  {
      public static bool GroupMacro(MacroCallContext context, GroupItemInfo group)
     {
          var generAttrs = context.Properties.Attributes;// The report attributes check.
           if(!generAttrs.HasAttribute("attrName")
                 || generAttrs["attrName"].ValueAsBool == false)
                 return true;
           //%%TODO: fill in
           return true;
     }
   }
}
```

You need to add link to TFPSCadReportGenerator for the macro compilation.

Element frag

The element is used only in tables, one element per cell. Fragment is inserted instead of it.

One of the four ways can specify the path to the fragment:

1. The filename is taken from the product structure column: *{frag name="Remarks"/}*;

2. The filename is specified explicitly: *{frag source="<3D Assemblies>Cam.grb"/}*;

3. The name matches the current element name (the fragment from which the data was raised will be inserted): *{frag from_item="true"/}*;

4. The name is received from the macro (look through source_macro):

{frag source_macro="Gen.Gen3.SourceMacro"/}.

You can specify page name in the fragment document, its image will be inserted into the cell:

{Frag Source="<Fitting_v2>Sketch.grb" page="sketch_p"/}

You can also use *filter* attribute (filter="…") similar to *param* and *outcome* elements.

The *auto_width* attribute allows you to automatically select the width of the fragment so that it fits into the cell.

Element table hider

You can add several tables between tags that specify an element. Element *table hider* allows to specify logic for hiding tables.

Main attributes:
Index – an index of the table, which is embedded into the parents tag (indexing from 0).

filter – a filter for hiding tables.

Attributes

In the general case the attributes are recorded like *attrName="attrValue"* in the tag that opens element.

Attribute filter

Example of a filter record:

{group filter="section = 'Company standard parts'"}

First lexical unit – product structure column name

Second lexical unit – operator

Third lexical unit – value

> The filter operator (in this example - '=') is recorded with the space in opposite to the attributes that have no space. A string value is recorded in single quotes.

You can specify logical relation between expressions: AND / OR.

{group filter="Section = 'Company standard parts' OR Section = 'Unknown'"}

You can use parentheses in the filter ( ).

{group filter="(Section = ‘A' AND Quantity = 1) OR (Section = ‘B' AND Quantity = 2)"}

Special symbols in the column name processing

If the column name contains spaces or symbols, you need to change them

- ampersand "&" replace with "&"
- less "<" replace with "<"
- more ">" replace with ">"
- quotes """ replace with """
- apostrophe "'" replace with "'"
- space " " replace with a " "

For example:

{param name="Total Quantity" filter="Total Quantity != 0"/}

When the condition formed &#032 is replaced by the space.

Attribute hide_table

The attribute is similar to the attribute *filter*. It specifies the condition for the current element. According to the condition, the table will not be displayed. It can be used for the elements that are framing the table.

Example:

{group hide_table="Section = 'No' OR Section = '' "}

...

{/group}

For the groups without the section or it is “No” the heading table will not be created.

List of valid operators:

- =
- !=
- >
- >=
- <
- <=
- IsOneOf – included in the list
- IsNotOneOf – not included in the list
- IsNull - does not contain data
- IsNotNull – contains any data
- ContainsSubstring - contains
- NotContainSubstring – does not contain
- StartsWithSubstring – starts with
- EndsWithSubstring – ends on
- IsEmptyString - does not contain text
- IsNotEmptyString – contain text
- MatchMask – merges mask
- NotMatchMask – does not merge mask

Attribute str_process

The attribute is used in *param* and *out_come* elements for the post-processing of text value that is received from the element.
Example: The value of the attribute is recorded using C#. The resulting variable is “str”.
str_process="str = str.Replace('R', 'V')"
There are pre-determined methods allowed in “P” class:

- P.SpecGroup – processes full BOM section name and lefts only subsection name after last ‘\’.
  str_process="str = P.SpecGroup(str)".
- P.AsDouble – converts string to real number:
  str = (str.AsDouble() * 1.2).ToString()
- P.AsInt – converts string to integer number:
  str = (str.AsInt() * 2).ToString()
- P.Format – converts number to string with formatting (by default there are two characters decimal places):
  str = (str.AsDouble() * 1.2).Format(1) -one character decimal places.
  str = (str.AsDouble() * 1.2).Format() - two characters decimal places.

Example of composite processing:
{Section str_process="str = P.SpecGroup(str).Replace('R', 'V').ToLowerInvariant()"/}

Attribute str_proc_macro

The attribute is used in *param* and *outcome* elements for the post-processing of a textual value, retrieved from product structure record. The attribute value should specify the post-processing macro name.
The attribute example:
{param name="Title_1" str_proc_macro="Gen.Str.ConstVarReplacer"/}

The macro example:

```
using System;
using TFlex.Model;
using TFlex.Model.Model2D;
using TFlex.Model.Model3D;

using TFlex.CadReportGenerator;

namespace Gen
{
 public class Str
 {
```

```
        public static string ConstVarReplacer(MacroCallContext context, String originalValue)
        {
                if(String.IsNullOrWhiteSpace(originalValue))
                        return originalValue;
                else if(originalValue == "ConstPart")
                        return "Variations constant data:";
                else if(originalValue == "VarPart")
                        return " Variations variable data:";
                else
                        return originalValue;
        }
 }
}
```

Attribute from_item

The Boolean attribute used with *frag* element. It means that the path to the fragment is formed from the current element (a fragment from which the data was raised will be inserted).
Example:
{frag from_item="true"/}

Attribute source

Used with *frag* element. Specifies a path to the fragment file that should be inserted:
Example:
{Frag Source="<Fitting_v2>Sketch.grb"/}

Attribute source_macro

Used with *frag* element. Specifies a path to a fragment file that should be inserted using macro:
Example:
{frag source_macro="Gen.Gen3.FragSourceMacro"/}

Macro example:

```
using System;
using System.Linq;
using TFlex.Model;
using System.Collections.Generic;
using TFlex.Model.Model2D;

using TFlex.CadReportGenerator;
```

```
namespace Gen
{
 public class Gen3
 {
        public static string FragSourceMacro(MacroCallContext context, ItemInfo item)
        {
                string val = item["Annotation"];
                return String.IsNullOrWhiteSpace(val) ? "<Fitting_v2>Sketch.grb" : val;
        }
 }
}
```

Attribute page

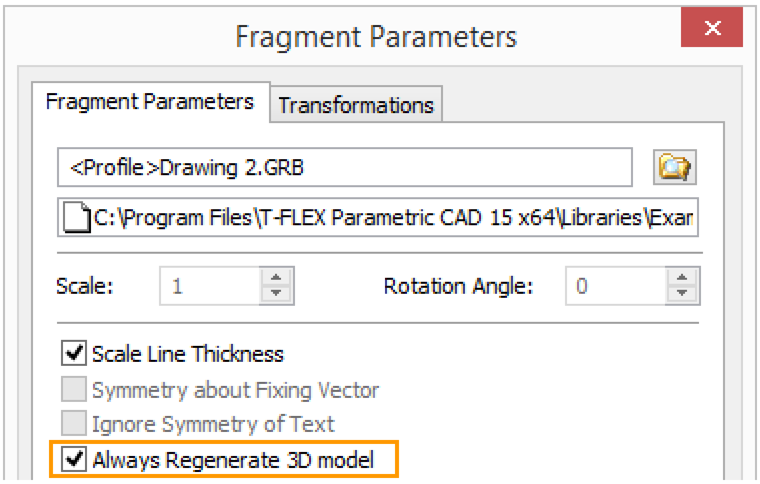
Used with *frag* element. Specifies name of inserted fragment page:

Example:

{Frag Source="<Fitting_v2>Sketch.grb" page=”Page_2”/}

Attribute regenerate 3d

Used with *frag* element. Activates flag of the inserted 2D fragment.



Example:

{Frag Source="<Fitting_v2>Sketch.grb" page=”Page_2” regenerate_3d=”true”/}

Attribute recursive_template

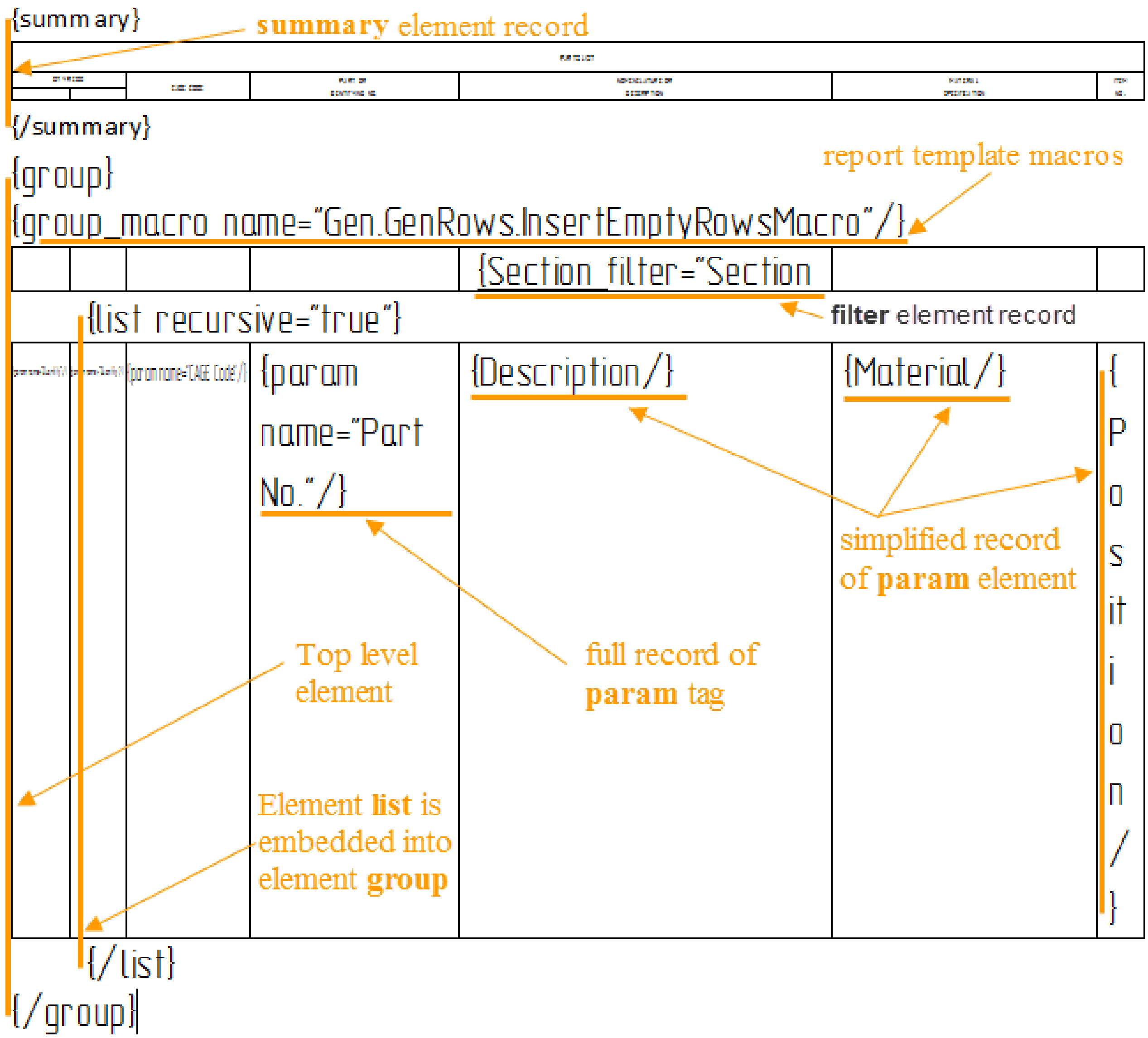
The attribute is used in *group* elements.

The attribute is important for representations, which consider hierarchy when grouping.

Example:

recursive_template="true"

Template example



Variant Report/BOM Generator (TFPSCadVersionReportGenerator)

The generator is a specialized version of standard (TFPSCadVersionReportGenerator) generator described above. It is used to generate reports for variant product structures.
The main differences are:

- *Container* element is used instead of *group* element;
- *Param* element has special syntax.

EXAMPLE OF REPORT TEMPLATE CREATION

Report template is a GRB file, which describes report format and data in file according to the certain rules. You need to create a table in the report template file using **TE: Create Text** format. Structure of the report is described in its cells.

Creation of report template:

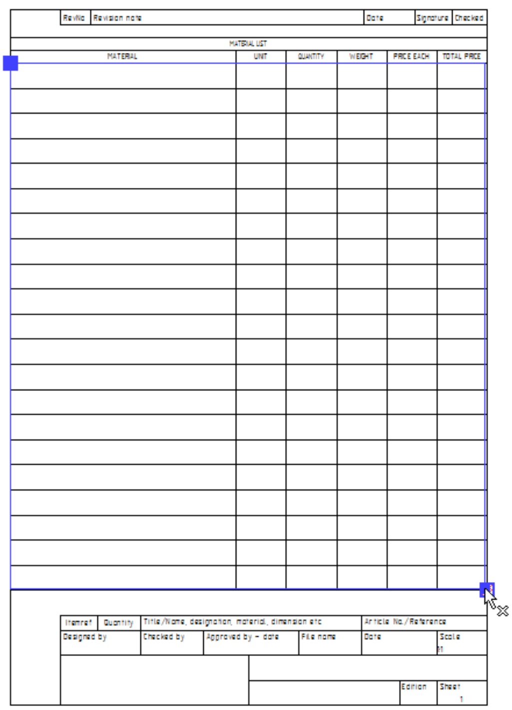
1. Create a new document based on “ISO Drawing” template and save it into the folder with templates. Path to report templates is set in **SO: Set System Options** command on the **Folders** tab.

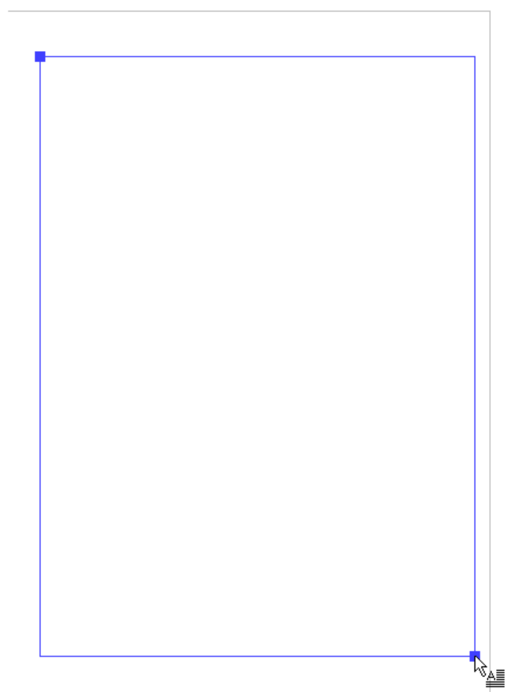
2. Set A4 format and portrait orientation on the **Paper** tab in the **ST: Set Document Parameters** command.

3. If the template will be used for creation of reports on a separate page or in another document, insert the 2D fragment of a title block, which contains a table header and lines.

If the template will be used for creation of reports on the current page, you can skip this step.

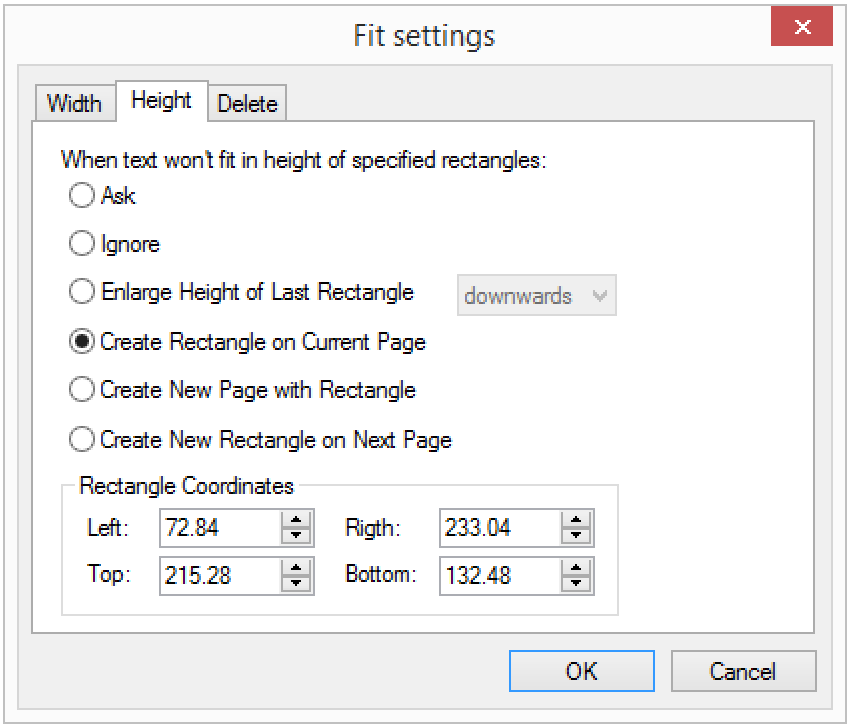
4. Create a paragraph text using command **TE: Create Text**. Attach it to the nodes of the title block fragment or in absolute coordinates.





Press after selection of attaching points. After that, you need to specify coordinates for new text rectangles using **Change paragraph size parameters** option from automenu. Records, which don't fit on the first page, will be moved to new rectangles.

If the template is for the current page, select "Create rectangle on current page". Note that its position should not coincide with the coordinates of the existing rectangle.



If the template is for a new page select "Create New Page with Rectangle". Its coordinates should coincide with the nodes on the following BOM or report pages.

You should set rectangle coordinates manually in the **Rectangle Coordinates** section.

Press **[OK]** button.

5. Add a table header into the created paragraph text.

If the template is for a separate page, you can skip this step.

Header should be between *{summary}* and *{/summary} tags*. You should create a table with column names there. The table is filled in manually.

{summary}

Description	Units	Quantity	Weight	Price

{/summary}

6. Add tags *{group}* and *{/group}* for group of records output.

{summary}

Description	Units	Quantity	Weight	Price

{/summary}

{group}

{/group}

7. Create a table for a group header. Name of the BOM section is a header of the group in standard BOM.

> Note! Total width of all tables should match with the first table.

Add *{Section/}* tag to a column. It will display name of the section, which is used for grouping records in the report.

{summary}

Description	Units	Quantity	Weight	Price

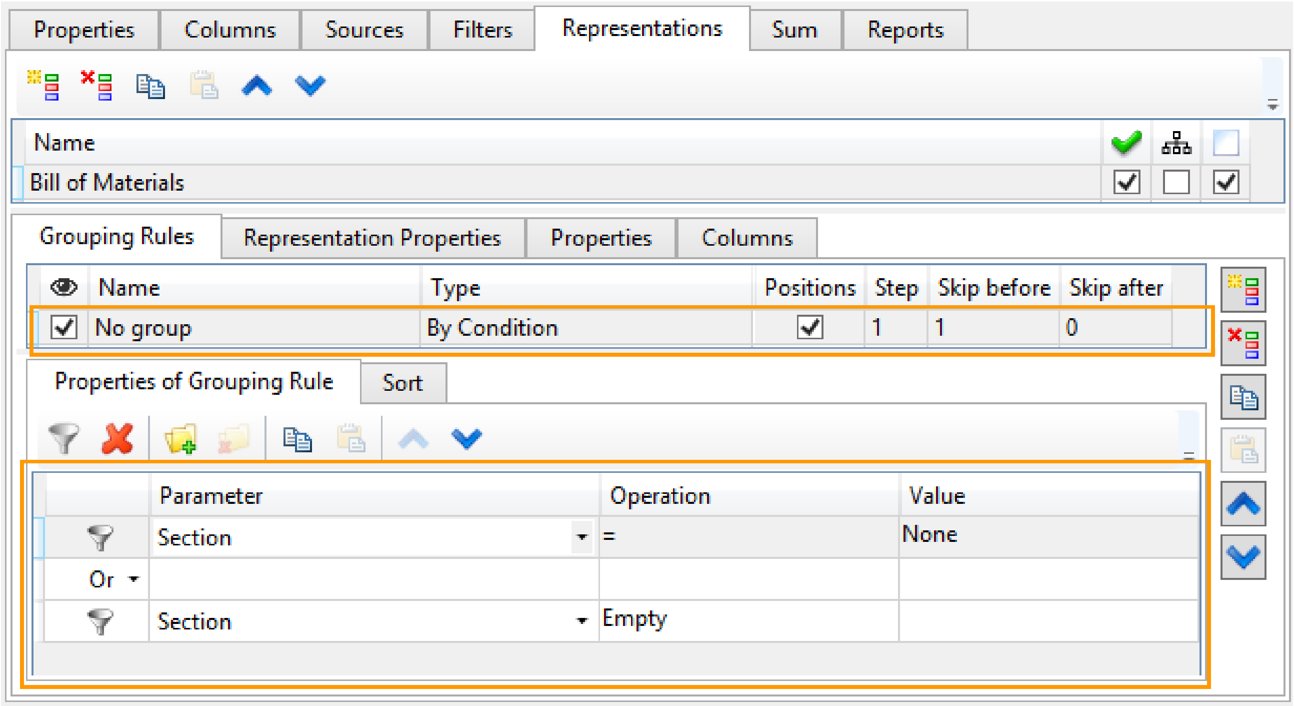
{/summary}

{group}

			{Section/}	

{/group}

8. If a group header has no text (for example, if it is a group of records without section), the table will be empty and should not be displayed in the report. You need to add a condition. First, you need to find name of the corresponding group in the product structure. Its name is “No group”.



You need to add the filter *{group hide_table="Group_name = 'No group'"}* which will replace the opening tag of the group.

{summary}

Description	Units	Quantity	Weight	Price

{/summary}

{group hide_table="Group_name = 'No group'"}

			{Section/}	

{/group}

9. Add table for new BOM records. Add *{list}* and *{/list} tags.*

{summary}

Description	Units	Quantity	Weight	Price

{/summary}

{group hide_table="Group_name = 'No group'"}

			{Section/}	

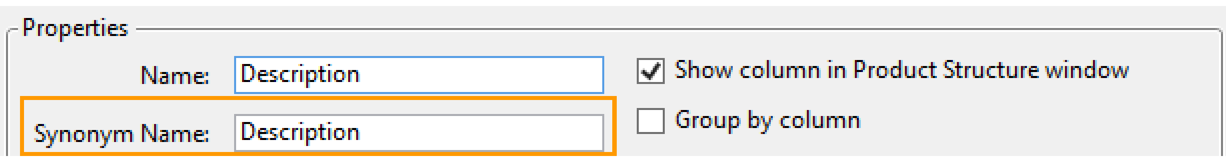
{list}

{/list}

{/group}

10. In the third table, you should specify names of product structure columns. Data from the columns will be added to the report.

If there are no spaces in the column name, you should type *{column_name/}*. You can use a synonym name instead of the name of the column.



If there is a space in the column name, use the following record:

{param name="Column name"/}

Example of filling of the third table cells, which do not have additional filters:

{summary}

Description	Units	Quantity	Weight	Price

{/summary}

{group hide_table="Group_name = 'No group'"}

			{Section/}	

{list}

{Description/}	*{Units/}*		{Weight/}	{Price/}

{/list}

{/group}

You can manage value output using filter. For example, the following filter allows you to display only the non-null field values in the column:

{Quantity filter="Quantity != 0"/}

{summary}

Description	Units	Quantity	Weight	Price

{/summary}

{group hide_table="Group_name = 'No group'"}

			{Section/}	

{list}

{Description/}	*{Units/}*	{Quantity filter="Quantity != 0"/}	{Weight/}	{Price/}

{/list}

{/group}

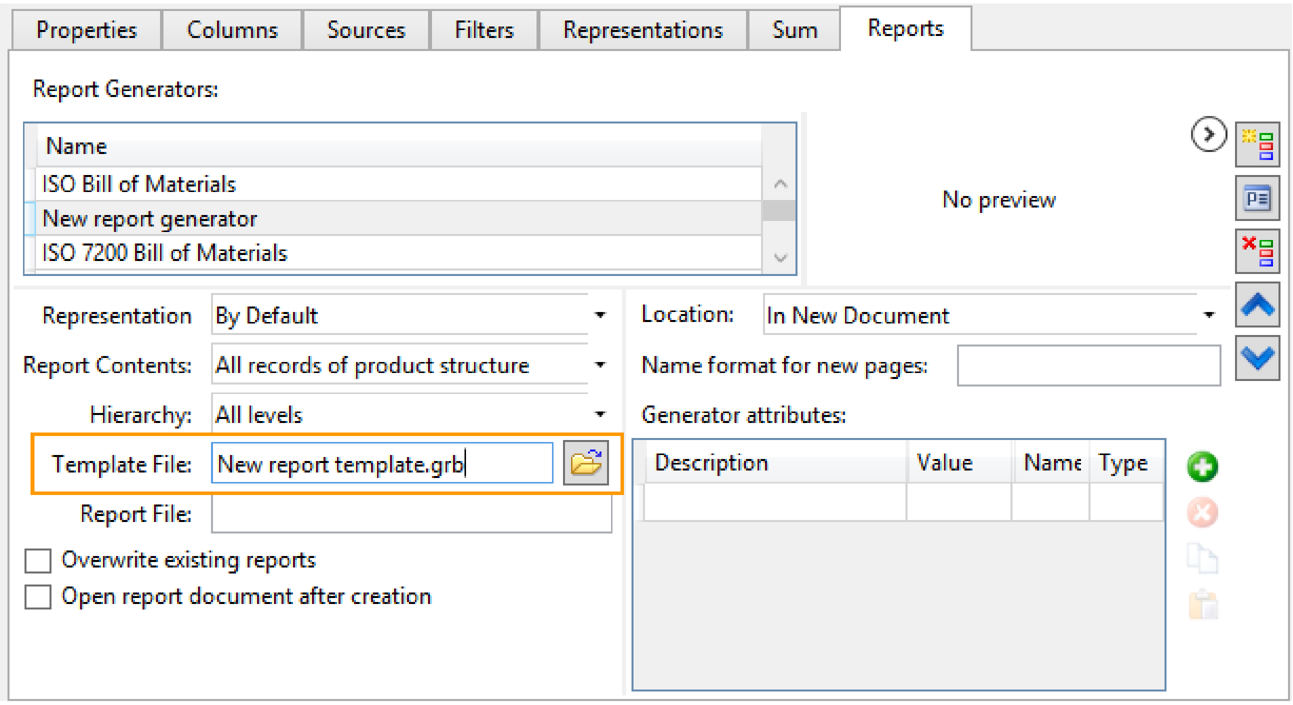
Creation of the simplest report template is finished. Finish paragraph text edition using  and save your document.

> Note! Text of report template should fit paragraph text rectangle. It should not go beyond it by height or width.

Report template should be used for a report generator.

> Report generator properties are described in "Reports" section of "Product structure types" chapter.

If a new report generator is created for a product structure type, not for a product structure in the document, it can be used in all documents where the product structure type is used.



> It will be necessary to update product structure properties for the existing documents according to the properties of the updated product structure type.

Additional Features

✓ Add a summation result to the report.

For example, if you need to display total weight use *{outcome name="Weight" out_operation="numeric_sum"/}* record in a separate table between *{summary}* and *{/summary}* tags.

{/list}

{summary}

	Total Weight:	*{outcome name="Weight" out_operation="numeric_sum"/}*

{/summary}

{/group}

✓ Formatting of the value in the column may be performed using *argument* parameter.

For example, the following record will display the summation result with accuracy of two decimal places:

{outcome name="Weight" out_operation="numeric_sum" argument="F2"/}

✓ You can display variable value in a report using *variable name* element.

For example, {variable name="$Date"/}.

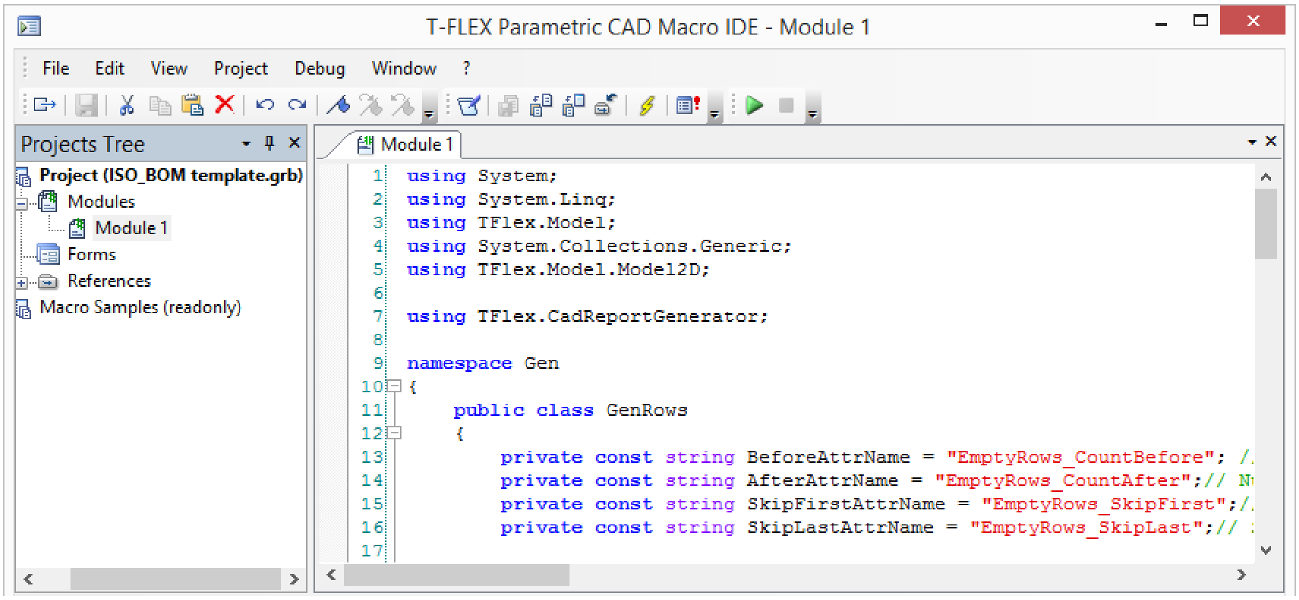
The variable should exist in the file based on which the report is created.

✓ You can display records of all levels for the representation using *recursive* attribute.

{list recursive="true"}

> You can find description of elements and attributes for report template in “Report/BOM template” section.

✓ You can add macros. Additional data processing can be performed using a macro, which may be added to the report template file. You can add and edit a macro in the **Macro editor**.



A macro can process product structure records or data, which is displayed in a table cell.

In the first case a macro is called using *{group_macro name="Name "/}. Name* is a name of the macro, which should be run before the table is filled in.

You can use *filter* attribute, when you call a macro. The macro will be applied only for groups that correspond the condition.

Example: {group_macro name="Gen.Gen.GroupMacro" filter="Section = 'Industry Standard Parts '"/}

The macro for processing of textual value in the table cell is used in *param* and *outcome* elements. The macro is called using *{param name="Column name" str_proc_macro="Name"/}. Name* is a name of the macro for string processing.

Example: *{param name="Quantity$$00" str_proc_macro="Gen.Gen.XAmount"/}*

WORKING WITH BOM ON PROTOTYPES

Mechanism of BOM creation used in the earlier program versions will be described below. Unlike BOM on templates mechanism, BOM on prototypes mechanism is left inside the program for compatibility reasons only and is not recommended for usage for the new projects due to its limitations.

This mechanism also provides tools for automating bill of materials (BOM) composition in the formats recommended by various standards and in customized user-defined formats.

To work with this mechanism you need to use the set of commands from **Tools > Reports / BOMs** text menu or options from **BOM** main toolbar.

You can use standard BOM prototypes or custom prototypes when creating BOM.

The prototype is a BOM with empty records. It defines table contents and the rules for its formatting. The prototype also defines text type for the BOM. The text can be of two types:

- Multiline text (text is not translated to another line cause of growing strings number);
- Paragraph text (text can be translated to a new page or a new area on the existing page cause of growing strings number)

User defines columns number and their data in BOM properties. You need to select standard fields for the BOM table from the list and to create additional BOM table fields. Information for the table is taken from fragment files or is inserted manually.

For automatic filling of the BOM table, it is necessary that:

- BOM data is defined in the **Product structure** window of the fragment documents (otherwise the string of the table that corresponds the fragment without data will be empty);
- The parameter **Include in product structure** in the fragment properties is set to one of the options:
    - **Without Embedded Elements** – the fragment data is included into the product structure. The elements that are embedded in the fragment drawing are not included;
    - **With Embedded Elements** - the fragment data and the embedded elements data are included into the product structure;
    - **Embedded Elements Only** – only embedded elements are included into the product structure.

Each assembly fragment corresponds to the table record when you create a BOM. Additional strings are added manually when the table is edited. For manual editing you need to unselect  **automatic field** option for a cell/column/string in which you need to enter a new data.

Text formatting parameters, strings height and columns width are specified in the table properties.

The table header is a fragment, which contains text with the header table. That is why you need to make corresponding changes in the headers table if table columns content or columns width were changed.

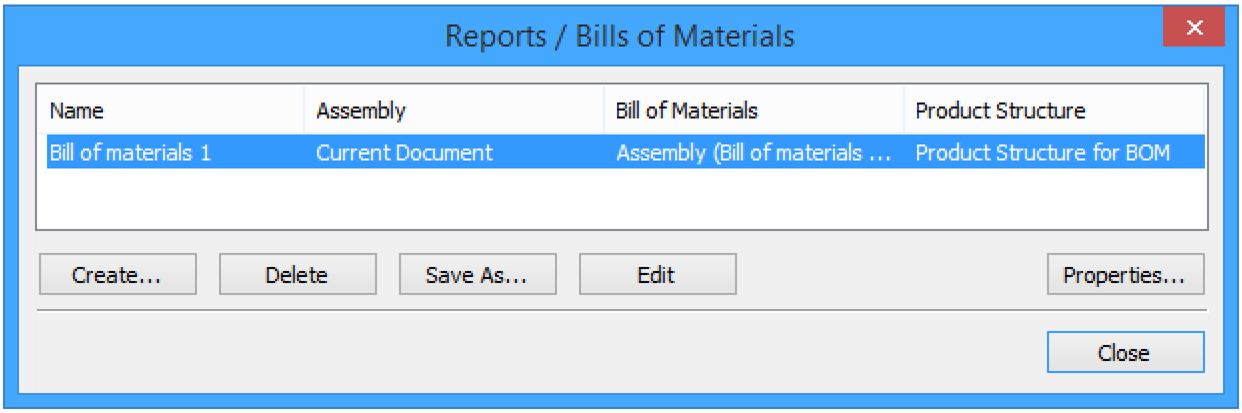
BOM sections structure is stored in the database file. The name and the path to the file can be specified in **SO: Set System Options** on the tab **Bill of Materials** (“BOM groups.mdb” file is specified by default).

One can view the list of existing BOMs in a document by using the command **BM: Reports/Bills of Materials**:

Icon	Ribbon
	Bill of materials → Bill of materials → List
Keyboard	**Textual Menu**
<BM>	Tools > Report/Bill of materials > Reports/Boms List

Upon calling this, the dialog box appears on the screen with the list of all reports and BOMs existing in the document. This dialog box allows creating a new BOM, deleting an existing one (by selecting from the list) or viewing its properties.

It also allows renaming a BOM file if it was created as a separate document. If such BOM was selected, the graphic button **[Save As...]** becomes enabled, allowing the name and folder change.



Note! It is required the separate document file and the current document file to be saved.

To create a new BOM, use the command **BC: Create Bill of Materials**. The command can be called by one of the following means:

Icon	Ribbon
	Bill of materials → Bill of materials → New...
Keyboard	**Textual Menu**
<BC>	Tools > Report/Bill of materials > New...

Upon calling the command, the dialog box "Create report/Bill of Materials" will come up on the screen. Use this dialog to create the BOM table either in a new document or as a new page in the current document. This command also allows creating a new prototype based on an existing one.

The delete BOM from the current document, use the command **BX: Delete Bill of Materials**:

Icon	Ribbon
	Bill of materials → Edit → Delete
Keyboard	Textual Menu
<BX>	Tools > Report/Bill of materials > Delete

Should multiple BOMs exist in the current document, a dialog box comes up for selecting the BOM to be deleted.

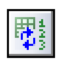
Another command is **BI: Include in Bill of Materials:**

Icon	Ribbon
	Bill of materials → Product structure → Included fragments
Keyboard	Textual Menu
<BI>	Tools > Report/Bill of materials > Included fragments

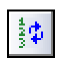
The command is intended for maintaining a binding between BOM and respective fragments. Call to this command results with the dialog box that comes up on the screen and contains the list of all existing fragments in the document. The controls provided by this dialog let you set or modify the method of inserting any fragment into the BOM.

The BOM table and callouts can be updated following the assembly modifications by the command **BRA: Update Bills of Materials and Callouts**:

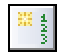
Icon	Ribbon
	Bill of materials → Edit → Update All
Keyboard	Textual Menu
<BRA>	Tools > Report/Bill of materials > Update All

Keyboard	Textual Menu	Icon
<BRA>	Tools > Report/Bill of Materials > Update All	

Should the callouts position numbers change in BOM (as, for instance, a result of changing the rules of sorting records), the command **BRP: Update BOM Callouts** lets updating the callouts in the assembly:

Keyboard	Textual Menu	Icon
<BRP>	Tools > Report/Bill of Materials > Update Callouts	

To set callouts in assembly, use the command **BL: Create BOM Callout**:

Keyboard	Textual Menu	Icon
<BL>	Tools > Report/Bill of Materials > Callouts	

For editing an existing BOM, use the command **BE: Edit Bill of Materials**:

Keyboard	Textual Menu	Icon
<BE>	Tools > Report/Bill of Materials > Edit	

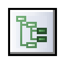
The command allows modifying the contents and parameters of the BOM table (see the section “Editing BOMs”). If several BOMs exist, the dialog for BOM selection appears first.

Next is the command **BT: Toggle: Assembly/BOM Pages**:

Keyboard	Textual Menu	Icon
<BT>	Tools > Report/Bill of Materials > View Assembly/BOM	

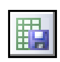
The command allows switching between the assembly and the BOM, if the BOM is located on another page or in a separate document. If several BOMs exist, the dialog appears first for selecting the BOM to switch to.

The command **BG: Edit BOM Sections**:

Keyboard	Textual Menu	Icon
<BG>	Tool > Report/ > Bill of Materials > Sections…	

This command allows modifying a database file containing information about BOM sections (the default file is "BOM Group.mdb"). The name and the path to the database sections can be defined/modified by the command **SO: Set System Options** > **Bill of Materials** (see the section “Editing BOM sections”).

Command **BS: Export BOM**:

Keyboard	Textual Menu	Icon
<BS>	Tools > Report/Bill of Materials > Export…	

This command allows you to export the BOM data or the BOM itself to the Excel file (xls). The export is carried out by a special macro located in the folder *“…/Program/BOMExport”*.

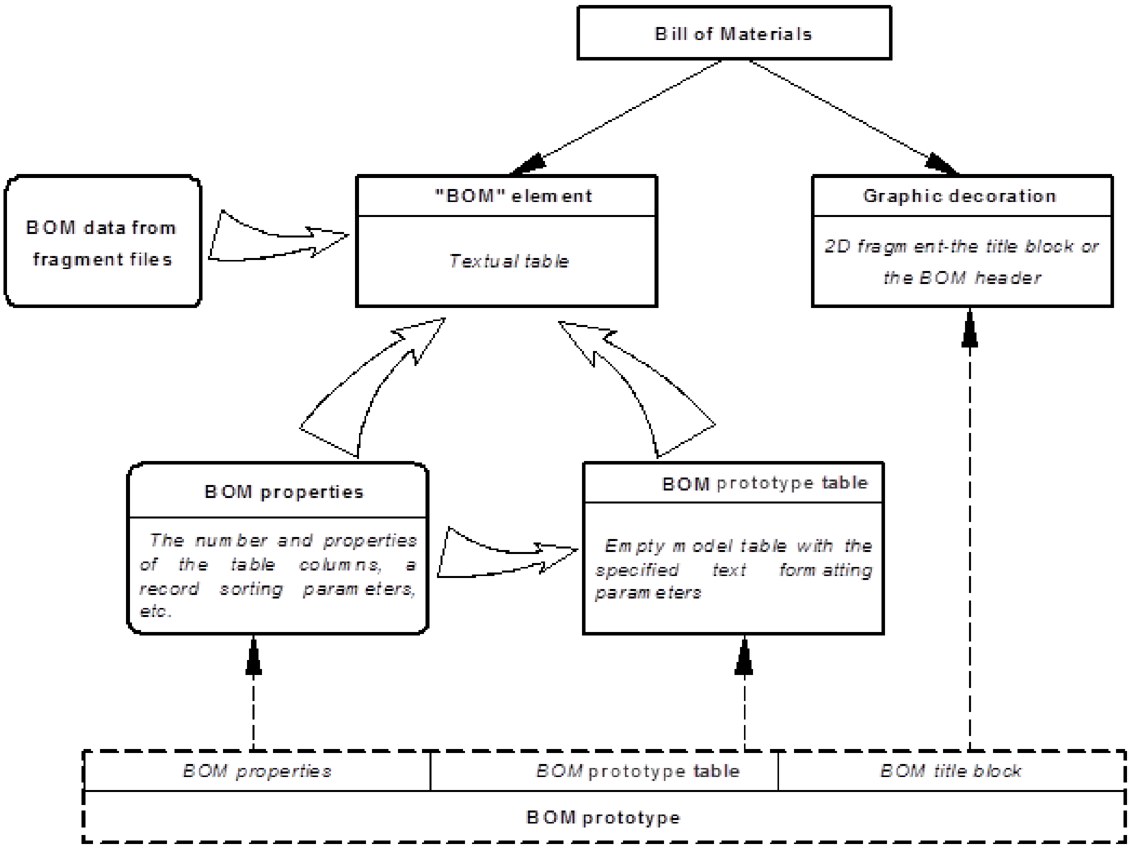
BOM Composition

Any BOM is composed of the following elements:

- BOM prototype table is a textual table with a certain number and type of columns. The table properties define formatting of the BOM columns. The table is created based on the

"Text" element type ("Paragraph text" or "Multiline text"). The text parameters define the behavior of the BOM as it is being filled with information: the direction of the BOM "growth", the capability of continuation on the new page. Thus, for instance, a prototype based on a "Multiline text" template element does not support multipage BOMs.

- A fragment with the title block image or just the BOM header (depending on the BOM type).



Interaction between BOM prototype table, title block and BOM table header depend on the type of the BOM. If the BOM is located on the drawing page, then the table fragment is used that describes the image of the table header with the column titles only. Such fragment is described in the BOM properties as the table header or footer. Since the table height is unlimited in this case and depends on the number of records in the BOM, the table grid lines are defined in the template properties as the table boundaries. Besides, automatic continuation to the new page is not supported for BOMs created on the drawing page.

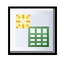
If the BOM is located on a separate page or in a separate document, the table size and appearance are predefined and do not depend on the actual number of records in the BOM. In this case, the fragment is inserted in the BOM page that contains the full image of the BOM title block, including the header, the main text and the table grid. The table boundaries are not specified in the template in this case. What is required in this case is that the distance between the horizontal lines of the table grid in the fragment was equal to the height of the template cells. In this case, the template (the textual table) is applied "over" the fragment and is tied to its nodes.

Such BOM is allowed to occupy several pages. Once the table is filled up on one page, the BOMs continuation is automatically created on the new page. The BOM title block fragment inserted in the new page is defined in the BOM prototype under the command **ST: Set Document Parameters**, the **Fragments** tab.

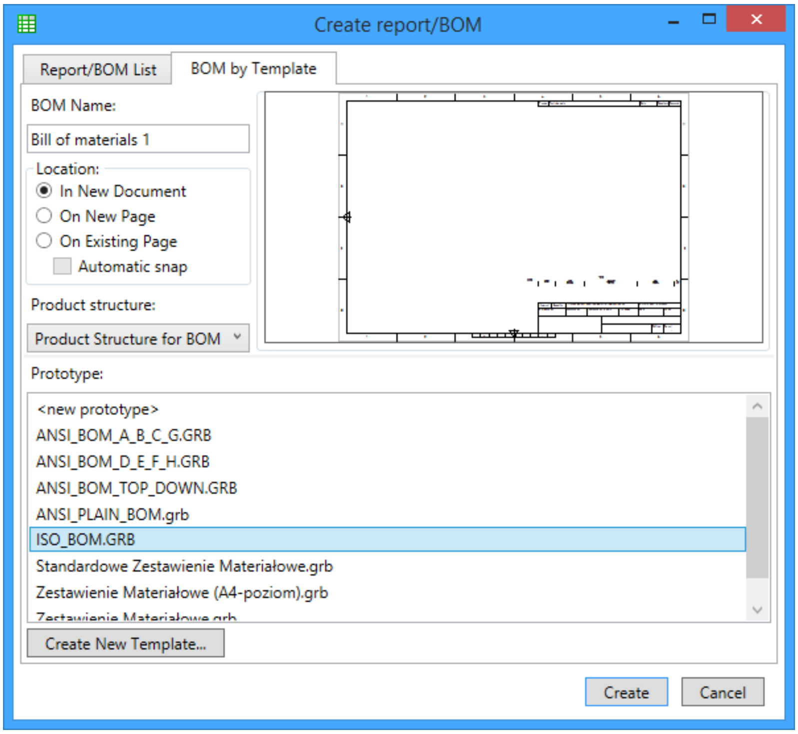
When creating a BOM, the template and the title block are created automatically according to the selected BOM prototype. The BOM prototype is a document containing the empty BOM only. It has the BOM prototype table (empty paragraph text with a table of certain type and formatting), BOM properties and the title block fragment. The drawing settings define which title block fragment to insert when continuing on the new page (for automatic creation of additional BOM pages). All these data are obtained from the mentioned prototype when creating a BOM. In future, when working with the created BOM, the user can manually change the BOM template, its properties and the title block.

Creating BOM

To create a BOM based on prototype, use the command **BC: Create Report/Bill of Materials**:

Keyboard	Textual Menu	Icon
<BC>	Tools > Report/Bill of Materials > New...	

Upon calling the command, the dialog box "Create Bill of Materials" will be displayed. Use the controls provided by the dialog box and create BOM table either in a new document or on a new page of the current document, or else on the current drawing page.



When creating a BOM, one can select a BOM prototype describing the structure of the columns and sections of the table being created. The list of prototypes is provided in the **Prototype** pane.

To create an individual BOM tables and new BOM prototypes, use the **“<Empty Template>”** template.

System offers BOM templates in various standards that are based on the paragraph text functionality. The “Preview” pane allows previewing the image of the selected prototype.

When creating a BOM on menu or the current page, it is assigned a unique name “Bill of materials 1”, “Bill of materials 2”, etc. The name of a BOM created as separate document includes the name of the respective assembly. For example, if the assembly is named “Assembly 1”, then its BOM created in a new document will be named “Assembly 1 (Bill of materials 1)”. The latter name can be modified in the command **BM: Bills of Materials**.

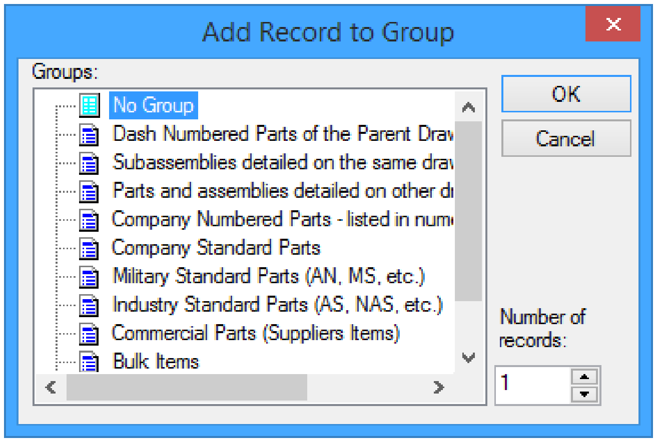
> When creating a BOM, we recommend setting the parameter ‘**Transparent**’ **Text editing** (**ST**: **Set Document Parameters** command, **View** tab), that allows entering BOM editing mode by simply clicking **one of the records.**

Creating BOM on New Drawing Page or New Document

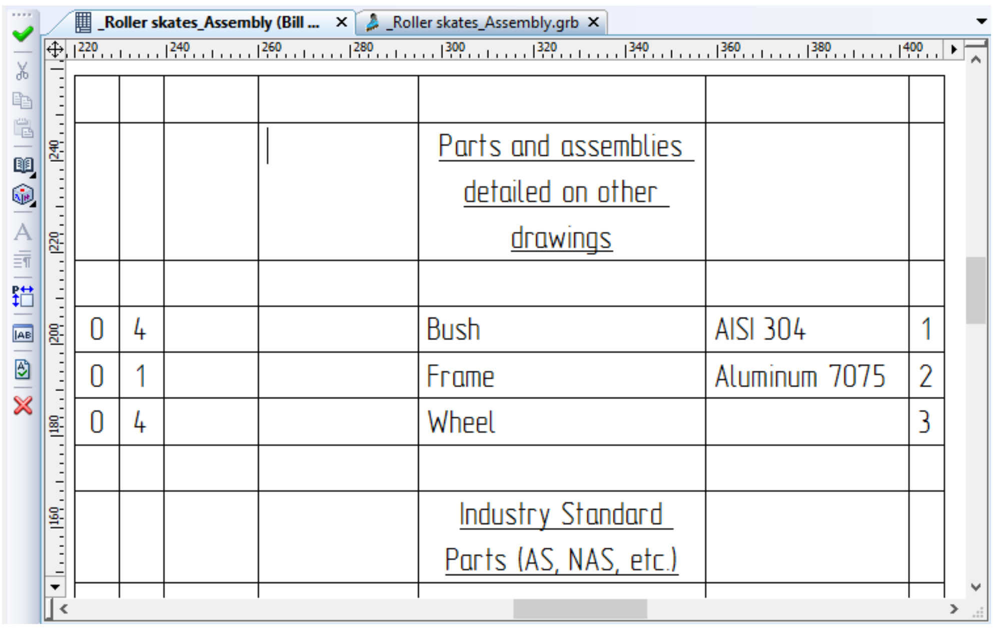
Call the command **BC: Create Report/Bill of Materials**. In the coming up dialog box set parameter **On New Page** and select a prototype (for example, “ISO_BOM.grb”). Press **[OK]**. As a result, a new page will be created in the document, displaying the BOM being created in editing mode (subsequent BOM pages are also created automatically).

If any data was specified for inserted fragments and they were included in the assembly product structure, this data will be added to the product structure after its updating. The data will be automatically added to the BOM.

In the cases, when the data was not defined or relation with fragments was not established, an empty BOM is displayed on the screen. In this case, the dialog box **Add record to section** will be displayed.



This dialog box appears any time when editing an empty BOM. By using this dialog, you can add the desired records to any of the sections. If the specified group does not exist in the BOM, it will be created automatically. You can abort BOM creation by pressing the button **[Cancel]**.



An edit mode is activated after BOM creation.

Display options and sorting rules of the BOM table will be taken from the prototype.

More information about BOM editing can be found in section “Editing BOM”.

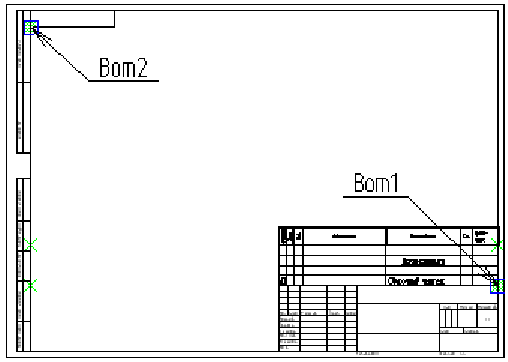
BOM creation is confirmed by option:

	<F5>	Finish input

Alternatively, you can right click, with the pointer being outside the BOM table area.

Creating BOM on Existing Drawing Page

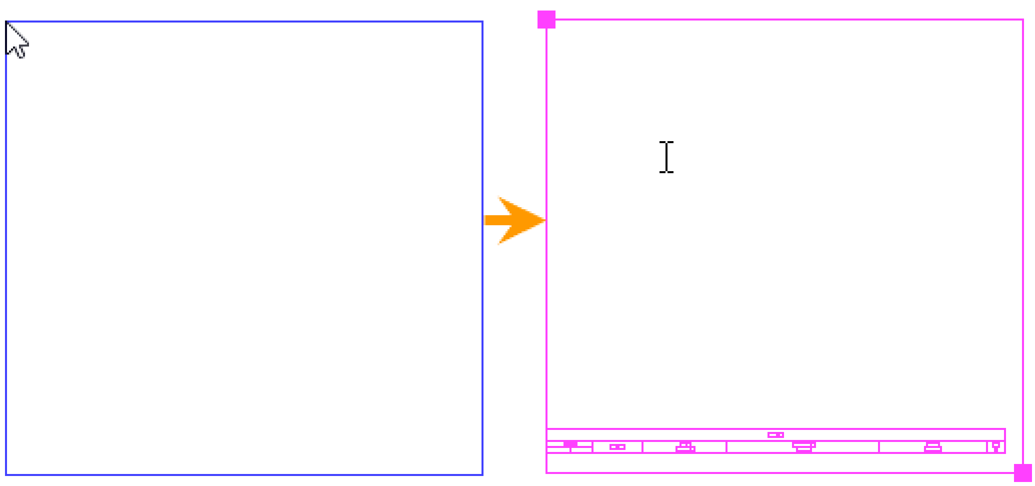
Call the command **BC: Create Report/Bill of Materials**. In the coming up dialog box set parameter **On Existing Page** and select the desired prototype. To automatically position the BOM prototype on the drawing, check the optional parameter "Automatic snap" (recommended). Usually, such BOMs are created on the drawings with their title blocks already in place.



If using the automatic snap, the system will automatically search for snapping nodes when creating the BOM. The standard title block fragment has the default nodes defined, with the reserved names "bom1" and "bom2". In this case, the BOM will be positioned above the title block. If such nodes do not exist on the title block, the system will search for them among other drawing elements. You can create such nodes manually. What is important is that the nodes are not aligned vertically nor horizontally, with the horizontal distance between them sufficient for fitting the BOM prototype in between.

For a BOM prototype based on a multiline text, just one node, "bom1", is sufficient. If the nodes not found, the system displays the respective message and switches to the mode of manual snapping point input.

In the case of manual BOM positioning, a box rubber bands on the screen defining the BOM boundaries. Position the box as desired and click . Note that in this case the BOM table is Lower Limit-justified and will grow upwards as records are added.



For confirming the BOM creation use the option:

	<F5>	Finish input

Creating BOM from Empty Prototype

By selecting the prototype named <*Empty Template*>, you will be independently creating a BOM based on an empty drawing. This is different from creating a new prototype (see the section “BOM prototype”) in that the parameters and BOM properties being defined will be used in the current document only. An empty prototype-based BOM can be used as a prototype for creating other BOMs only by saving the current document in the folder “...`\Program\Template\BOM`”.

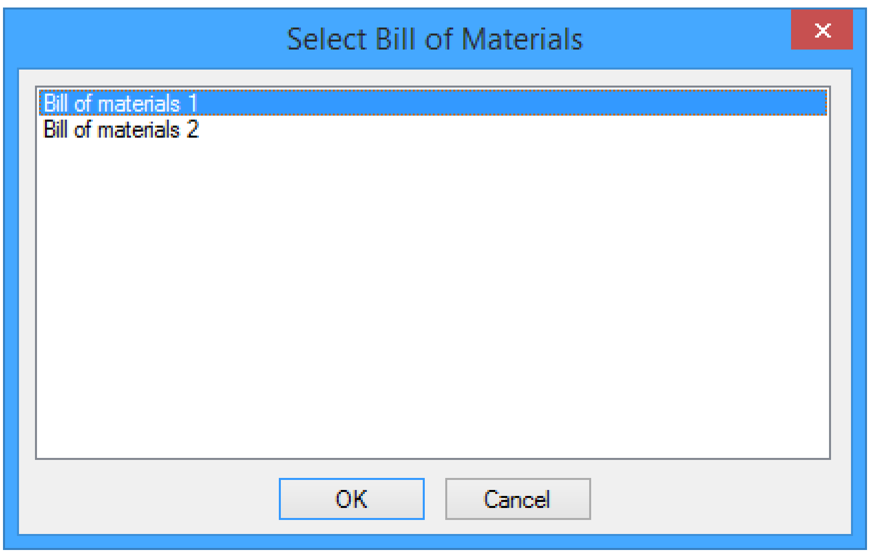
Switching Between the Assembly and BOM Document

The command **BT: Toggle: Assembly/BOM Pages** allows opening a BOM from the current assembly that is located on a different page or in a different document:

Keyboard	Textual menu	Icon
<BT>	Tools > Report/Bill of Materials > View Assembly/BOM	

If the document contains several BOMs the dialog box **Select Bill of Materials** appears.

Select the BOM to open and press **[OK]**. The selected BOM document will open. The returned to the assembly, called the command **BT: Toggle: Assembly/BOM Pages** again.



When creating a BOM in a separate document, there is yet another way of switching to the assembly drawing. You can select the item **Assembly** in the BOM context menu.

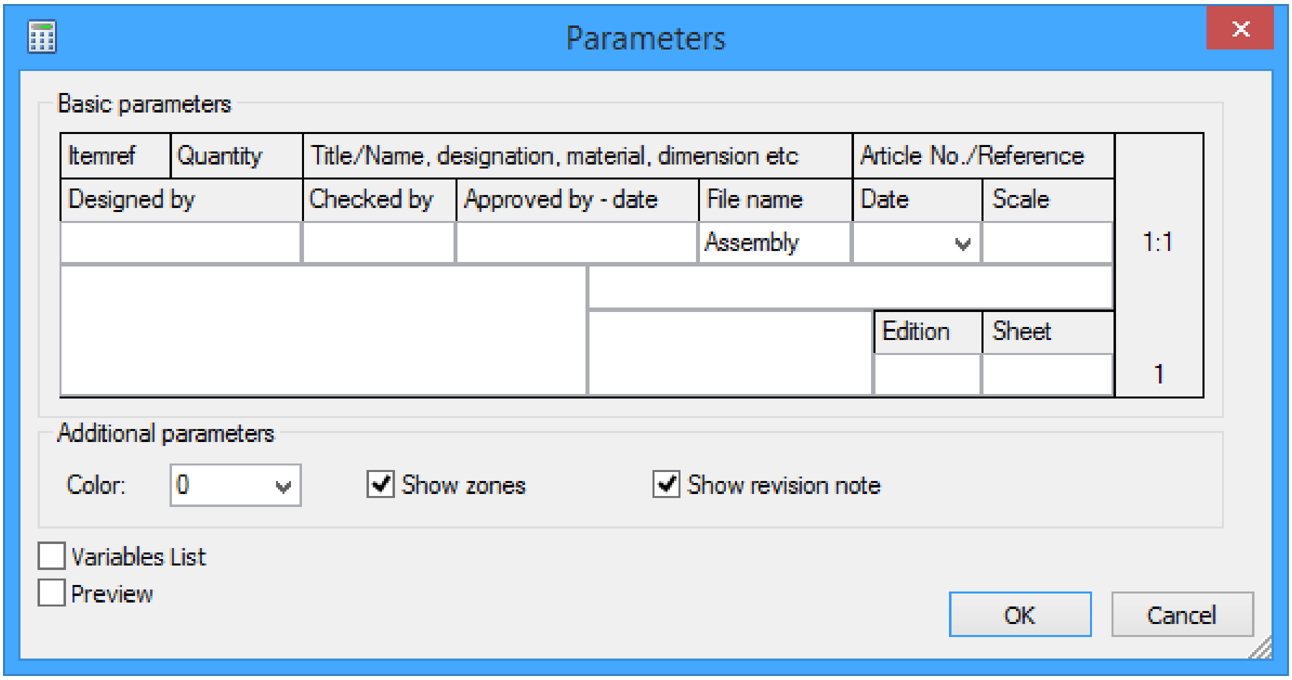
> If several BOMs exist in the document, one of which is placed on the first page of the assembly drawing, then any other BOM created on a separate page or in a separate document will have to be opened manually.

Filling in BOM Title Block

A BOM title block can be filled in by two ways:

1. Since the BOM table title block is a fragment, the title block can be filled using this fragment variables. To do this, call the command **EFR: Edit Fragment**.

Then select the BOM table fragment with the mouse. A dialog box will come up on the screen representing the title block of the BOM page that you selected. Alternatively, this dialog box can be called as follows. While the system is in the command-waiting mode, move the pointer over the BOM table fragment and right click . In the coming up menu, select the item "Variables".



Fill in all required entries of this dialog box. Upon confirming by pressing the graphic button **[OK]**, the records from this dialog box will be entered in the title block of the BOM page you have selected.

2. Another way of filling in the title block is by direct input in the drawing, without calling the fragment variables. To do so, set the textual pointer inside the title block box to be filled, and click . Start typing the text as soon as you see the blinking cursor in the specified box. The button with an arrow appears to the right of the box allowing selecting values from the list. In usually, the list is empty (except for the entries in the "Date" column). If it is necessary to fill in the list for selecting values from the list in future, fill the box with the desired text, and then in the context menu called by right clicking , select the item "Add Value to List".

BOM PROPERTIES

While in the BOM editing mode, the properties dialog box of the BOM being edited can be called using the option:

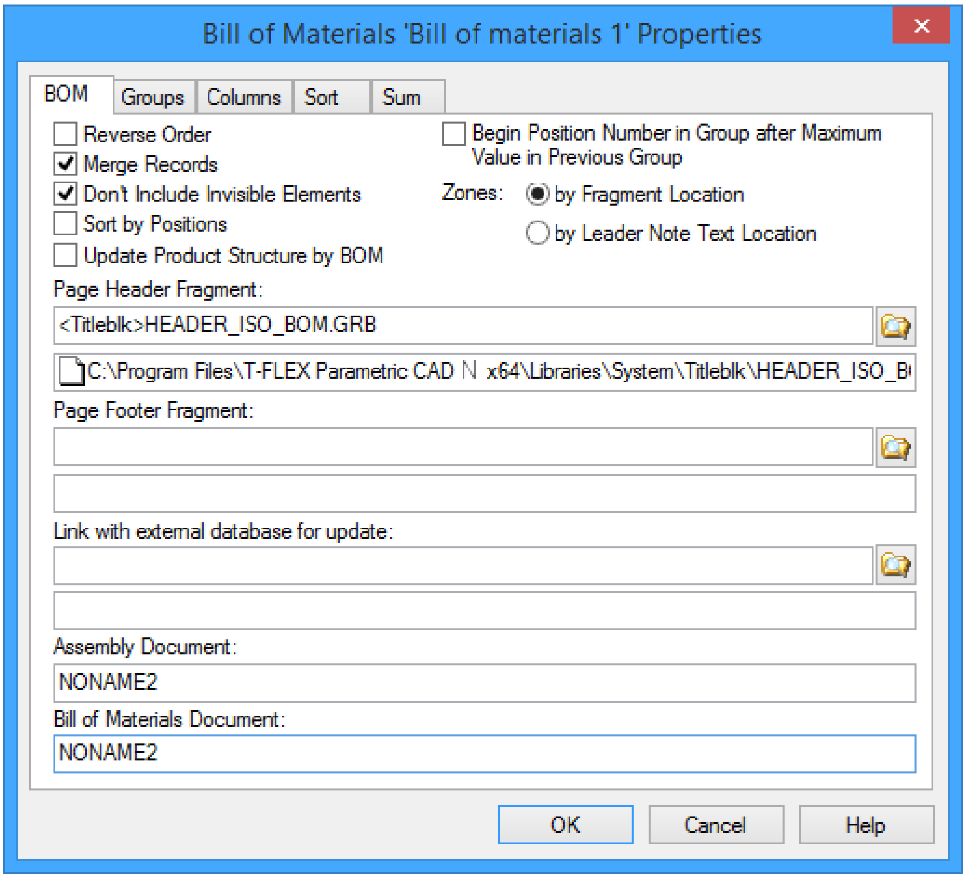
		BOM properties

This option is located on the main toolbar (button set "BOM").

The BOM properties dialog box can also be called by different means. For example, it can be accessed from the dialog box of the command **BM: Reports/Bills of Materials**. Alternatively, in the command – waiting mode, move the pointer over one of the current BOM records and right click, and then in the coming up context menu select the item **BOM Properties**. If this item is not available in the context menu, then first select the element **BOM** by using item **Other**.

"BOM" Tab

Reverse Order. With this flag set, the records in the BOM table will be coming in the reverse order that is starting with the row with the highest ID number. The order of records in the table is defined by the sorting rules (see below).



Merge Records. Allows merging BOM rows with identical records.

Example: suppose, the same fragment is inserted into assembly several times. In this case, the data are compared, and since there is no difference, the records of these fragments are merged in one row, as directed by the flag.

Don't Include Invisible Elements. When this flag is on, BOM will contain only visible fragments. Fragments invisible on the drawing (e.g. hidden with levels) will not be added to BOM.

Sort by Position. If BOM contains explicitly defined positions, when this flag is set, records in BOM tables will be sorted according to those positions along with the usual sorting rules.

Update product structure by BOM. When the flag is enabled, the BOM data is included in the product structure.

Begin position number in section after maximum value in previous section. Setting this flag will bring the following behavior: if any BOM section contains explicit positions with numbers greater than assigned automatically, numbering in the next group will begin from the values following the maximum explicitly specified number.

Zones. This parameter defines the zones when using the automatic zone definition mode (see the section "Preparing data for BOM table"). Takes the following values:

By Fragment Location. The zones will be defined by the fragment location.

By Leader Note Text Location. The zones will be defined by the location of the leader note callout that marks the fragment position.

The following two parameters define the name of the document containing the table header image, used in the prototype file:

Page Header Fragment. This parameter entry specifies the path to the file containing the BOM table header. (The case of using the header).

Page Footer Fragment. (The case of using the footer).

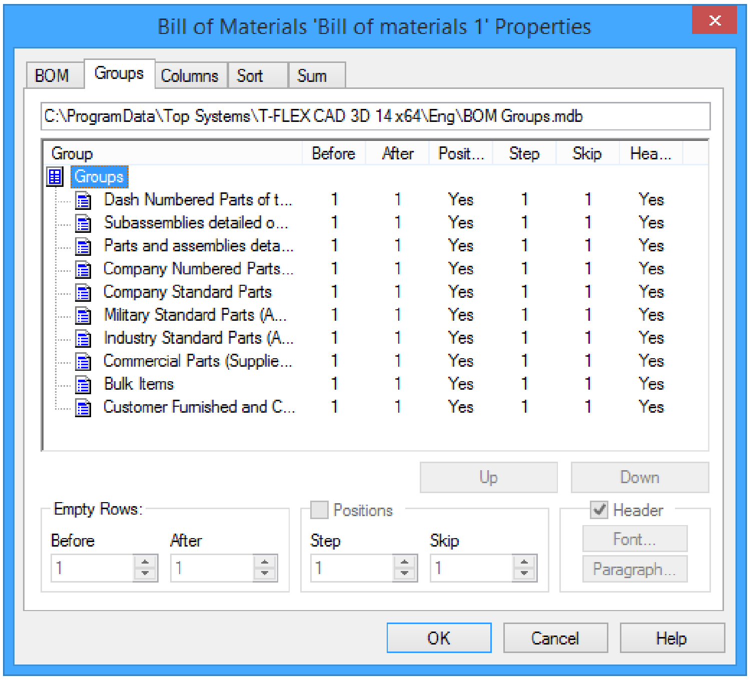
Link with external database for update. This parameter is used for connecting with a BOM based on an external database. Data exchange between the BOM and the database occurs only upon updating the BOM.

The parameters **"Assembly Document"** and **"Bill of Materials Document"** specify the path to the file or files containing the assembly and the BOM documents.

"Sections" Tab

A box in the upper portion of this tab displays the path to the active sections database.

This tab contains a table with the list of sections displayed as a tree, in its first column, and the rest of the columns displaying the parameters of the respective sections. The set of sections can be modified in the command **BG: Edit BOM Sections**. The order, in which the sections are listed in this table, will be maintained as the sections are inserted in a BOM. For example, in the case represented by the above table, the group “Military standard parts” upon inserting in a BOM will be located after the sections “Company standard parts”, and before the group “Industry standard parts”.



The existing group order can be changed using the graphic buttons **[Up]** and **[Down]**.

To modify group parameters, select this group in the list. The parameters of the active group will appear in the entries in the lower portion of the tab. You can input the desired values of the following parameters:

Empty Rows: Before and After. Define the number of empty lines before and after the group header. These lines serve as separators, not allowing entering records.

Positions: Controls the group positions appearance. With this parameter set, positions will be set. If the parameter unset, this group will be ignored when setting the positions.

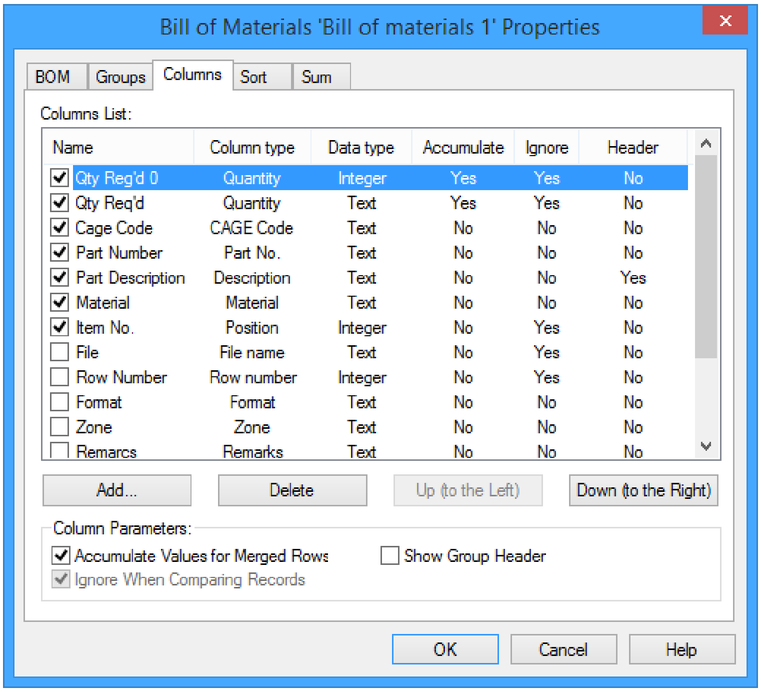
Step. Defines the step of setting the positions in the group.

Skip. Specifies the number of positions to skip for the beginning of the group. The number of the first position in the group will be equal to the number of the last position in the previous group plus the amount defined by this parameter.

Header. This parameter defines, whether the group header title is displayed in the BOM. The graphic button **[Font]** is used to call the dialog box for defining the font parameters of the group header title. The button **[Paragraph]** calls the respective dialog box for defining various parameters for group title formatting (see the chapter “Text”).

“Columns” Tab

This tab contains the list of columns and their respective parameters. The columns are inserted in the BOM table that are checkmarked in the box before the column name. The top-down order of columns in the list defines the column position in the BOM (left to right). A column position can be modified using the buttons **[Up (to the Left)]** and **[Down (to the Right)]**. Remember, however, that changing a column position will not affect the position of the respective column in the BOM table template, nor change its width.



The correspondence between the fragment data and the standard BOM table columns is defined by the type of the column, rather than its name.

The following are the only parameters of the current column allowed for modifications: **Accumulate Values for Merged Rows**, **Ignore When Comparing Records**, **Show Group Header**.

When inserting the same fragment several times into the assembly, its data are compared, and, since identical, entered into the BOM as one record (in the case when the parameter **Merge Records** is set on the tab **BOM**). In this case, it is necessary that the records be summed up in the column “Quantity”. This behavior is supported by the following two parameters.

Accumulate Values for Merged Rows. With this parameter set, the numerical figures will be summed up when merging the positions.

Ignore When Comparing Records. With this parameter set, the columns will be excluded from comparison (that is considered equal). If the parameter **Accumulate Values for Merged Rows** is set, then this parameter is also set automatically.

Show Group Header. The column with this parameter defined will display the group header.

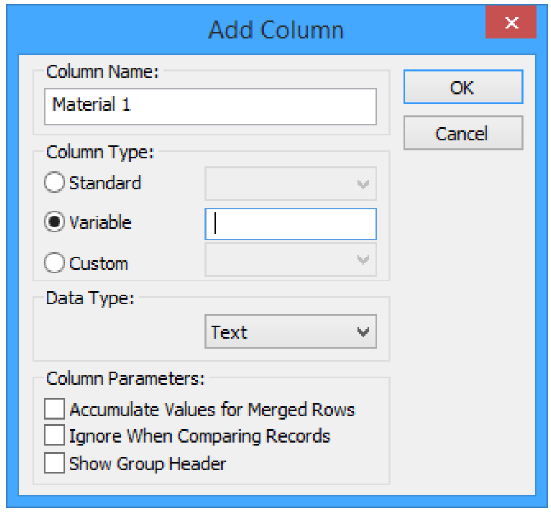
In text of the united records with flags **Accumulate Values for Merged Rows** and **Ignore When Comparing Records** it is not possible to use variables that change their values depending on the assembly parameters. Such variables will be replaced with constants on uniting.

The list of columns can be modified using the graphic buttons **[Add...]** and **[Delete...]**.

The graphic button **[Add...]** is used for creating a new column. It brings up a dialog box for defining all necessary parameters for the column being created.

Column Name. This entry is for specifying the column name.

Column Type. Defines what fragment data will be entered in the column being created.



Standard. Presents the standard BOM data. The entries are copied from the respective section of the fragment BOM data. The data entry name for the BOM is selected from the combo box on the right hand side.

By default, the column name coincides with the name of the selected entry. If necessary, the user can define an arbitrary column name. The BOM table can include two columns with the same standard BOM entry (for example, one can have a column presenting the weight of one part, and another column presenting the combined weight of several parts). If the list of columns contains records with coinciding entry names, the system of box a warning message.

Variable. Such a column will be displaying the value of the fragment variable. The variable name is entered in the box on the right hand side. The default column name will be the string "Variable 'variable_name'", however, if necessary, the user can specify an arbitrary column name. Only those boxes in the column will be filled in, that correspond to the fragments with the namesake variables defined. If the variable data type mismatches the data type of the BOM column being defined, the system automatically converts the data types.

Custom. Selecting this parameter provides access to the list of names for additional BOM entries. The list displays only the names of the entries that are added to the installation package (the file "*...\Program\BOM Custom Data.txt*").

If necessary, the file containing the list of additional BOM entries can be manually edited, with one's own data included in it. The thus defined entries will be displayed in the list of additional entries of the above-described parameter.

When you select the name from the list, the fragment data that was input in a respective cell of the **Product structure** window will be entered in the column.

When entering an arbitrary column name, there are two possibilities:

- If the data was defined under the same name in the fragment documents, then the created column will contain the respective data from the fragments;
- If the user creates a column with a unique name, then its data has to be entered manually. To use this feature, select the column while in the BOM editing mode and clear the default option “Automatic Field”.

The **Data Type** combo box offers one of the following values:

Integer. Integer numbers only.

Real. Real numbers only.

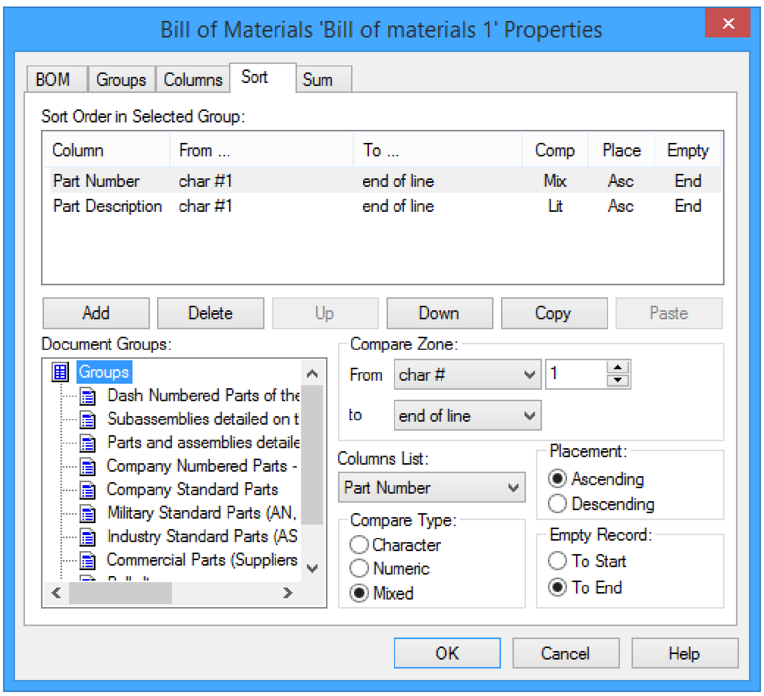
Text. Arbitrary textual information.

The delete a column, select it in the list and press the graphic button **[Delete]**.

“Sort” Tab

This tab is used for defining and editing the sorting rules for the BOM Records.

By default, this tab reflects the sorting rules specified in the BOM prototype that was used for the BOM creation.



The sorting rules are defined for each BOM section separately. Those appear as a set of conditions for comparing the contents of the selected BOM columns.

The pane **Document Sections** defines the BOM section, for which the sorting rule is being defined or modified.

The sorting defined for the section "Sections", affects the records that do not belong to any BOM section.

The rule itself is displayed in the entry **Sort Order in Selected Section** in the form of a table, in which all the conditions are collected, that will be used for comparing lines in the section concerned. The order of applying the sorting conditions corresponds with their order in the table. If sorting rules are not defined for a section, then the records in the table will be placed in the order of the entering in the database.

The diagram shows that in the example the row comparison in the "Sections" group will be done by two columns: "Part Number" and "Part Description". First, the lines will be compared in the column "Part Number", since this column appears first in the table. If several lines in this column will be identical, then their further comparison will be performed based on the records in the column "Description". The order of comparing the BOM lines by columns can be changed by altering the record positions in this table. This is done using the graphic buttons **[Up]**, **[Down]**.

To create or modify existing rules for sorting a section, do the following steps:

1. Select the desired section from the BOM section tree in the **Document Sections** pane. The list of columns with the currently defined set of sorting conditions will then be displayed in the **Sort Order in Selected Section** table.
2. Select the column whose sorting conditions are to be modified. The specified sorting condition settings will appear in the parameter fields on the right from the sections list.
3. Defines the required settings of the sorting parameters:

Columns List. Defines names of columns.

Compare Zone. Defines the excerpt of the table entry content to be used in comparison.

From. Defines the start of the excerpt to be compared.

Char # - the number of the character position (example: from the third character in the entry).

Substring – the number of a subsequent string occurrence in the entry (example: from the first occurrence of the string "ISO").

Char # from end – the number of the character position, counted from the end of line.

Substring from end - the number of the subsequent string occurrence in the entry, counted from the end of line (example: from the first occurrence of the string "ISO" from the end of line).

To. Defines the end of the excerpt to be compared.

Number of chars - defines the position number of the last character in the excerpt (example: until the character number 10 in the entry).

Substring - the number of the subsequent string occurrence in the entry (example: until the first occurrence of the string "-").

End of line – to the End of line.

Compare Type.

Character. Realizes comparison by ASCII characters (the contents of the two table entries are compared as two strings of characters). If the first character in the record entry is a number, the record is considered empty and is moved to the start or end of the group as prescribed by the parameter "Placement". If the several such records, their relative positioning will be random.

Numeric. Compares numbers (the contents of the two table entries are compared as two numbers). If the first character in the record entry is a letter, the entry is considered empty.

Mixed. If a non-numerical character appears on the first position of the first entry in the comparison, then the system identifies the substring of characters starting from the first position and compares it with the respective substring found in the second entry. If the second entry does not begin with a non-numerical character, then the first entry sub string is compared against the empty string. Next, the numerical substrings are identified, and the respective numerical values of the two entries are compared, starting from the first position after the last character string used in the previous step of the comparison. Thus, the contents of both entries are analyzed until reaching the end of the first entry. If a digit is found in the first position, the contents of the entry are considered an empty string of characters.

Placement. Defines the order of records placement. Case-sensitive.

Ascending. Begin with the first letter in the alphabet and the smallest number.

Descending. Last letters in the alphabet and largest numbers come first.

Empty Record. If empty lines exist in the group, intended for inputting records, this parameter will define whether these lines will be moved **To Start** or **To End** of the group.

To add a sorting condition in the table, first, unselect the existing records in the table. Next, set the necessary parameter values on the right of the list of sections, and press the **[Add]** graphic button. Alternatively, select one of the existing conditions in the table, then add it to the same table by the **[Add]** button and edit its settings.

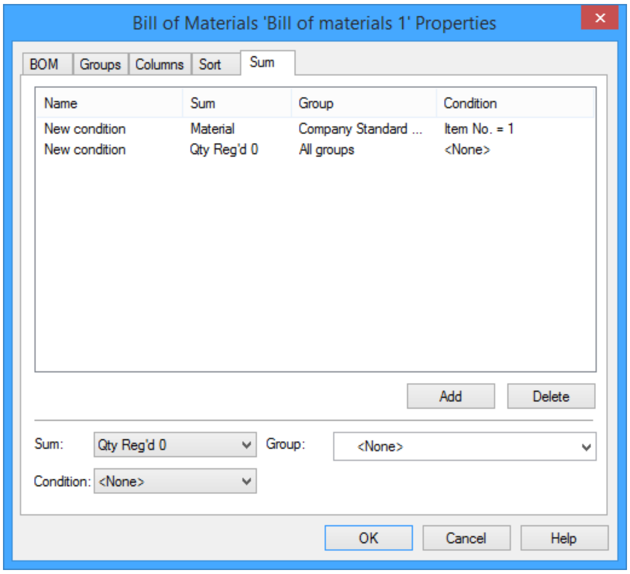
The graphic button **[Copy]** allows copying one of several lines of the table with the sorting parameters defined to the clipboard, and then pasting them in another group. To select all lines, hold down the <Shift> key, several rows with the <Ctrl> key.

To delete one of the sorting conditions from the table, select this condition and press the graphic button **[Delete]**.

With the "Automatic Update"  mode set, the new records added the BOM table are instantly placed in the position defined by the specified sorting conditions (except the manually edit records).

"Sum" Tab

This tab is intended for defining the rules of summing up the data in BOM columns for evaluating the bottom line. A specified rule is not recorded in a database; rather, it becomes an attribute of the text and can be used in future for evaluating the bottom line of a BOM column.



As an example, select the text used for creating some BOM table, and right click. From the coming up menu select the command "**Measure…**". The element with the name of the summing rule will be present in the list of the text attributes.

If necessary, you can create a variable with the value equal to the sum, calculated based on the specified rule.

To define a new rule for summing up a column, specify the following parameters:

Sum. Specify the target column of this summing rule.

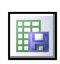
Section. Specify the BOM section to apply this rule to. The "<None>" setting means the rule will be applied to all records in the table.

Condition. Defines limitations on the BOM table records to be included in the bottom line evaluation. Select a column, a comparison condition and a value to compare the entry against. Selecting "<None>" means the condition is applied to all records in the table.

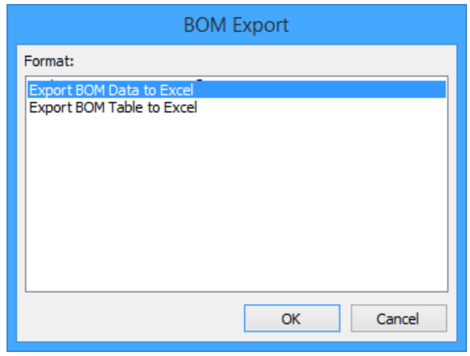
To input the defined settings, press the graphic button **[Add]**. The new rule will appear among the list of rules, named by default "New condition", with the summing parameters defined. The graphic button **[Delete]** deletes the current record from the list of summing rules. To modify a whole name, point at the entry "Name" of the desired column and double-click. The rule name will then be focused on, so that you can edit it.

BOM Export

For exporting BOM data or the BOM itself to the Excel file (xls), the command "**BS: Export BOM**" is used:

Keyboard	Textual Menu	Icon
<BS>	Tools > Report/Bill of Materials > Export…	

After calling this command, the dialog window with the list of export options appears on the screen. From the list the user can select **Export BOM data to Excel** and **Export BOM table to Excel**.



When exporting the BOM table, only the contents of the BOM table is transferred to the Excel document. The sequence and the composition of columns in the table, the row sequence, etc. are retained.

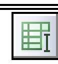
When exporting the BOM data to the Excel document, all internal data, on the basis of which the BOM was created, is transferred. All sorting rules and rules for combining the records, used in the BOM, are ignored. The result of the export operation will be different from the initial BOM appearance in the program document.

If the original program document contains several BOM, the result of the export operation will be the multipage Excel document. Each BOM is exported to a separate page (sheet).

> The export is carried out by macro located in the folder *"…/Program/BOMExport"*. If necessary, for exporting BOM to different formats, the user can add his own macro to this folder.

EDITING BOM

When creating a new BOM, the BOM editing mode activates automatically. To activate this mode for an existing BOM, simply move the pointer to the BOM area and click (this will work in the case when the transparent text editing setting is activated for the page, by the command **ST: Set Document Parameters** on the **View** tab). Alternatively, you can call the command **BE: Edit Bill of Materials**:

Keyboard	Textual Menu	Icon
<BE>	Tools > Report/Bill of Materials > Edit	

Besides, you can enter BOM editing from the context menu. To do so, move the pointer over the BOM text and right click . In the coming up menu, select the item **Edit Contents**. Any of these efforts lead to entering the BOM editing mode.

On the main toolbar of the program (if its state is not locked), the button set "BOM" which has options for editing Bill of Materials will appear.

The automenu options are intended for inserting various elements and formatting the text of the BOM. The text can also be formatted by using the system toolbar. Details of these options and the system toolbar were provided in the description of the command **TE: Text**, the sections "Paragraph Text" and "Multipage Text". If after the text formatting it is necessary to restore the formatting defined in the BOM table default template, simply highlight this text and press the option:

		Default Format

To edit the BOM table template, use the option:

		Edit Template

For the system will enter the table creation command. On the main toolbar the button set "Table" will appear with the help of which editing the template (see the chapter "Text").

If the following option is turned on the main toolbar (in the button set "BOM"):

		Automatic Update

Then, the BOM will be updated automatically (updating the table accounts for the specified sorting conditions). This mode is recommended; this is why the icon is turned on by default. If the automatic updating hinders the operation, this icon should be turned off. In this case, the BOM records can be updated manually, using the icon:

		Update

Remember, that the sorting conditions are not applied automatically when saving the BOM.

To update the BOM outside the editing mode (as, for instance, after assembling fragments), one can use the context menu. To do this, position the pointer over one of the BOM records and right click. In the coming up menu, select the item "Update BOM".

The Automatic Field option is turned on for all records into the BOM automatically from the inserted fragment data, as indicated by the pushed icon:

		Automatic Field

This means, the manual input is disallowed in the fields of this record. If such a necessity exists, this option can be turned off. Remember however that the later modifications to the assembly do not affect the records created manually.

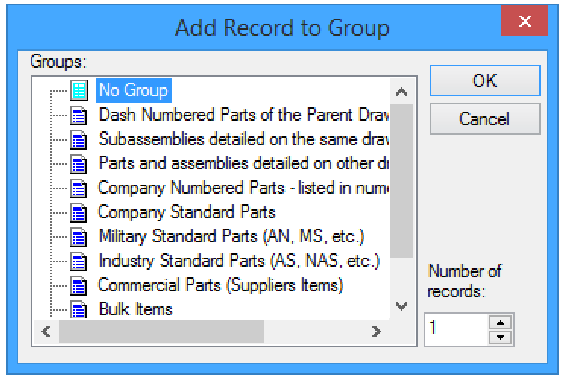
To manually input a record in the BOM table, first insert an empty row. To add a row to the current section, simply position the pointer at the row, before or after which the record should inserted, and press one of the following icons:

		Add Record Before
		Add Record After

As a result, an empty row will be inserted in the current section, allowing for entering a record manually. The position will be defined for the added record. Remember, that upon updating (the automatic update does not work in this case), the added rows will be moved according to the defined sorting parameters (see "BOM Properties").

To add a record to an arbitrary BOM section, use the command:

		Add Record into Section or Create Section



In the coming up dialog box select the section to which the record should be added, and enter the number of rows in the parameter “Number of records”.

Upon confirming, empty rows will be created in the section. This records are also accounted for, when setting the positions. Empty rows appear in the beginning or in the end of the section depending on the sorting rules.

If the specified section does not exist in the BOM, then the specified section header will appear in the BOM upon confirmation. This capability can be used when searching for a section in the case of multipage BOMs.

The delete a record, place the pointer over the record entry to be deleted and press the icon:

		Delete Record

If all records are deleted in the section, then the section header will also be deleted.

The deleted records still exists in the BOM, they are just not displayed. Those can be viewed using the option:

		Show Hidden Records

Upon turning on this option, all deleted records will be displayed in the BOM. The record entry is will be color-marked (the color can be modified in the command **Customize** > **Options**, the tab **BOM**, the parameter **Hidden Records Background Color**). To restore a deleted records, simply select it with the mouse and press the icon:

		Restore Hidden Records

To hide deleted records again, turn off the icon **Show Hidden Records**.

The hidden records that were entered manually are kept until the BOM is updated.

A record can be moved up and down by the options:

		Move Up
		Move Down

These options are only accessible when the sorting is turned off (see “BOM Properties”), and only for the records that can be a moved without violating the sorting conditions (such as identical records).

If the records should be moved in such a way, that one record always follow the other specific one, it should be tied to the previous using the option:

		Snap to Previous Record

To have a BOM row ignored when setting the positions, one should set the option for it:

		Skip Position

To insure that a record has the same position under any assembly or BOM modifications, set the option for this record:

		Lock Position

The options:

		Insert Empty Row Before Current Record
		Insert Empty Row After Current Record

Low editing rows, serving as record separators, before or after the record pointed by the mouse. Such rows are assigned the parameter “Automatic Field”. No text can be entered in such rows.

The accessed by the following option BOM properties dialog box allows defining the order of setting positions, decoration of the BOM section headers, column properties and sorting conditions (see the section “BOM Properties”):

		BOM Properties

BOM editing is completed using the automenu option:

	<F5>	Finish input

Deleting BOM

Deleting Whole BOM

To delete a BOM, use the command **BX: Delete Bill of Materials**:

Keyboard	Textual Menu	Icon
<BX>	Tools > Report/Bill of Materials > Delete...	

If several BOMs exist in the document, the system offers selecting the BOM to delete from the list.

When deleting a BOM, only the textual table of the BOM is actually deleted. The fragment title block stays in the drawing and needs to be deleted separately.

Creating and Editing BOM Prototype

A BOM prototype is a program document, containing only an empty BOM with specific defined properties. Since a prototype is not an assembly, its BOM does not have any records. The text formatting parameters and the tables defined in the prototype will drive the graphic shape of the BOM based on this prototype.

Should a BOM prototype need modifications, it can be opened as a common program document and be edited. One can also create one's own custom BOM prototypes and use them in future.

The files used as prototypes are stored in the folder whose name and path are defined by the command **SO: Set system parameters**, the tab **Bill of Materials**, the parameter **Template Folder**. The default folder setting is "...\Program\Template\BOM". The standard BOM table prototypes are stored there upon the system installation.

To create a custom BOM prototype, use the command **BC: Create Report/Bill of Materials**. In the coming up dialog box, select one of the prototypes in the BOM list. If the new BOM has just insignificant difference from a standard BOM, then the most closely matching prototype should be selected. To create a BOM, whose table differs in a relevant manner from the tables of the standard BOMs, use the "Empty Template".

Upon selecting a BOM prototype, press the graphic button [Create New Template].

Creating BOM Prototype Based on Existing Prototype

The first step will be defining the name of the new prototype (a new prototype is always created in a new document). By default, the document is created in the folder defined in the command **SO: Set System Options**, the tab **Bill of Materials** (by default, "...\Program\Template\BOM"). This insures the presence of the created document in the list of prototypes when creating a new BOM. After that, a window opens with the new document created based on the selected one.

To make modifications to the table header, open the title block or the BOM title fragment and modify the names of columns, then number and with as desired. Save the title block under a new name. Returned to the document of the prototype being created. If the prototype being created is intended for creating a BOM, that will occupy a separate page or separate document, specify the name of the edited title block in the properties of the title block fragment. In the case of a BOM prototype for positioning on the drawing page, modify the fragment name in the BOM properties. If the prototype provides for automatic continuation of the BOM to the new page, make the respective changes to the subsequent pages of the title block fragment, as specified in the parameter **Insert Fragment on new Page creation** in the command **ST: Set Document Parameters**, the **Fragments** tab.

Upon modifying the title block or the BOM header fragment, modify the properties of the BOM template as well. In the BOM context menu select the command "BOM Properties..." and enter the necessary changes (see the section "BOM Properties"). The BOM template will be automatically edited according to the introduced changes. The same can be done by calling the command **Tools > Report/Bill of Materials > Reports/Bills of Materials...** and pressing the button "Properties".

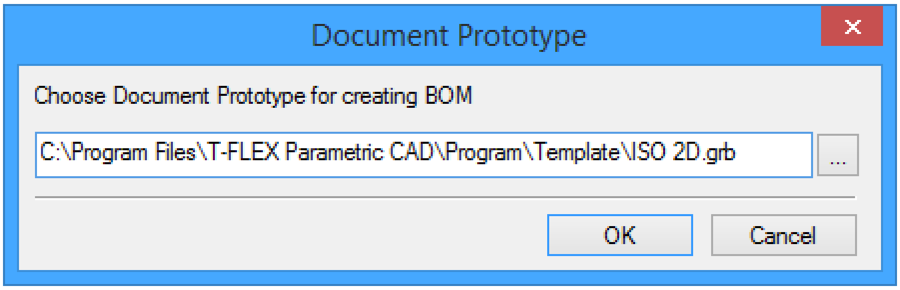
The size of the table cells and their contents formatting parameters is done by using the option "Edit Template", called in the BOM editing mode.

Creating BOM Prototype Based on "Empty Template"

When selecting the "Empty Template" as the prototype, BOM creation stars "from scratch", based on an empty drawing. In this case, the BOM table has to be created manually.

First, in a separate document create either the complete BOM title block with a header and the ruled BOM table, or just the table header, depending on the purpose of the prototype being created. Upon saving the created title block, one can proceed with creation of the BOM prototype itself.

Call the command **BC: Create Bill of Materials**. Select the “Empty Template” as the base prototype. Upon pressing the graphic button **[Create New Template]** a window appears for selecting a new document prototype.



Next, define the name of the prototype being created and its directory. The system will then open a new window with the created document, and the automenu will display the options for creating the BOM template text:

	<M>	Create Multiline Text
	<R>	Create paragraph text
	<P>	Set Text parameters
	<N>	Set relation with Node
	<Esc>	Exit command

Creating BOM title block

A prototype for a BOM on a separate page requires creation of a complete BOM title block including the title block text, header and grid lines of the BOM table being created. The table itself will be created with indivisible borders in this case.

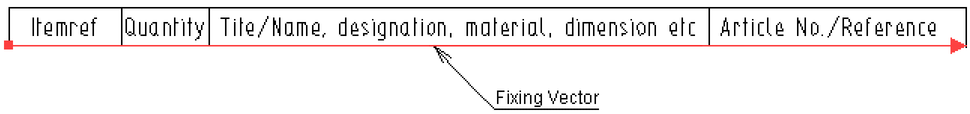
Fixing of the title block as a fragment in the BOM prototype document will be done in absolute coordinates. Therefore, it is not necessary to create fixing points or a fixing vector. All you need is just to position the title block drawing correctly with respect to the page borders.

The title block text entries are created based on “Text” elements, driven by the external variables in the document. To simplify entering the title block text, one can create custom dialog boxes.

To create a BOM prototype to be placed on an existing drawing page, all you need is to create just the table header. The grid lines will be defined by the borders of the table text, that should be made visible in this case.

A fixing vector needs to be created in the table header file, using two points. The fixing vector should pass along one of the horizontal borders of the created table. The fixing vector direction determines the position of the header as the BOM grows.

The fixing vector should be the directed from left to right in the case the left justification is set for the BOM table template, and the right to left if the right justification is set to for the template. When creating the BOM, the system will fix the header at upper or lower nodes of the text, reserving the necessary space for the header.



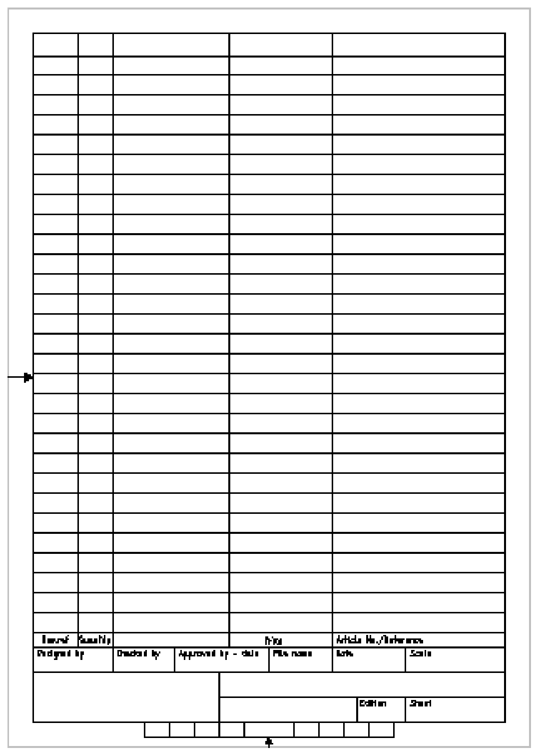
If the prototype is intended for creating multipage BOMs (usually, those are BOMs located on a separate page), one needs to create an additional title block for the second and the subsequent pages (also in separate document).

> As an example, one can review the standard BOM title blocks located in the library *"Standard"*, the folder *"Titleblk"*.

Creating prototype for BOM located on separate page

As an example, let's create a BOM prototype, whose table will have the following columns: "Item No", "Part Quantity", "Part Description", "Part Number", "Part Material", "Part Price".

Let's begin the prototype creation with creating BOM title blocks for the first and subsequent pages in separate program documents.



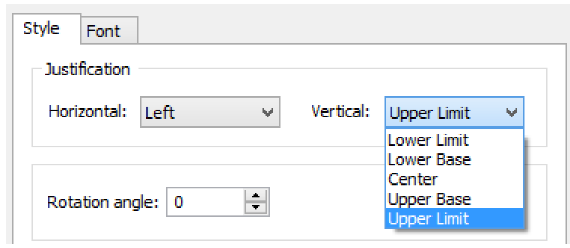
After that, one can proceed with creating the BOM prototype itself. Call the command **BC: Create Report/Bill of Materials**. In the coming up dialog box select "<Empty Template>" and press the graphic button **[Create New Template]**. After selecting the new document prototype and entering the name for the prototype being created, the system activates the text creation command.

The BOM table is created based on the paragraph text or multiline text. If you want the system to support automatic continuation of the BOM table onto the new pages as records are added (with automatic new page creation) or into a different area on the current page, then select the paragraph text. If you select the multiline text, then your BOM table will grow high without the capability of transferring it onto another page.

Select, for example, the option  **Create paragraph text**. You should set vertical justification among the text parameters of the table being created. This should better be done before entering the text.

Call the text parameters dialog box using the  option.

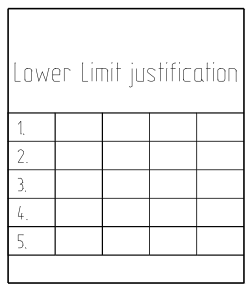
If you want your BOM to grow from bottom up as records are added, then set Lower Limit justification, to grow from top down use Upper Limit justification (Upper Limit justification is set by default). When creating a BOM prototype on separate page, the vertical justification is usually required to be the Upper Limit one.

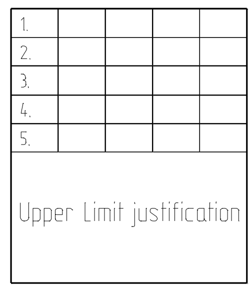


If you use the paragraph text, the rest of parameters can be left unchanged. When using the multiline text, additionally set the right horizontal justification.

When using the paragraph text, the vertical justification can later be changed by calling the properties dialog box for the rectangular text area (see the chapter "Text").

After that, we need to define the boundaries of the box where the created BOM table will be placed. To do this, specify two arbitrary points in the case of the paragraph text, or just one in the case of the multiline text.





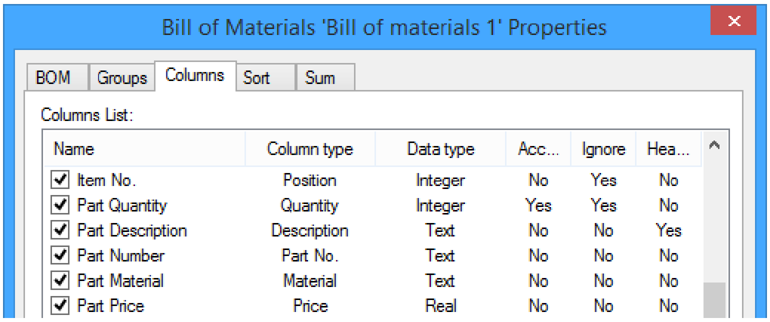
To continue with the BOM creation, click  inside the paragraph text or press . The BOM property window will appear on the screen (see the section "BOM Properties"). In the case of using the multiline text, the BOM property window will appear right after selecting the fixing node.

> Default BOM properties (names and types of the columns, the sorting rules and the sections) are copied from the file …*Program**BomStructure.mdb*.

The number of columns, their position and structure can be defined on the tab “Columns”.

The columns that are checkmarked in the table on this tab are the ones that will be included in the created BOM table.

In the example shown on the diagram, the BOM table will have the six check marked columns, their positioning left to right being determined by the relative positions of the checkmarked entries on this tab in the top down direction.



> Note that the number, names and order of the columns should correspond with the BOM title block.

The **Sort** and **Sum** tabs allow defining (or modifying default) sorting rules of the BOM records and summing up rules for its cells.

Upon confirming the specified properties, the system will activate the BOM table editing mode. The dialog box **Add Record into Section or Create Section** will appear on the screen. The BOM prototype should not contain any records; therefore, close the dialog window by pressing the graphic button **[Cancel]**.

To format the columns of the created table, enter the template editing mode. This can be done using the option:

	Edit Template

This option is located on the main toolbar (in the button set “BOM”). Upon calling the option, the system activates the table editing mode. On the main toolbar (if it is not locked), the button set “Text” will be shown. The options of this set allow changing the height, width and formatting of the columns. Besides, the following options become available in the automenu:

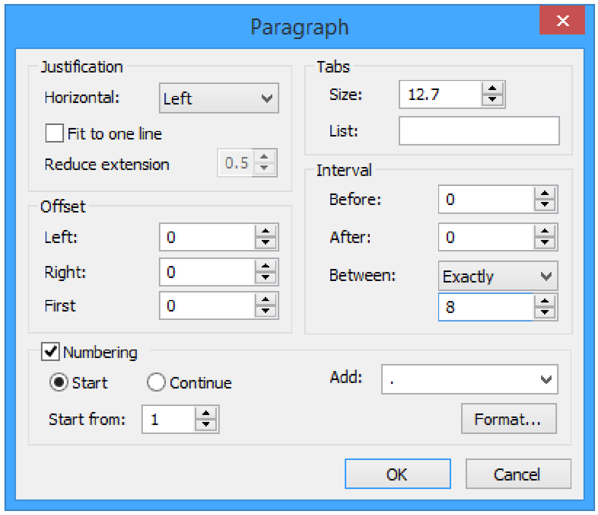
	<F9>	Format Font
	<F10>	Format Paragraph
	<Ctrl><F5>	Parameters of changing Paragraph size
	<F11>	Edit in separate window

Let’s format the created table. The columns width was set automatically when inputting the text based on the size of the input paragraph text. (When using the multiline text, the width is set by default). The width of each column must be changed according to the width of the columns in the BOM title block. The height of the template rows must be equal or (for multiline records) a multiple of the distance between the horizontal grid lines in the title block.

Select the whole table by clicking or with the following option on the main toolbar (button set “Text”):

		Select Table

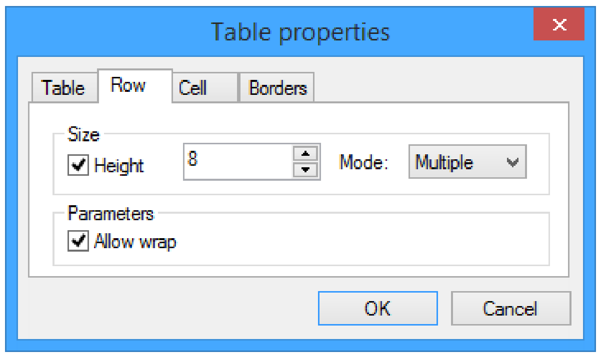
Then, using the option , call the paragraph parameters dialog box. The parameter “Between (interval)” should be assigned the value “Exactly” and the numerical value equal to the height of the rows in the title block table (in this example - 8). Press the **[Ok]** button to close the dialog box.



The height of the table rows should be changed accordingly, and the border display should be turned off. To do this, keep the table highlighted and call the option on the main toolbar (button set “Text”):

		Table Properties

On the “Row” tab, set the following row parameters: the row height, as well as the line spacing interval, is set equal to the distance between the horizontal grid lines in the title block. The parameter “Mode” should be assigned the value Multiple. This allows increasing the height of the rows with long entries in the table so that they exactly match the rows of the title block.



Additionally, one can set the parameter “Allow wrap”. This will allow prolonging the BOM table on the next page.

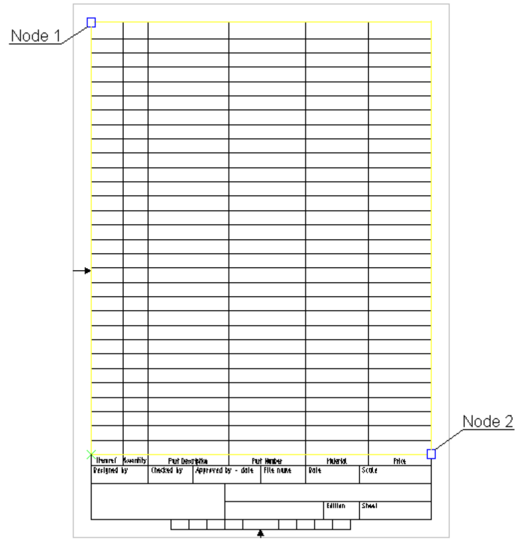
Since the BOM table grid is defined by the title bar, the display of the template table border lines should be turned off. To do this, on the tab **Borders** of the same dialog box clear the respective flags. Upon confirming the entered changes, the system returns to the BOM editing mode.

Keep calling the same dialog box for each separate cell in the template table, setting the desired width of each cell in the **Cell** tab, along with text offsets from the beginning and the end of the cell, if necessary. Besides, one can set horizontal justification mode for various table cells, and force no-line-break for the text, using the automenu option .

Table creation and editing capabilities are described in details in the chapter “Text”.

To complete BOM template editing, one can use the option:

	<F5>	Finish input

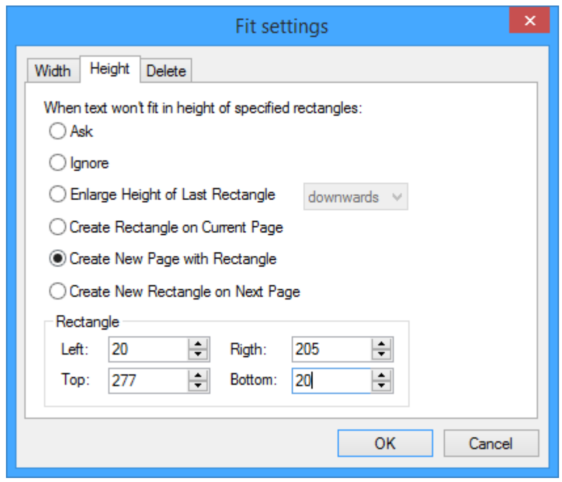


The system will then return to the BOM editing mode.

To complete working with this functionality, one can also use the option . After that, the system returns to the text creation command. To continue working with the system, with this command.

Next, let's insert the predefined title block fragment in the drawing. To do this, use the command FR: Create Fragment.

Then call the command “Edit” from the BOM context menu and snap the created text to the fragment nodes (“Node 1” and “Node 2” on the diagram) for the exact match between the BOM template and the title block lines.



The final step will be insuring the correct BOM continuation onto the subsequent pages. For this purpose, call the command **ST: Set Document Parameters** and on the **Fragments** tab specify the title block fragment names for the second and other subsequent pages of the BOM.

You will have to enter the template editing mode again and call the option . In the coming up dialog box on the tab “Height” set the action upon the text outgrowing the rectangle bounds as “Create new rectangle on next page”.

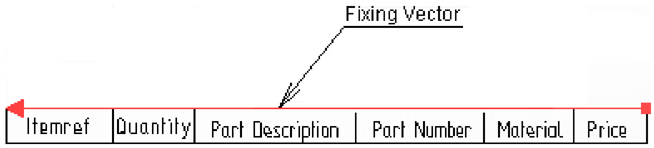
The new rectangle coordinates are set according to the coordinates of the outer nodes of the table (similar to the nodes on the previous diagram) on the title block drawing for the subsequent BOM pages.

> Note that the described order of steps in defining the BOM template parameters is not mandatory. All these steps can be performed in an arbitrary order.

Creating prototype for BOM located on existing drawing page

The sequence of steps for creating such a prototype is mostly the same as in the previously described situation. As in the case of creating a prototype for a BOM on the separate page, the prototype creation should begin with creating the BOM header in a separate file.

In the following example, let's create the table header consisting of the same four columns as in the previous case. Let's position the fixing vector along the top border of the header directed from right to left. The vector direction choice is based on the requirement, that the BOM table template located on the existing page has right justification.



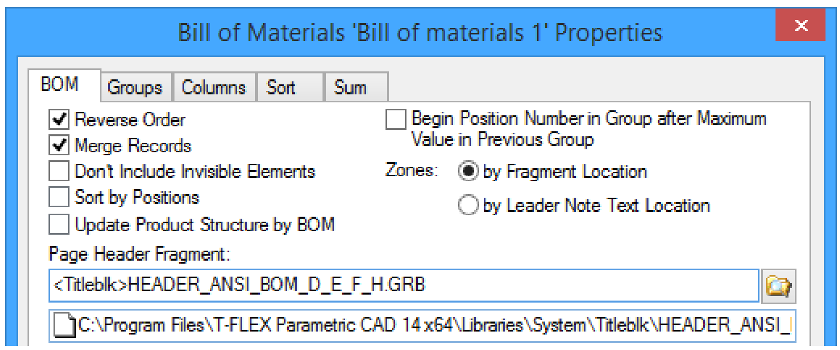
Save this file. Then call the command **BC: Create Bill of Materials**. In the coming up dialog box select "<Empty Template>" and press the graphic button **[Create New Template]**. After selecting the new document prototype and defining the name of the prototype being created, the system activates the text creation command.

In the text parameters dialog box, called by the option , set the Lower Limit vertical justification (a BOM located on the drawing page should grow from bottom up). In the case of the multiline text, additionally define the right horizontal justification.

After that, define the boundaries of the rectangular area for fitting the BOM table being created. Upon defining the text boundaries, the BOM property window will appear. In it, select the required BOM columns, sorting and summing rules, etc.

Additionally, specify the table header (footer) fragment on the tab **BOM** of the same window.

If the header is used in the table, specify the path to the header fragment. Otherwise, if the footer is used in the table, enter the path to the footer. The path can be defined using the button .



Upon confirming the specified BOM properties, the system will switch to the BOM table editing mode. The dialog box "Add Record into Section or Create Section" will again appear on the screen. On the main toolbar (if it is not locked), the button set "BOM" will appear. The BOM prototype should not contain any records; therefore, close the dialog box by pressing the button **[Cancel]**.

To format the columns of the created table, enter the template editing mode using the option . in the Paragraph parameters dialog box (the automenu option ) specify the line spacing for the table cells, as well as the text horizontal justification modes and the requirement of keeping the text as one line.

In the **Table properties** dialog box, called by the option  on the main toolbar (set "Text"), change the width of each column according to the width of the columns in the header fragment. The height of the table rows does not have to be set in this case. Also, turn on the table border display (is on by default). On the **Table** tab, the right horizontal justification should be set for the table. This will allow automatically keep the BOM being created toward the right hand side of the drawing above the title block text.

> Note that the justification defined in the table parameters must correspond with the fixing vector direction of the BOM header.

The template editing, as well as the whole BOM editing, can be completed using the option . Exit the text creation command and save the resulting prototype.

Printing Documents

PRINTING DOCUMENTS

T-FLEX CAD documents are printed out using the command **File > Print…** (**PT: Print Model**), which is similar to printing commands of most Windows applications. This command allows outputting a single (current) T-FLEX CAD document to a printer or other hardcopy-making device.

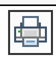
However, one may often need to print a whole batch of drawings, arranging those economically on one or several sheets of paper of the specified format. In this case, one can use an additional command “Print documents”. This is an external T-FLEX CAD application, which allows composing the required batch of drawings and optimize their placement for printing on sheets of the specified size. The composed batch is created in the current T-FLEX CAD document. The drawings included in the set will be represented in the resulting joint document as internal 2D pictures with the preserved connection with the source files. The resulting document is then printed out just like an ordinary T-FLEX CAD document.

Follow below are the descriptions of both methods for printing T-FLEX CAD documents.

PRINTING A SINGLE DOCUMENT

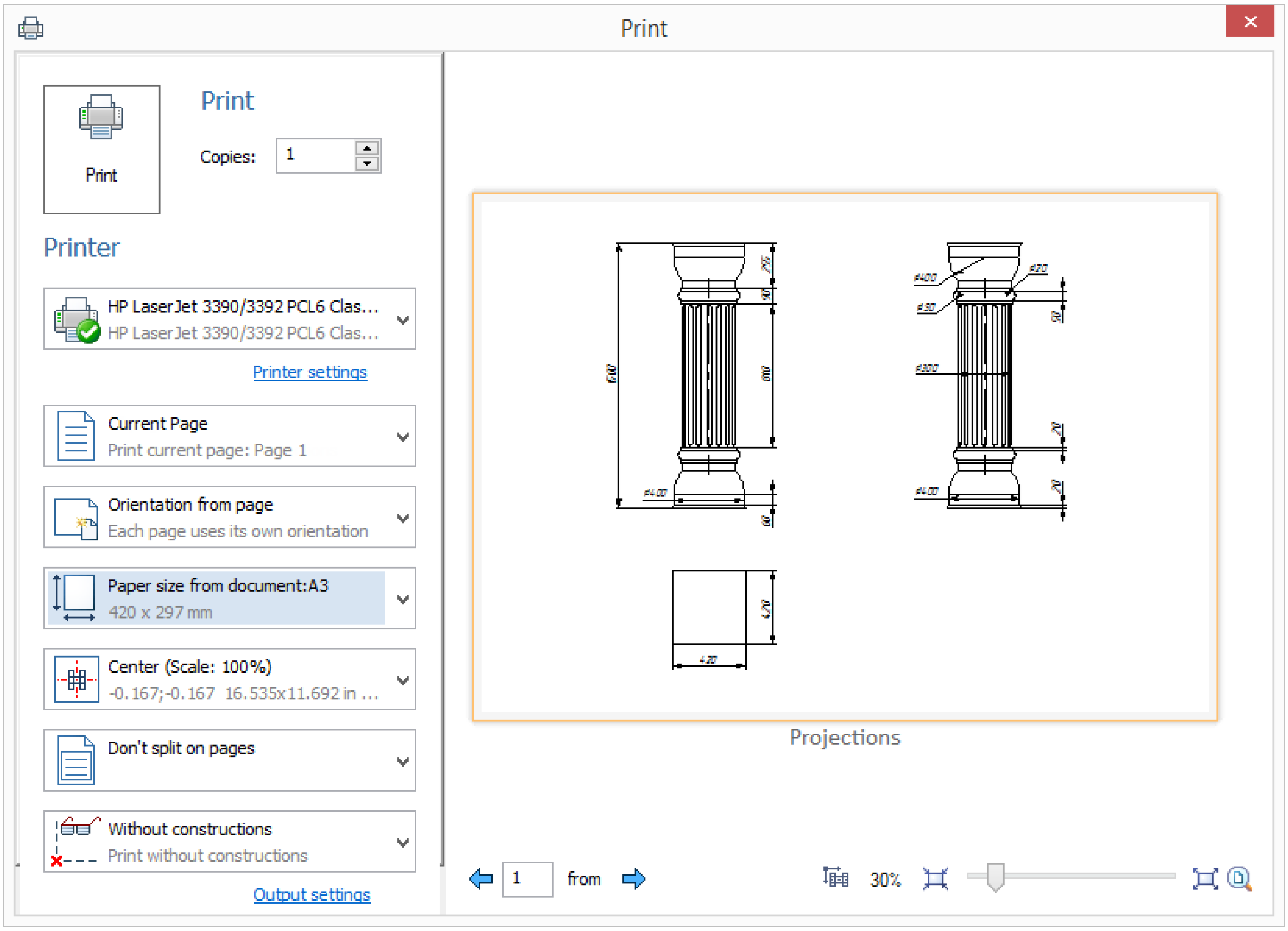
2D drawings can be printed as well as 3D models. It is important, which window was active at the time of calling the print command. Printed are all elements that are visible on the screen, except those on the layer assigned the «Screen only» attribute. Please also keep in mind that construction elements are not printed by default.

To print out a drawing, call the command **PT: Print Model**:

Icon	Ribbon
	
Keyboard	Textual Menu
<PT>, <Ctrl><P>	File > Print

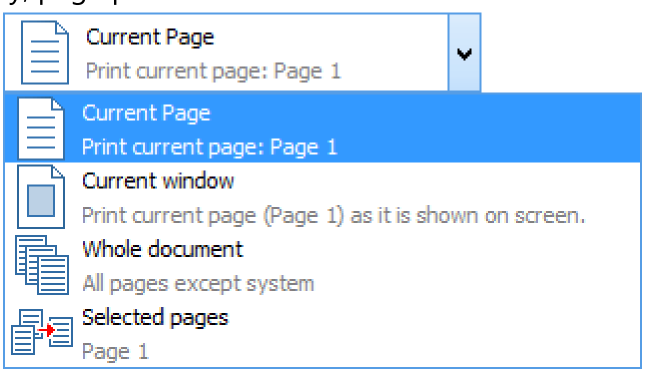
Upon calling the command, the Print dialog box appears.

Copies. Sets the number of document copies to print.



Printer. Specifies the printer from the list. The list displays all installed printers.

The button Printer settings calls the Windows dialog for setting up properties of the selected printer; it allows specifying print quality, page parameters etc.

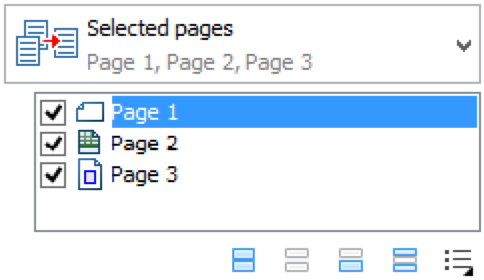


Current Page. Upon setting this parameter, the printout will contain the image displayed in the current window, including portions of the image outside the format frame.

Current window. Upon setting this parameter, the printout will contain only the contents that fit inside the format frame.

Whole document. All document pages of "Normal", "Text" and "Bill of materials" type will be printed. If a 3D view was active upon calling print command, it will be also printed.

Selected pages. When printing a multi-page drawing, you can specify which pages to print. A list with available pages appears. You need to set the flag near required pages or use one of the options.



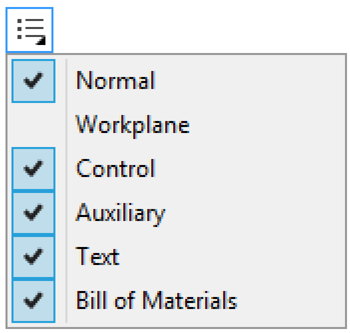
Select all. All pages available in the list will be selected.

Deselect all. Cancels pages selection.

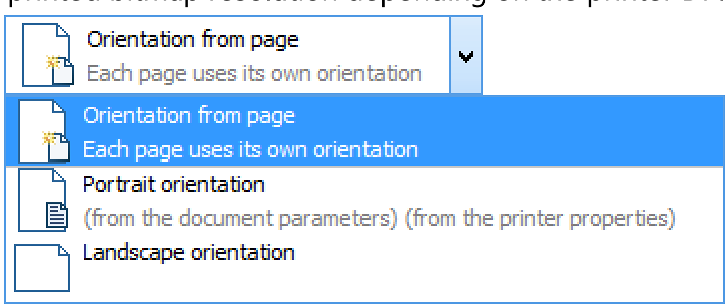
Invert selections. Invert pages selection.

Only print odd pages. Only odd pages will be selected from the list.

Filter. Allows to select pages types that will be displayed in the list.



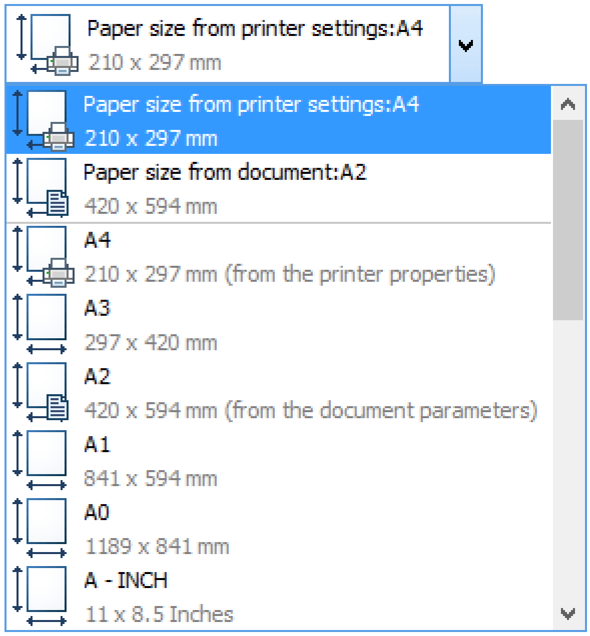
You can also select 3D view for printing, if it is available in the current window. Upon 3D view printing, dynamic selection of the printed bitmap resolution depending on the printer DPI and scale is performed.



Orientation from page allows to specify each page orientation individually.

Portrait orientation. Sets portrait orientation for all printed pages.

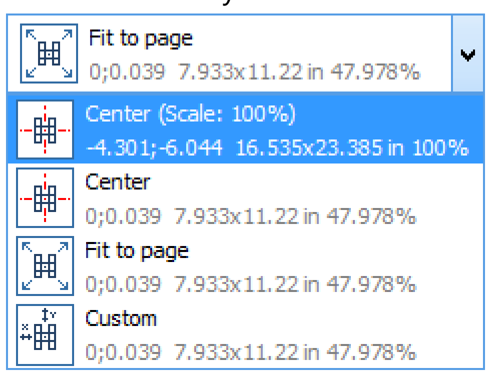
Landscape orientation. Sets landscape orientation for all printed pages.



From document. Paper size is controlled by settings of the current document.

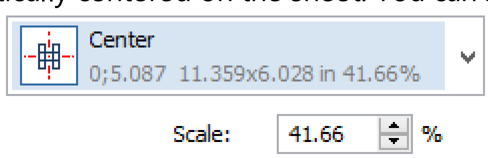
From printer's settings. Size of the paper that will be printed out will be determined by settings of the printer. This mode is enabled by default.

You can also select paper size from the list manually.



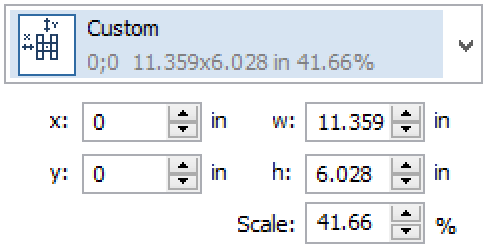
Center (Scale: 100%). Makes the image automatically centered on the sheet at 100%.

Center. Makes the image automatically centered on the sheet. You can set its scale manually.



Fit to page. Setting this parameter scales the drawing so as to fully fit on one page.

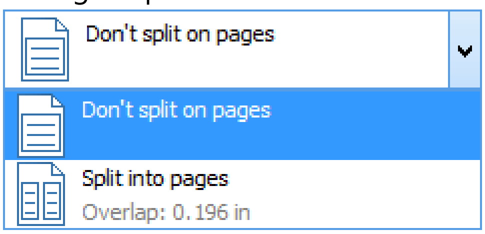
Custom. Allows you to specify image position on the drawing and its scale manually.

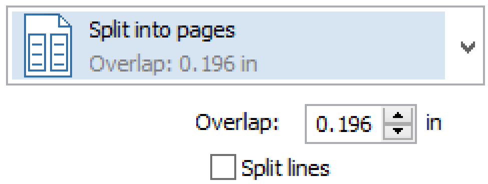


Left and Top. These parameters define the left and top margins of the paper sheet, respectively.
Width and Height. Set the width and height of the printed image.

Scale. Sets the image scale.

If one of the three latter parameters is manually modified, the other two are adjusted automatically to maintain the drawing's aspect ratio.



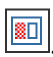


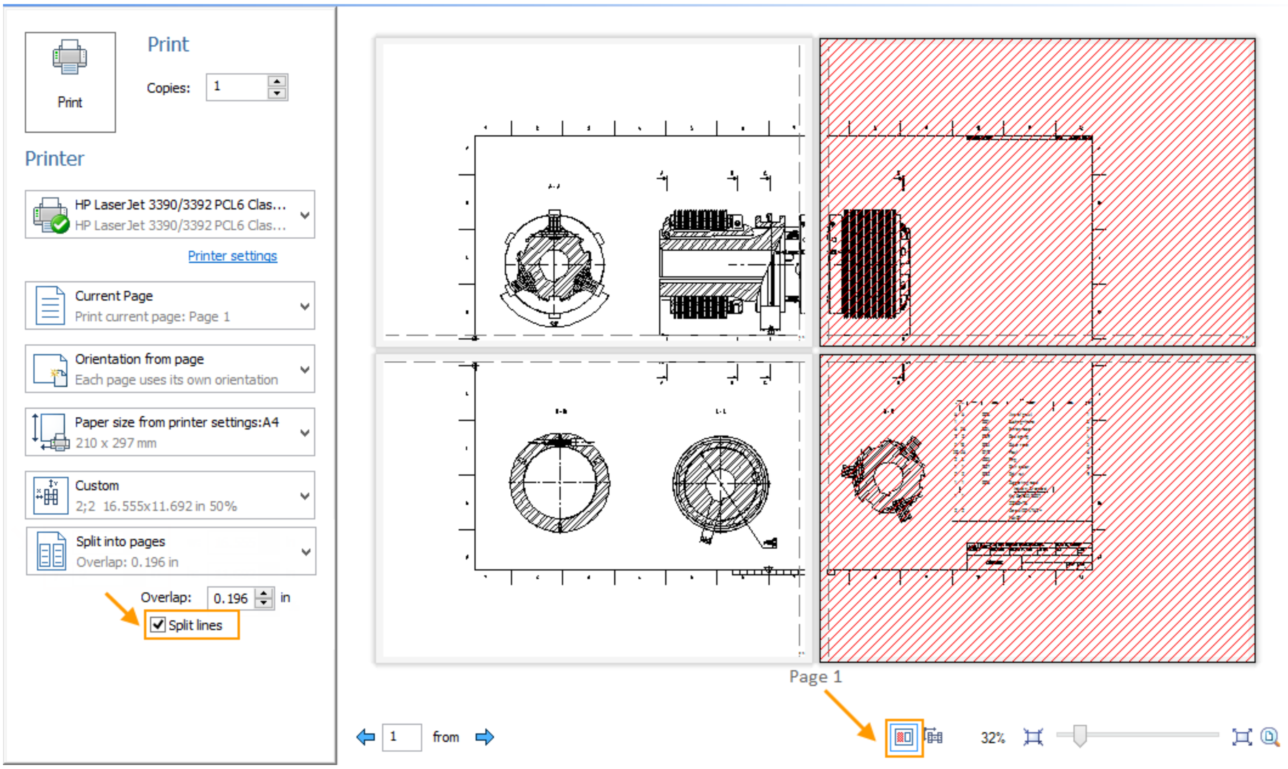
Don't split into pages. A drawing will be printed on one page.

Split into pages. Use this option when a drawing needs to be printed at a strictly defined scale while the full image does not fit on one page. The option will let you automatically distribute the drawing over multiple pages. The result can be preliminarily evaluated in the preview pane.

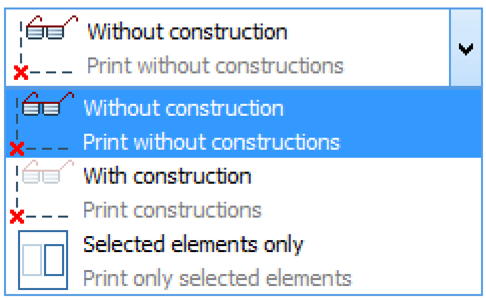
> Upon selecting the option **Custom** parameter for image location is activated.

Activate **Split lines** option to print split lines. The option appears only in **Split into pages** mode.

You can select skipped pages using option .



Overlap. This option sets the amount of overlapping between the neighboring sheets when using the Split into pages option. The overlap can be used for gluing separate printed sheets together.



Without constructions. Construction elements will not be printed.

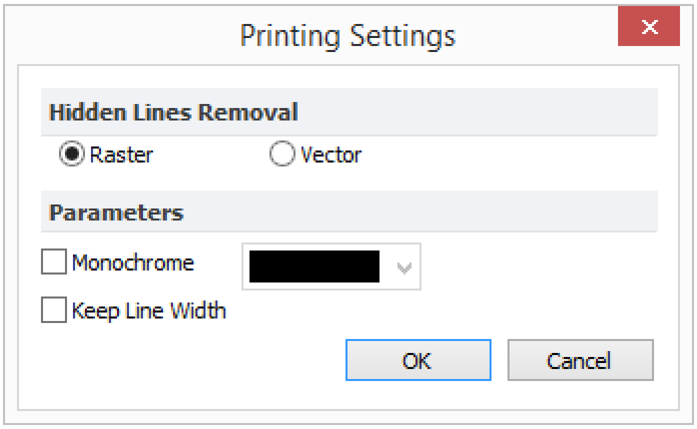
Print Constructions. Setting this parameter makes construction elements printed as well.

Selected elements only. Enabling this flag allows us to print out only selected elements of the drawing.

The parameters managed in Print dialog are retained and displayed upon subsequent opening of the print dialog.

Printing Settings dialog.

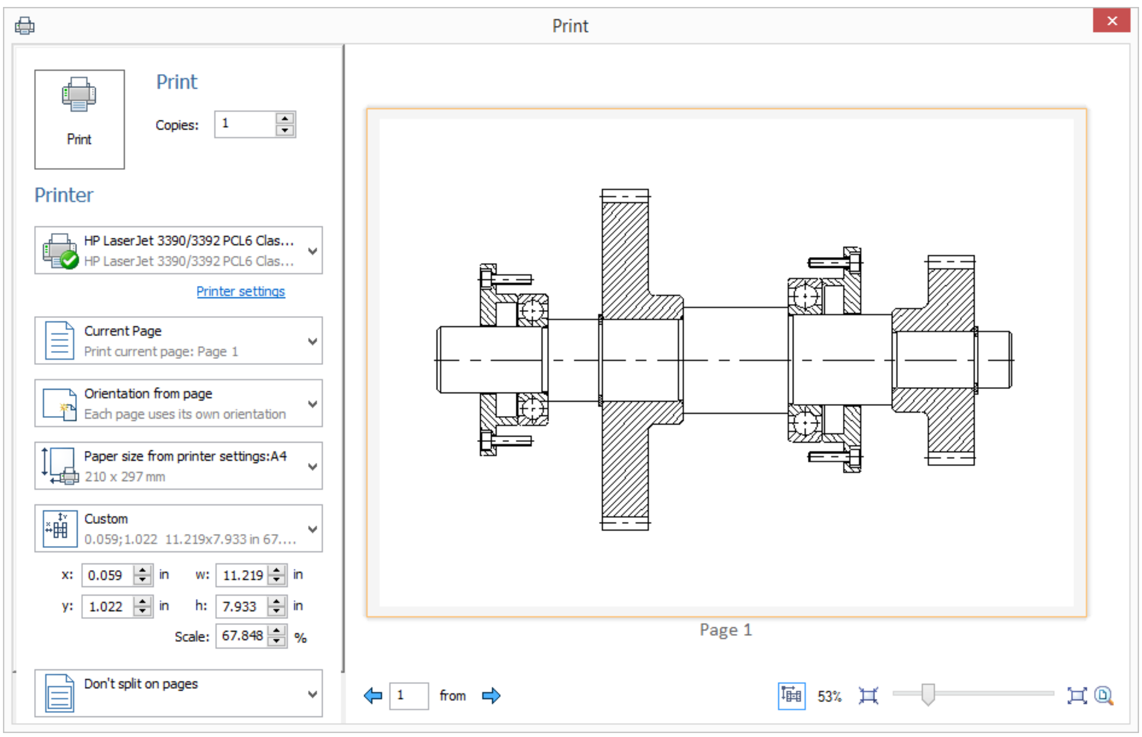
Hidden Line Removal. You can specify the method of removing hidden lines from the drawing: Raster or Vector removal. This option is important when outputting a drawing to pen plotters that require vector hidden lines removal.



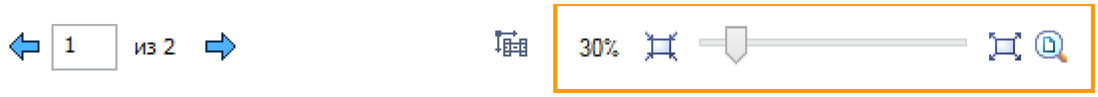
Monochrome. This option makes the drawing printed all in one color.

Keep Line Width. Setting this parameter helps avoid variations in line thickness when printing a document at a scale other than 100%.

Preview window is located in the right pane of the dialog.



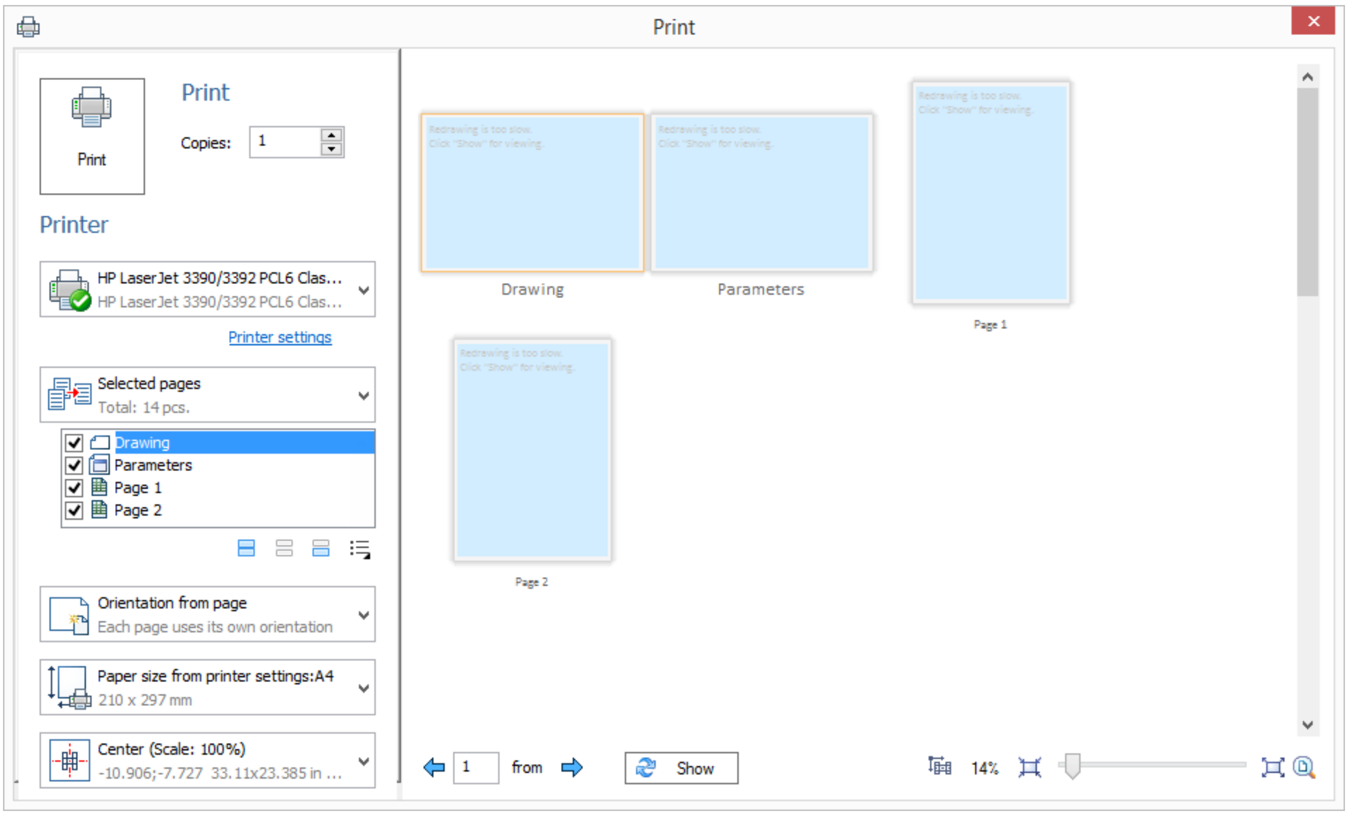
Its scale can be specified in the right bottom corner.



To zoom in and zoom out use options and . Option allows to fit page.

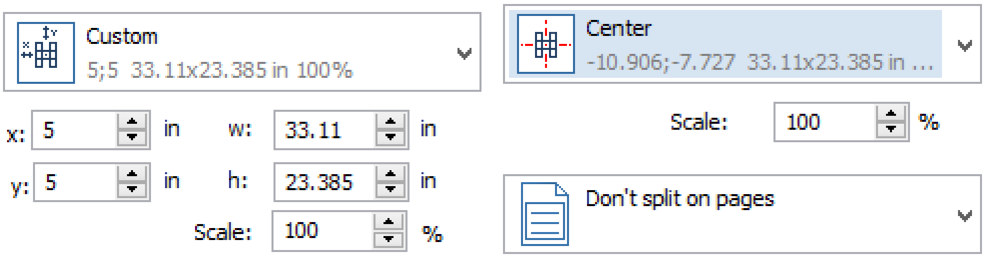
Options and allows to select pages in preview window.

Several pages can be shown in preview window when you zoom out. If their display takes a long time, their content is not displayed and Show button appears in the bottom of the dialog.



Selected page is highlighted with orange frame.

Option allows to select position and scale of the printed area manually. You can move printed area with pressed left mouse button. Mouse wheel rotation allows to change scale of the printed area when the option is active. The drop-down list switches to the corresponding location parameter according to the changes.

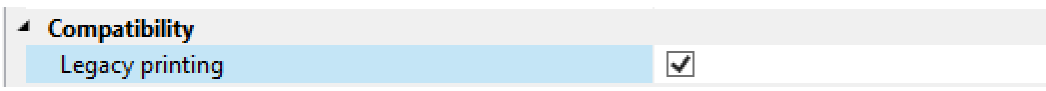


Printers' fields are showed with grey in preview window. There are no fields for virtual printer.

After all parameters setting press Print button in the top left corner.

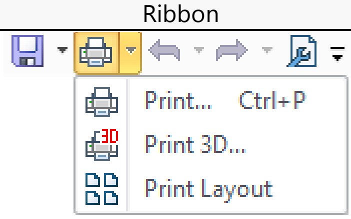
Dialog from the previous versions of T-FLEX CAD is also available.

If you want to use legacy version of printing dialog box, you need to set option **Legacy printing** on **Additional Options** tab in **Options** command.



Printing Several Documents

The "Document printing utility" application can be called by the following T-FLEX CAD command:

Icon	Ribbon
	
Keyboard	Textual Menu
	File > Print Documents

To work with the application, make sure that it is running. By default, it is always started. If, however, the application is shut down for some reason (this can be instantly determined by the absence of the command in the textual menu), it can be loaded by the command **AP: External Applications**. When the application is running, the list of T-FLEX CAD toolbars contains an additional "Print Control" bar.

The batch of documents will be composed on the basis of the document opened in the active window of the T-FLEX CAD upon invoking the print module. The pages of the base document correspond to the print paper sheets. The images of the documents which need to be printed out are placed on these paper sheets. When carrying out the batch layout for printing, new pages can be automatically added to the base document, and also the settings of already existing pages can be modified. Thus, before calling the print module, it is recommended to create a new blank drafting which will become the basis for compiling the batch of documents.

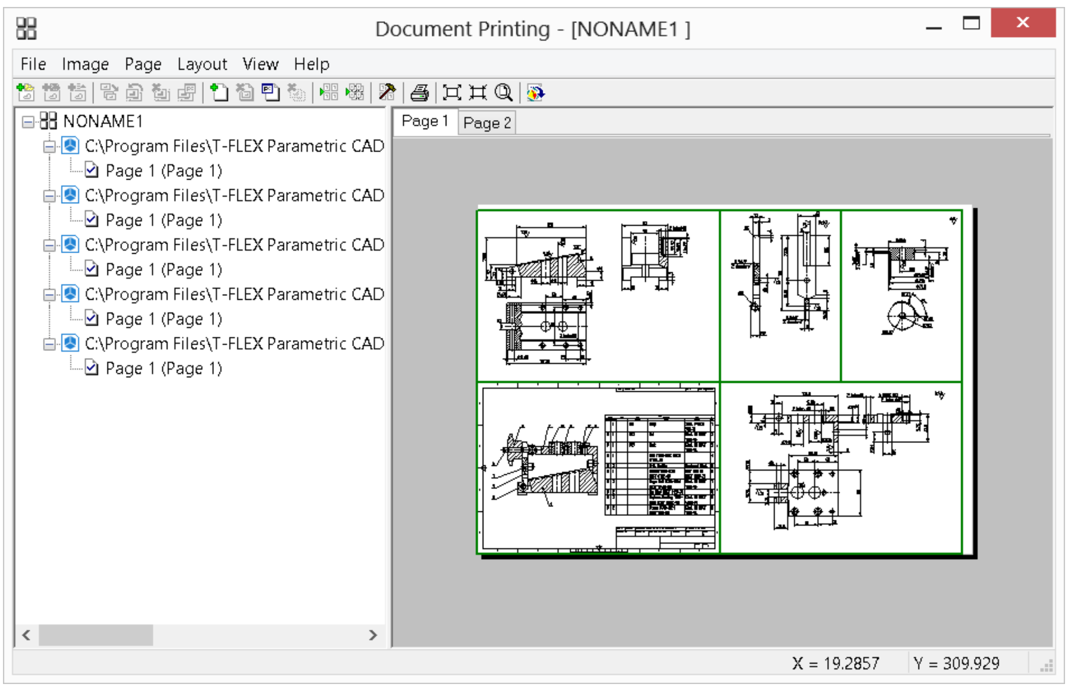
In the print module, only the pages of the base document having the type "Normal" are displayed and used.

The documents selected as components of the batch are put into the base document as internal pictures with the preserved connection with the source files.

Formation of the batch of documents is carried out in a separate window "Print documents". This window is split vertically into two panes: the structure of the composed batch of documents is shown on the left, the window with the preview of the composed document is shown on the right.

Formation of the batch for printing is usually performed in the following way: with the help of the commands of the textual menu and toolbar the documents which have to be included into the batch are indicated. A user can select the T-FLEX CAD documents and the documents of all graphics formats available for insertion in the command "IP: Create picture". For the T-FLEX CAD documents added to the batch and multi-page pictures, it is possible to specify the pages which will be printed.

The images of all documents included into the batch for printing, by default are placed on the current page of the base document. Afterwards, during the batch layout, the images are distributed in a rational way over pages of the base document.



The batch layout (creating new pages, modifying parameters of already existing pages, placing the images over different pages) can be carried out both manually or automatically with the help of the special command.

The compiled batch of documents can be sent to a printer straightway or can be saved as the T-FLEX CAD file.

Forming the Structure of Documents Batch

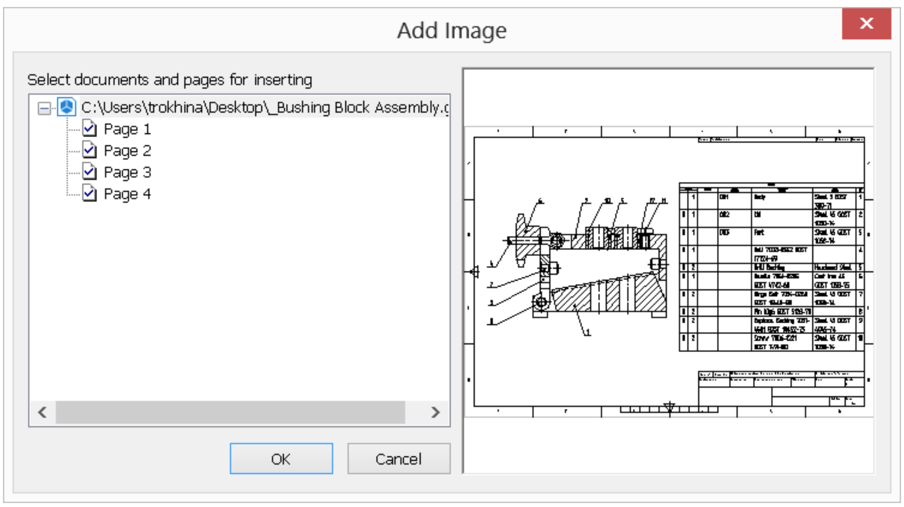
For adding the document to a list, the following commands of the print module are used:

Keyboard	Textual menu	Icon
	File > Add Image from File…	
	File > Add Images from DOCs	
	File > Add Image from Open Document…	

The command "**File**|Add Image from File…" allows a user to add arbitrary T-FLEX CAD documents to the batch, and also raster and vector images. When this command is invoked, the file selection window appears.

When among the selected documents there is at least one multi-page document, the dialog window "Add Image" is opened additionally. The list of the selected documents together with the specified pages of the given documents as well as the area for the preview of the pages contents are displayed in this window.

By default, in the dialog all pages of the selected documents are marked with the tick (next to the page name). If everything is left unchanged, after pressing **[OK]**, the images of all pages of the selected documents will appear in the preview window of the print module. If for some page the tick is taken off, the image of that page is not put into the compiled batch of documents. To take off the tick, it is enough to point at the desired tick with the cursor and press .



Now the added documents will appear in the batch structure (in the left pane of the print module window). Note that if dimensions of the added image are larger than the dimensions of the current page of the base document, the system can automatically change the scale of the image so that it could fit on the page (the system actions will depend on the settings of the print module).

The command File|Add Images from DOCs allows to add images from DOCs to the batch. The command is available only when an integration with DOCs is configured. For this purpose you need to run Customize >> Options >> T-FLEX DOCs and select any installed DOCs system from the drop-down list.

The command "**File|**Add Image from Open Document…" allows a user to add to the composed batch the T-FLEX CAD documents opened at the present moment. After invoking this command, the window "Add Image" (analogous to the described above) with the list of the currently open documents and their pages will appear.

Before closing the dialog "Add Image" the system checks if any changes were made in the open T-FLEX CAD documents (since the last save).

If the unsaved modifications were found in any of these documents, the system displays the message with the prompt to save the given document. If the user denies saving, the given document can not be added (it will not be shown in the list of the documents in the window "Add Image").

After compiling the list of the documents of the batch (with the help of any command described above), additional customization of the batch can be carried out by taking off/setting on flags for any image in the batch structure. The images for which the ticks are taken off remain in the batch structure but will no longer be displayed in the preview window of the print module (and as a result will not be printed out). The tick controlling page visibility in the batch can be taken off/set on with the help of or by using commands «**Image|Show**» and «**Image|Hide**»:

Keyboard	Textual menu	Icon
	«**Image|**Hide»	
	«**Image|**Show»	

When forming the structure of the documents batch, the following commands of the print module can be also used:

Keyboard	Textual menu	Icon
	Image > Dublicate…	

This command allows a user to "duplicate" selected images (from the list of the documents already added to the batch being formed). The images can be selected in the batch structure (in the left pane of the print module window) or directly in the preview window of the print module. The dialog in which the number of the copies being created will have to be specified appears on the screen. Thus, several copies of any image can be printed out.

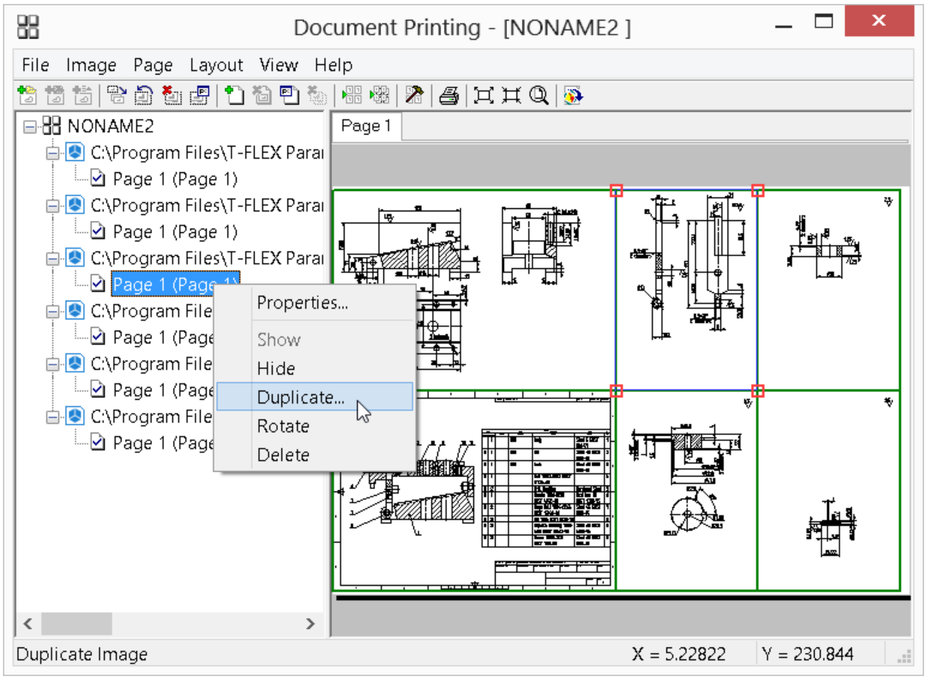
Keyboard	Textual menu	Icon
	Image > Rotate	

After invoking this command, the images of the documents selected in the list or in the preview window turn by 90° counterclockwise.

Keyboard	Textual menu	Icon
	Image > Delete	

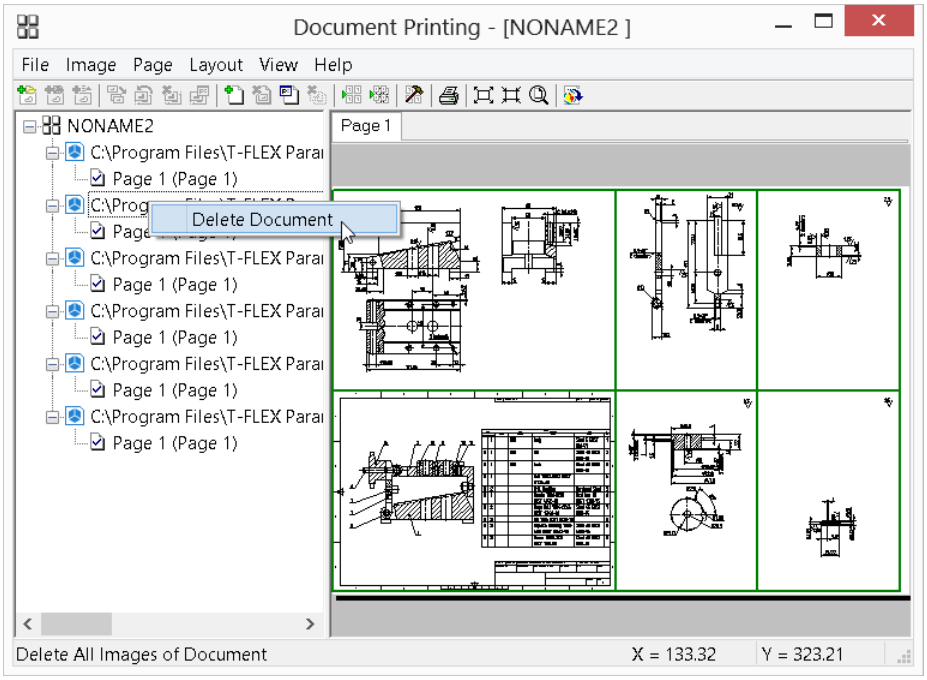
This command removes selected images from the list of images of the composed batch.

All commands for dealing with the images of the batch can be also invoked from the context menu of the image.



Keyboard	Textual menu	Icon
	File > Delete Document	

This command allows a user to remove the selected document entirely from the structure of the composed batch. This command can be also called from the context menu for the document.



Keyboard	Textual menu	Icon
	File > Update Images	

This command allows a user to synchronize the contents of the base document in the main T-FLEX CAD window and in the print module.

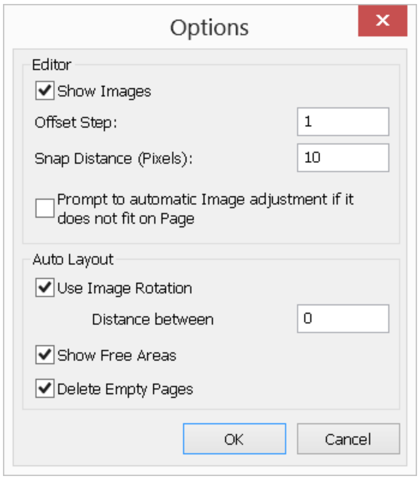
The document print module works in parallel with the main T-FLEX CAD window. This allows a user to make changes in the base document concurrently in the print module and in the main T-FLEX CAD window. For example, simultaneously with the print module functioning, via the T-FLEX CAD window of the base document a user can modify the page settings, picture parameters or modify the documents on the basis of which the images have been created in the print module. As a result, the situation may arise when the print module lags behind in tracking down these changes. The command **Update Images of Document** is intended to resolve these situations. It removes all data on created images and used documents from the print module, and then restores this data according to the contents of the base document from the T-FLEX CAD window.

Print Module Options

Options of the print module are specified in the command **Set Editor Options**:

Keyboard	Textual menu	Icon
	View > Options...	

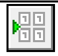
This command invokes the dialog “Options”:



- The parameters group “Editor” defines general settings for dealing with the print module:
  - Show Images. This parameter controls the display of the images in the preview area of the print module. When the flag is turned off, only the image bound box is displayed in the preview area. By default, this flag is turned on.
  - Offset Step. This parameter defines the offset step for moving the images selected in the preview area with the help of the keyboard keys (<↑>, <↓>, <←>, <→>). The step size is specified in page units.
  - Snap Distance (Pixels):. This parameter defines the maximum distance in pixels at which the sides of the selected images will be snapped to the sides of other images on page (horizontally and/or vertically). This parameter is taken into account when moving the image with the help of the mouse (recall that the images can be also moved with the help of the keyboard keys (<↑>, <↓>, <←>, <→>).
  - Prompt to automatic Image adjustment if it does not fit on Page. This parameter defines the system response in cases when the image does not fit on the page of the base document (for example, when adding a large image). When this parameter is turned off, the system automatically moves and scales the images in such a way that they will fit on the page. When the parameter is on, the prompt about the necessity of image translation/scaling will be shown.
- The parameters group “Auto Layout” defines layout parameters:
  - Use Image Rotation. Allows a user to apply rotation to the image by 90 degrees when selecting ways of the image placement.
  - Distance between Images. This parameter sets the minimum allowable distance between the images after the layout. The distance magnitude is specified in the units of the corresponding page of the source document.
  - Show Free Areas. If this parameter is turned on, the areas of the base document pages not occupied with the images are marked with green color after performing the layout.
  - Delete Empty Pages. This parameter affects the work of the mode of layout of all pages in the print module. When this flag is enabled, pages, which after performing the automatic layout, ended up being empty (i.e., not containing any images) are automatically removed. When this flag is turned off, such pages remain in the structure of the print batch (they can be deleted manually afterwards).

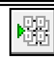
Auto Layout of the Documents Batch

For automatic layout of the images within the scope of one page, the following command is used:

Keyboard	Textual menu	Icon
	Layout > Layout Current Page	

This command distributes images placed on the current page within the scope of this page in such a way that they will not overlap with each other and occupy the least area. In case when the system is not able to place all images on one page, the message **Layout errors** appears with the list of "problem" images.

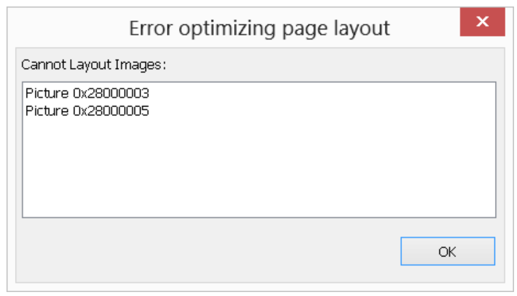
For automatic layout of all images of the documents batch, the following command is used:

Keyboard	Textual menu	Icon
	Layout > Layout All Pages	

This command distributes the images of the batch over different pages of the base document. When carrying out the layout, the required number of the base document pages is automatically added. The size of the added pages will be the same as that of the last existing page of the base document. If after successful completion of the layout, the blank pages will still remain in the base document, they can be removed automatically (this operation is performed by setting on the flag "Delete empty pages" in the dialog of the print module options).

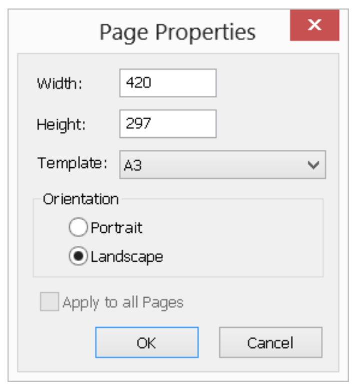
The pages are not considered empty if they contain hidden images (images which were added to the batch for printing but for which the flags were taken off later in the batch structure). Such pages can be deleted only after removing hidden images.

If the size of some image exceeds the size of both existing and automatically created pages of the base document, the dialog window "Error optimizing page Layout" will appear on the screen with the list of problem images. The flag "Change Image Size and Run Optimization Again" found at the bottom of the given dialog allows a user to choose further actions of the system.



If this flag is disabled, after pressing **[OK]** the layout will be completed. The images which could not be placed automatically will have to be placed manually.

When this flag is on, after pressing**[OK]**, the dialog for page format selection, similar to that invoked by the command **Page > Page Size...**, will appear. By default, the smallest by area format which accommodates any visible images counting gaps is set in this dialog (see chapter "Print Module Options"). After selecting the format and pressing **[OK]**, the dialog "Page parameters" is closed, parameters of all pages of the base document are automatically modified and the repeated layout is carried out.



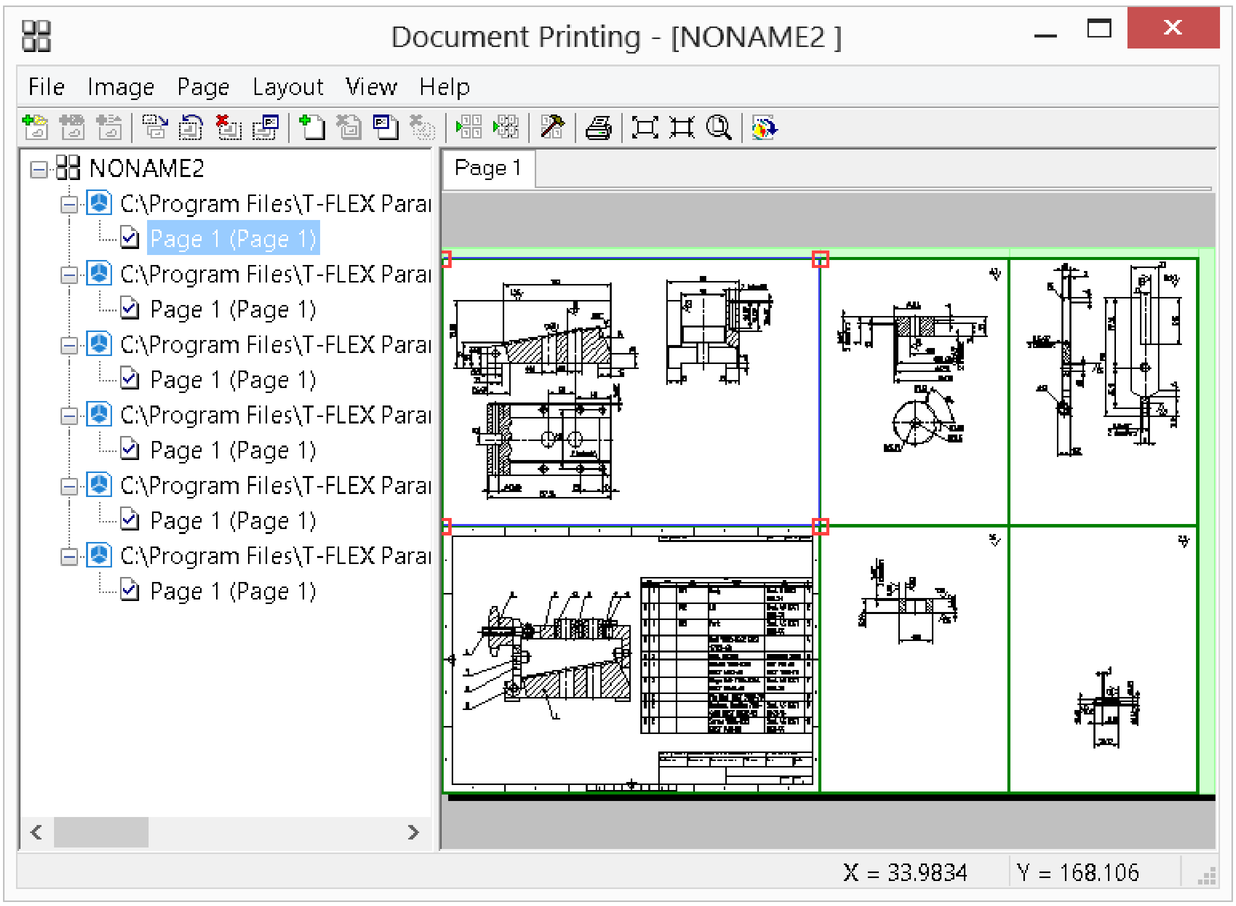
“Manual” Layout of Documents Batch

Print module allows a user to enter “manual” corrections into results of automatic layout: change location and size (scale) of images, their distribution over the pages of the batch, etc.

To make the work with the created batch more convenient, the frames of all images in the preview area are marked with a certain color. Nonactive images (i.e., not selected for modification of position and size) are marked with a frame of dark-green (if they fit on the current page) or dark-red (if they do not fit on the current page) color. If the placement of the nonactive image could not be adjusted upon automatic layout, the color of its frame will be dark-red.

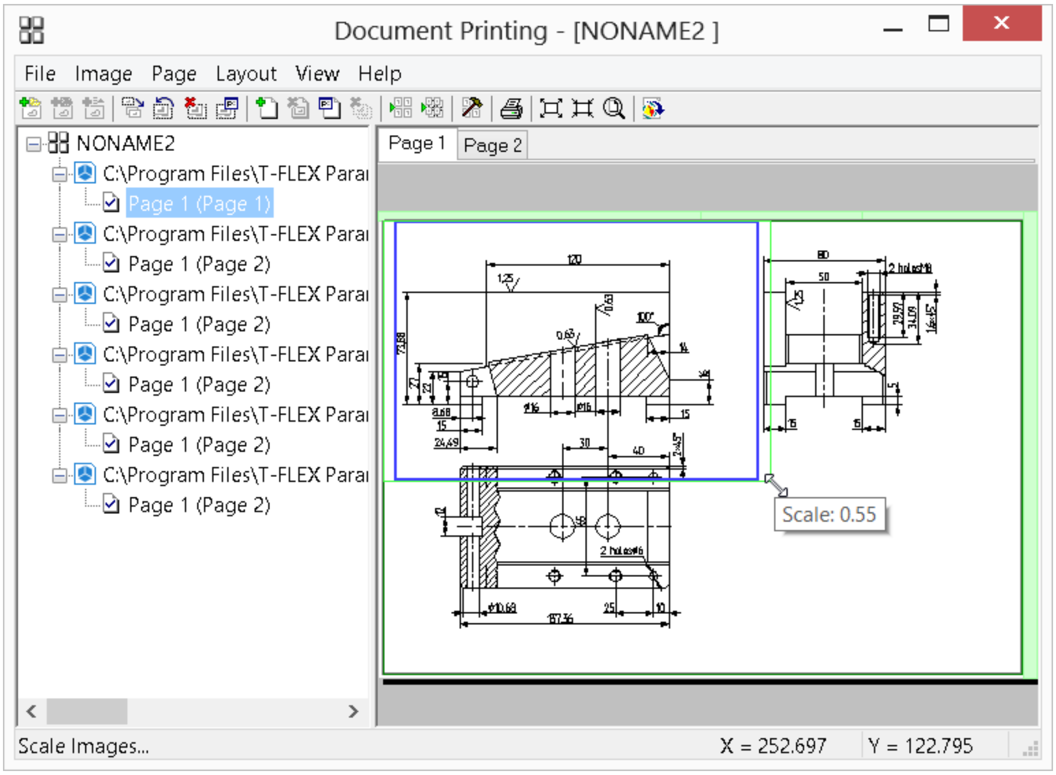
Selected (active) image is marked by default with sky-blue color (the color is specified in the command **SO: Set System Options**, tab **Colors**, parameter **2D Elements Highlighting/Additional Color**). In addition, at the corners of the frame of the selected image, square-markers are displayed.

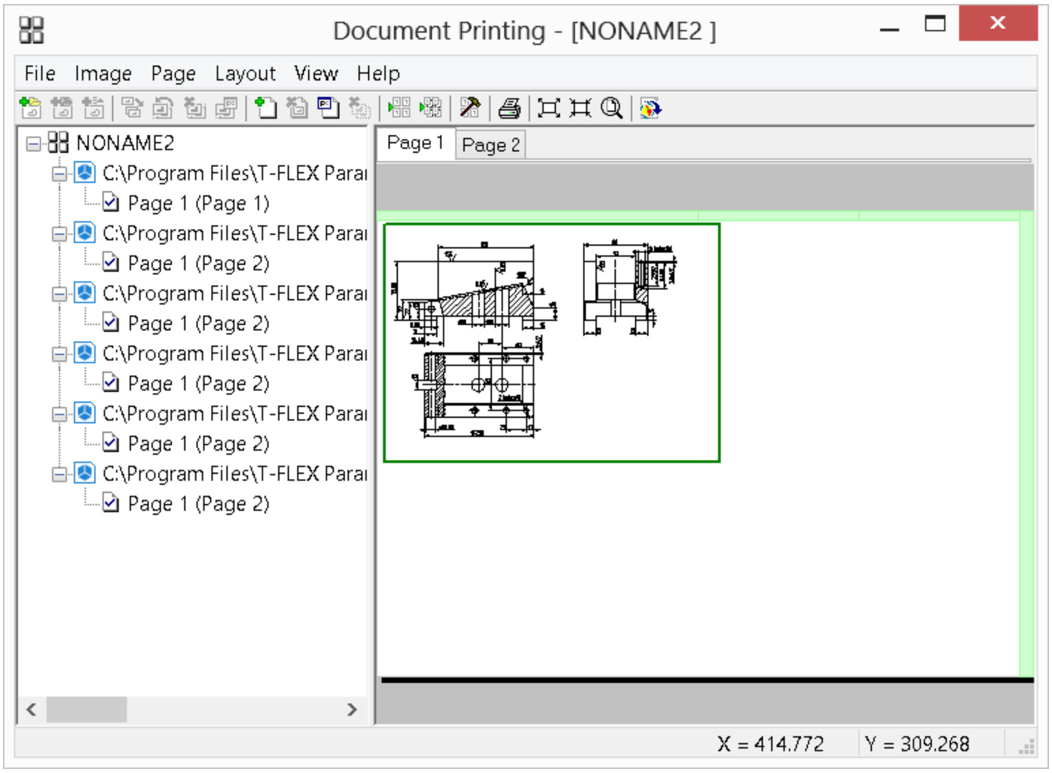
A user can select the image in the list or directly in the preview window with the help of .



When bringing the cursor to the borders of the frame of the selected image or to the markers at the corners, the cursor takes the form of a two-directional arrow (↕, ↔ or ⤡). If at this moment a user presses and without releasing the pressed mouse button, moves the cursor, the image boundaries will move along with the cursor. After releasing the mouse button, the size of the image bound box will be modified (thus, the scale of the image view will be modified). The same result can be achieved if after the first click on , a user releases the mouse button straightway, moves the cursor to the desired place (corresponding to the desired location of the image bound box) and presses the again.

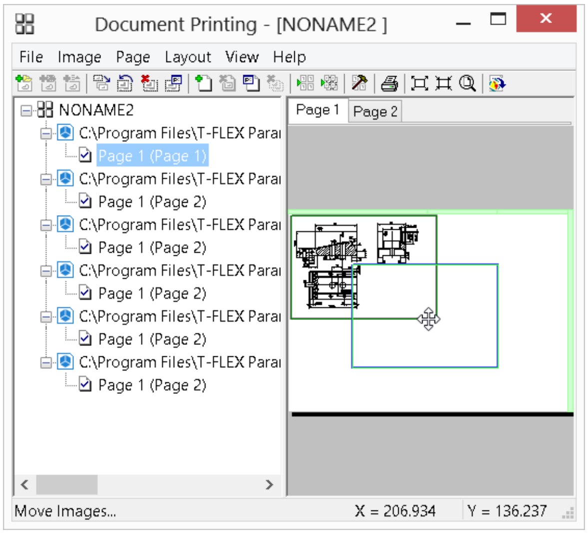
When moving the cursor, the scale magnitude is rounded off up to the second digit after the comma. A user can refuse to round off the scale by keeping the key <Ctrl> pressed while moving the borders of the image bound box.

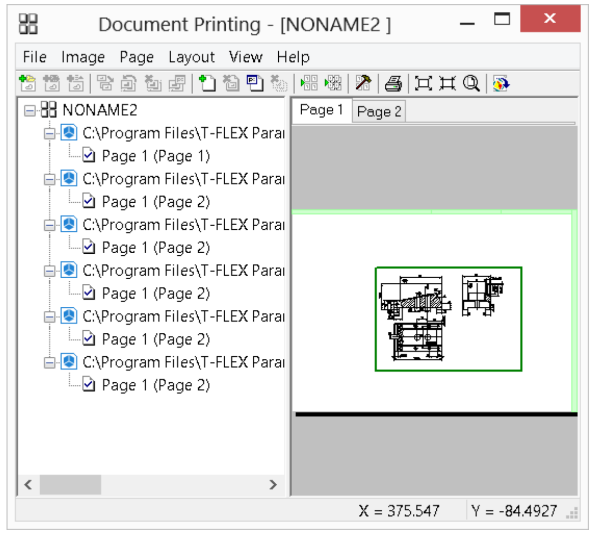




In order to move the image over the page, after the image selection put the cursor in any place inside the image bound box. The cursor will take the following form: . Press and without releasing the mouse button, move the cursor to the required place on the page. Also, another way of working with the dragger can be used here – if after pressing , the mouse button is released, the image bound box will start to follow the cursor motion. Second click on will fix the image in a new place.

When moving the image with the help of the mouse, the snapping of the selected image to the sides of other images on the current page of the document is performed (taking into account the value of the parameter “Editor/Snap Distance (Pixels)” in the options dialog of the document print module).





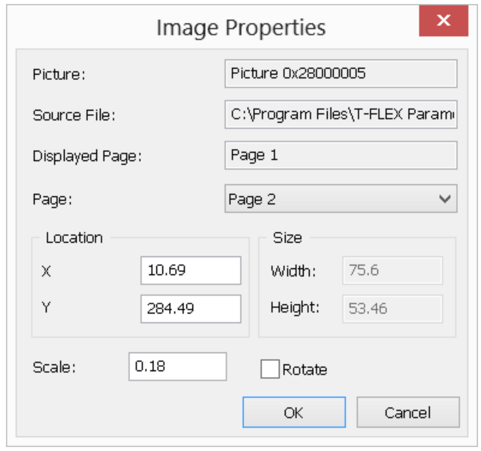
It is also possible to move the image with the help of the keyboard keys <↑>, <↓>, <←>, <→>.

Selected image can be also moved to another page of the base document. To do it, point with the mouse inside the image bound box, press and without releasing the pressed mouse bring the cursor to the tab of the desired page. The selected page will be automatically opened in the preview area. After that, it will remain only to place the image on that page and release the mouse button. If after the image selection, a user presses and immediately releases the mouse button, the translation will be completed after the second click of the (already on the selected page of the batch).

The scale, the placement page and the location of the selected image within that page can be also modified with the help of the command **Edit Image Properties…**

Keyboard	Textual menu	Icon
	Image > Properties…	

Upon invoking this command, the dialog window “Image properties” with the parameters of the selected image is opened:



Picture. This field carries picture ID in the base document of the batch.

Source File. Path of the source file with the given image

Displayed Page. This parameter indicates the page in the source document used for creating the given image.

Page. This parameters shows the page of the base document on which the current image will be placed. The drop down list allows a user to select another page of the base document for placing the image.

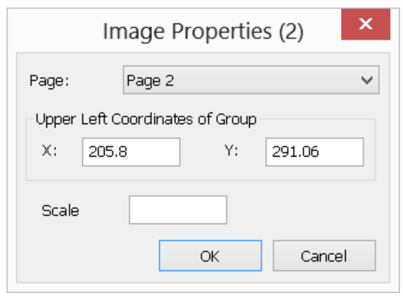
Location. Parameters of this group define the image location within the page of the base document.

Size. This is the parameters group showing the real dimensions of the given image (taking into account the specified scale).

Scale. Scale of the view of the current image on page of the base document.

Rotate. When this flag is enabled, the image is turned by 90° (with respect to the initial position). This parameter is enabled automatically if the image was rotated upon the auto layout.

For modifying scale and position of several images on one page at once, multiple selection can be used. For example, you can use selection with window (in the preview area) or successive selection with the help of <Shift>+ (<Ctrl>+ cancels your selection of the image). Note that multiple selection can be used only for images placed on one page of the base document.



Selected images are marked with common frame. Modification of size (scale) and location of a group of images is carried out the same as when dealing with one selected image.

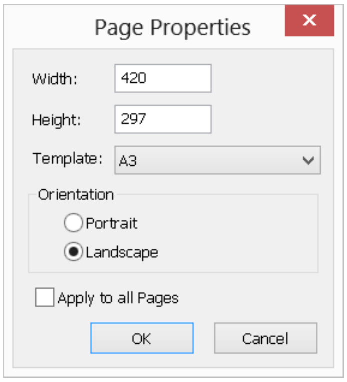
In this dialog, the scale field will be left empty if selected images have different scales. If the scale magnitude is specified, then after pressing **[OK]** and closing the dialog, the specified value will be set for all selected images. If the field is left empty, the scale of the elements will not be changed.

Invoking the command **Edit Image Properties…** when selecting several images at once makes the dialog window “Image properties” appear. In the title of this window it is shown to how many images the settings of this dialog will be applied.

Change of Parameters for Pages of Base Document. Adding/Deleting Pages

To modify parameters of any page of the base document, the command **Change Page Properties found in the print module is used**:

Keyboard	Textual menu	Icon
	Page > Page Size...	



Upon invoking this command, the dialog "Page Properties" is opened. With the help of this dialog, you can modify the format (by selecting it from the list of the standard formats or by specifying desired width and height) or orientation of the current page of the base document. The flag "Apply to all pages" allows a user to apply specified settings of the given dialog to all pages of the base document at once.

If needed, a user can add a page to the base document manually. To do it, the command **Add new Page** is used:

Keyboard	Textual menu	Icon
	Page > Add	

A user can delete the current page of the base document (under condition that it does not contain any images), with the help of the command **Delete Current Page**:

Keyboard	Textual menu	Icon
	Page > Delete Current Page	

Note that if on the apparently empty page there are hidden images (i.e., the images for which the flag next to the name has been taken off in the batch structure), then the removal of such image will be impossible. In this case the command **Delete All Hidden Images** can be used:

Keyboard	Textual menu	Icon
	Page > Delete Hidden Images	

This command removes from the document structure unused images located on the current page of the base document. After executing this command, the current page can be deleted.

Printing Composed Batch of Documents

A user can print out the composed batch of documents directly from the print module with the help of the following command:

Keyboard	Textual menu	Icon
	File > Print...	

This command in turn invokes the command **PT: Print Model**.

The composed batch of documents can be also sent to a printer from the main T-FLEX CAD window by calling the command **PT: Print Model** for the base batch of documents.

PRINT 3D

The Print 3D command activates STL export command:

Keyboard	Textual menu	Icon
<3PD>	File > 3D Print..	

When exporting to the “Stereo Lithography” format, the geometrical shape of the 3D model surfaces is translated by faceting surfaces into a set of triangular elements. The accuracy of the surface representation depends on the three-dimensional mesh parameters.

> More information about export to STL can be found in “Exporting and importing documents” chapter.

Service Commands and Tools

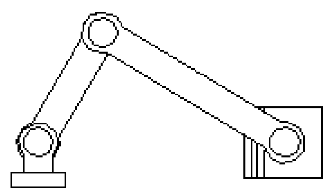
Animation

Animation is an effective mechanism that helps designing products. Animation helps analyze the behavior of kinematic mechanisms and mutual situation of parts in assemblies. Animation creation is a logical continuation of developing a parametric model. It allows visually inspect the impact of parameter modifications on the shape and position of the objects in the 3D scene, model operation of kinematic mechanisms, record and analyze the process of exploding the parts of an assembly. This system capability is yet another advantage of using parameterization when creating separate parts as well as assemblies. Analyzing a parametric model by means of animation allows mitigating errors at an early stage of designing a product. Use of a movable camera in animation allows creating realistic clips supporting the effects of zooming in and out, and spinning the object being viewed.

In animation, the system renders the state of the model under continuous modifications of its parameters. A requirement for creating an animation is presence of variables controlling the model parameters. For example, to have a part spinning with respect to a fixing point in animation, a variable is introduced whose value is equal to the rotation angle. In other words, the user must define variable parameters when building that model (geometrical shape or element positions) and assign the driving variables to these parameters.

T-FLEX CAD has two means for creating an animation. The first and simple one is using the command **AN: Animate Model.** When animating using this command, one can vary the value of one variable from its starting to the ending value with a specified step. A more powerful means for creating animations is a stand-alone application **Animation Screenplay**. This application provides control over an unlimited number of variables in the model and drive variable variations by complicated diagrams. A T-FLEX CAD document can have any number of animation scenarios included within, representing various model modification schemes.

Animating Model by Command "Animate Model"

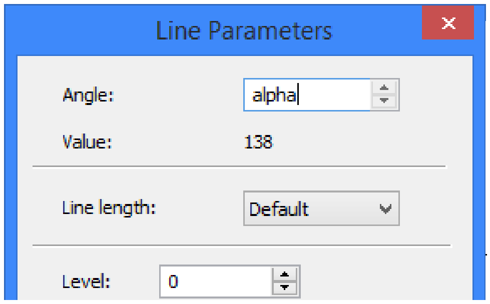


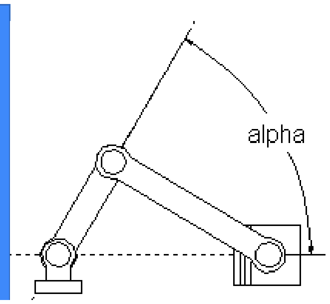
Drawing animation is implemented by a step-by-step modification to some parameter defined by a variable. The drawing is simultaneously redrawn at each step.

Suppose, we have created a drawing of a kinematic mechanism. The drawing is assembled from fragments, each of which is a link in the mechanism. Now, we would like to view the behavior of the mechanism as the position of the driving link is varied.

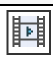
The driving link position is defined by the parameter of a construction line defined as a line passing through a node at an angle to the horizontal. This parameter is the angle of rotation.

One can assign a variable to drive this parameter. Let's call it *"alpha"*.

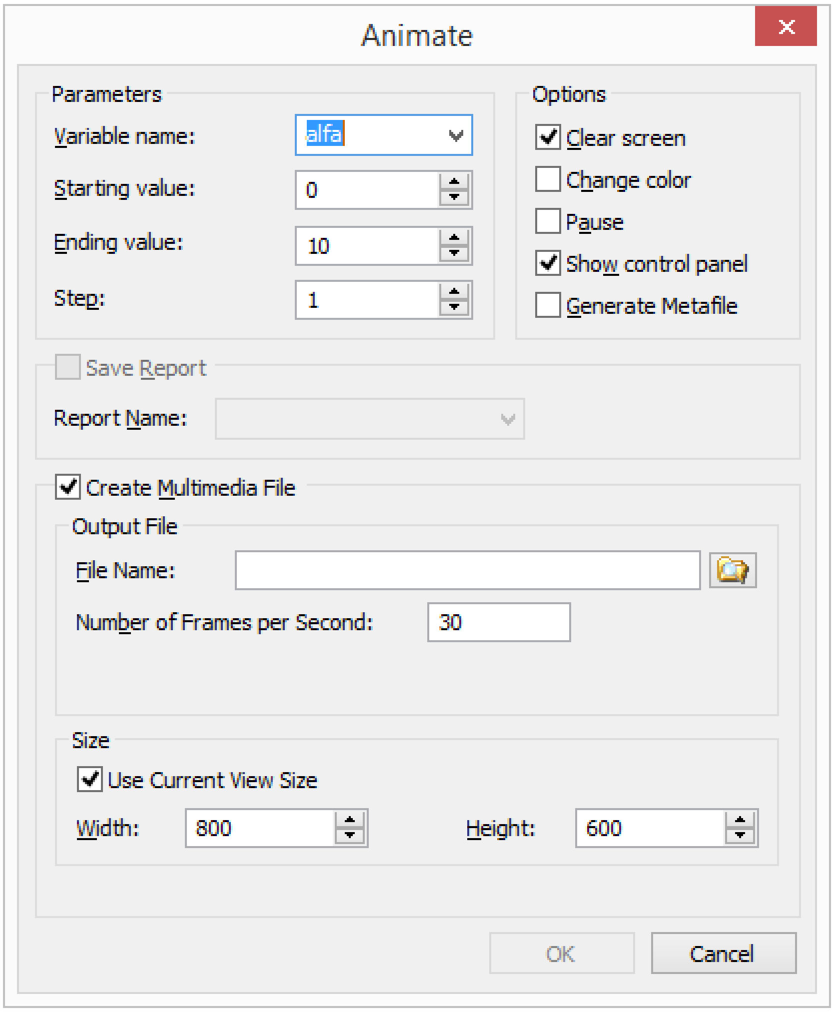




To "animate" the mechanism, use the command **AN: Animate Model**:

Icon	Ribbon
	Tools → Animation → Animate
Keyboard	Textual Menu
<AN>	Parameters > Animate

When entering the command from the 2D window, a dialog box with the command parameters is displayed on the screen.



The parameters are as follows:

Variable name. You can specify the name of the variable whose value will change. The variable's name can be specified manually or selected from the suggested list. All real variables of the drawing are included in this list. The variable cannot be a text variable.

Starting value. This is the starting value of the variable to vary.

Ending value. The target value of the variable at the completion of the command.

Step. The increment value added to the variable at each step of the animation.

Clear screen. With this option set, the screen will be redrawn at each step. Otherwise, the images will be overlapping to show the progress of the movement and drawing modifications at each step.

Change color. With this option set, the image at each step will be displayed in its own color. This option is useful when you want to compare results when varying the values of different drawing parameters.

Pause. With this option set, the system will require confirmation at each step, before drawing another frame.

Generate Metafile. With this option set, the animation clip will be saved in a T-FLEX CAD metafile. This metafile can later be output to a printer or plotter, included in a T-FLEX drawing using the command "**IP: Insert Picture**" or exported into another format. Note, that with the "Clear screen" mode turned on, a metafile will not be output.

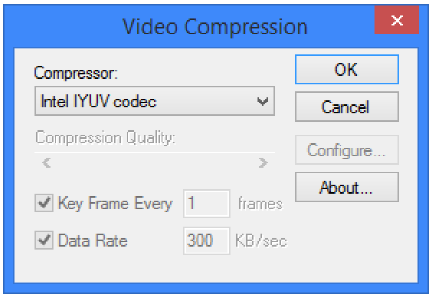
Save Report. With this option set, the result of animation at each step will be output in a log file (the filename must be specified in the operation parameters). This parameter is available only when the model has at least one report template, defined in the command **REP: Create Report**.

Generate multimedia file. Setting this option allows creating an *.avi file and defining its parameters:

File name.

Number of frames per second. The recommended frequency is 24 frames per second.

The graphic button **[Compression...]** for the *.avi format invokes the dialog box for customizing multimedia file compression parameters:



Compressor. Selects a compressing application.

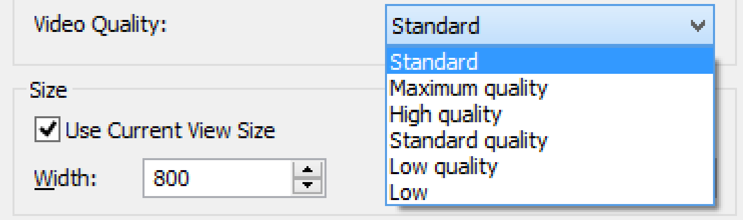
Compression Quality. Defines the quality grade for multimedia file compression.

Key Frame. Defines the number of frames between the key frames.

Data Rate. Defines the data transfer rate (kilobytes per second).

The graphic button **[Configure...]** brings up the dialog box for entering the required settings of the selected compressing application.

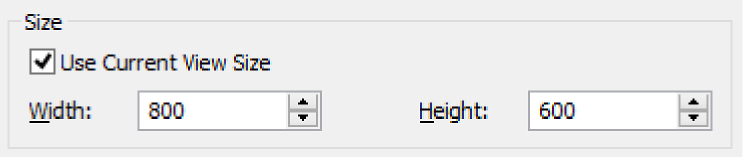
Video quality parameter is available for the *.wmv format.



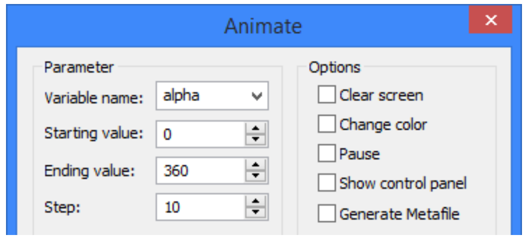
You can specify sizes for the video in the **Size** group.

Use current view size. With the flag turned on, the whole contents of the current 3D view window is recorded. With the flag off, the user can define one's own **width** and **height** parameters of the image being saved.

is specified the video size will be equal to the T-FLEX CAD window size.

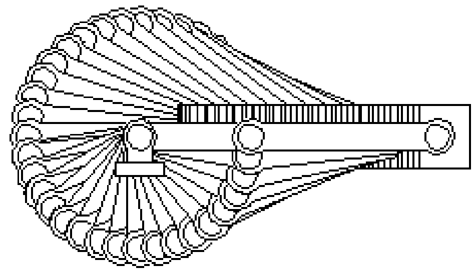


To interrupt a running animation, press <Esc>.



Returning to the above example, let's define the first four parameters.

The specified values realize the variation of the *"alpha"* variable from the value *"0"* to *"360"* with the step equal to *"10"*. The drawing will be displayed at each step.

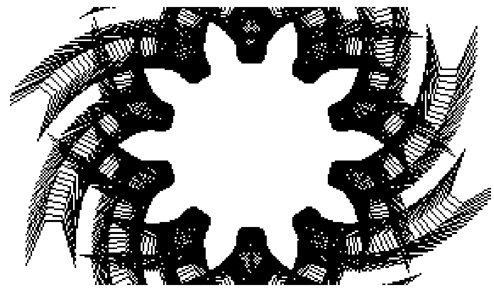


Press **[OK]**, and the result shown on the right hand side diagram will soon be displayed on the screen.

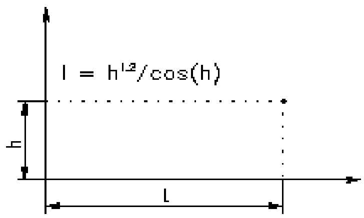
By performing these steps on your computer, you will see the drawing "animate".

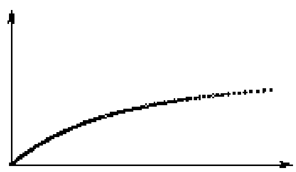
Consider a few more examples. In those, several variables are made dependent on the driving variable, which results in an interesting behavior.

An animation of the drawing of a cutting tool appears as if "machining" of a cog wheel:



Next is the example of a drawing that creates a function diagram by using animation.





What was entered in the variable editor is as follows: the variable *"h"* is equal to "`10`", and the variable *"l"* is defined by the expression *"^1.2/Cos(h)"*.

Note, that when calling the command **AN: Animate Model** while in the 3D window, the animation parameters dialog box has a slightly different appearance. In this case, to create a multimedia file, one needs to define the following groups of parameters:

You can select one of three variants in the **Photorealistic view generation** section.

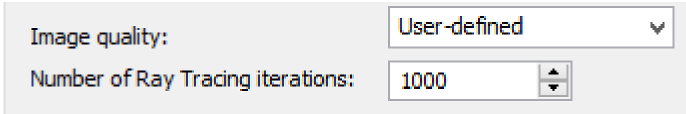
Non-photorealistic rendering – the exploded view scenario is recorded according to the current view in 3D scene.

The following variants are used for photorealistic view creation. Each frame of the video is processed using selected photorealistic view mechanism.

Photorealistic rendering using OptiX – the video is created using NVIDIA OptiX mechanism.

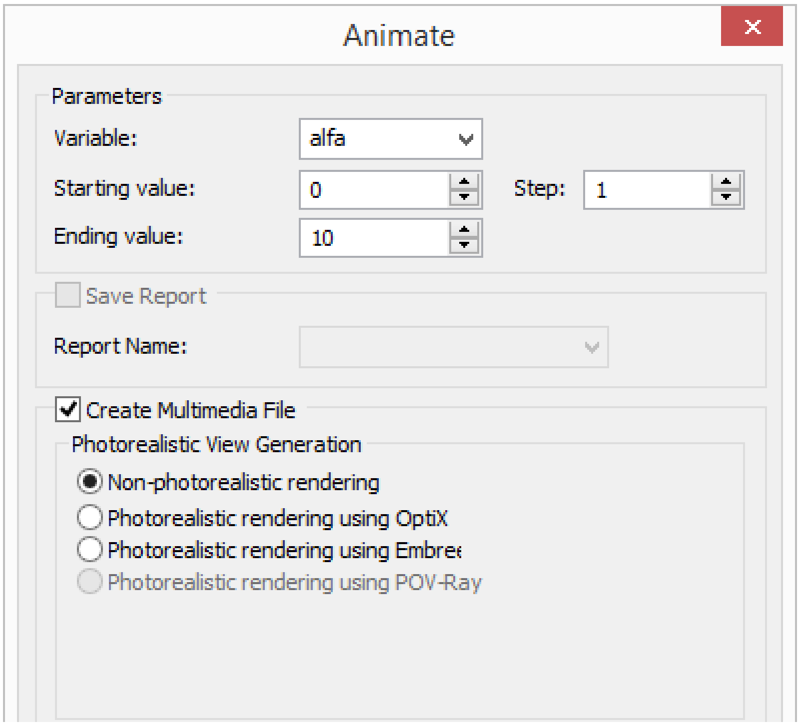
Photorealistic rendering using Embree – the video is created using Embree mechanism.

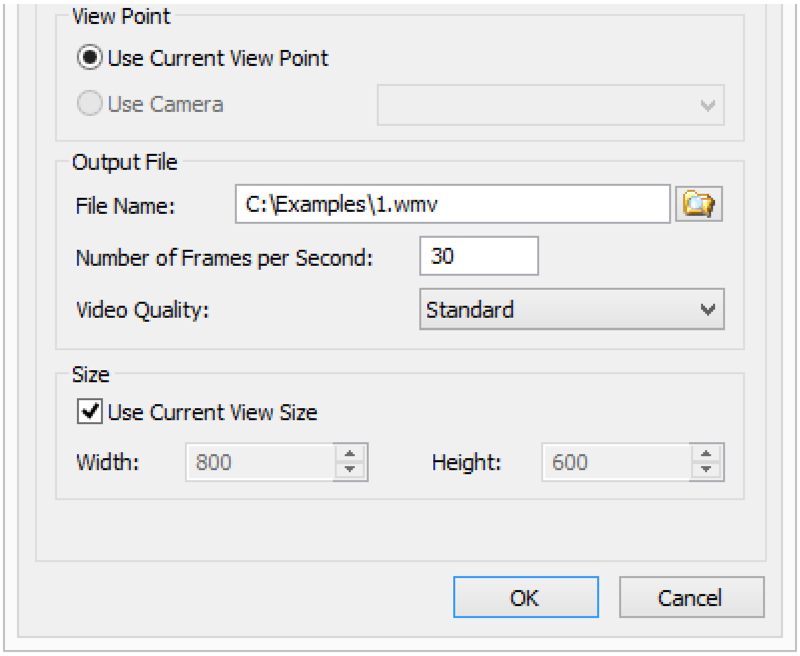
You can set **Image quality** and **Number of Ray Tracing iterations** for the previous two variants.



Photorealistic rendering using POV-Ray – the video is created using the POV-Ray mechanism.

You can find more information about photorealism in “Photorealistic view” chapter.





View point. Defines the position and direction of the animation recording camera.

Use current view point. The view point is defined by the system camera present in any 3D window by default.

Use camera. The other possibility is specifying a user-defined camera.

"Animation Screenplay" Application

This application is intended for making animations of 2D drawings and 3D models. Creation of the animation screenplay implies defining a dependency of several variable values in time. Such dependencies are introduced in the form of diagrams. Each diagram is represented in the time and variable value coordinates.

The animation functionality allows controlling parameter values and viewing or recording in a file the dynamically changing state of the model. The application is activated via the command **"AP: External Applications"**. If the application is running, then it can be called similar to other system commands:

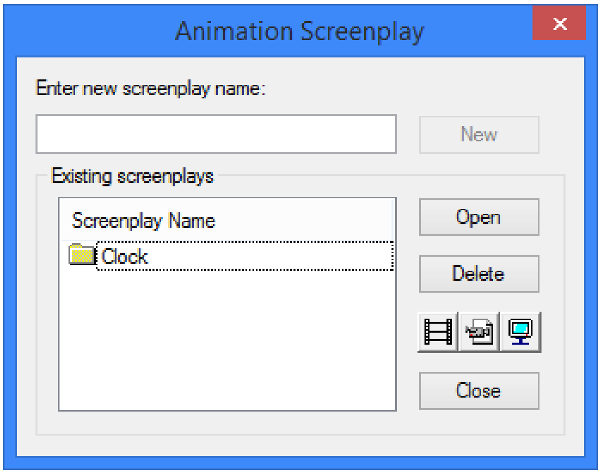
Icon	Ribbon
	Tools → Animation → Animation editor
Keyboard	Textual Menu
	Parameters > Animation editor

Let's review the general scheme of using the application when creating an animation screenplay:

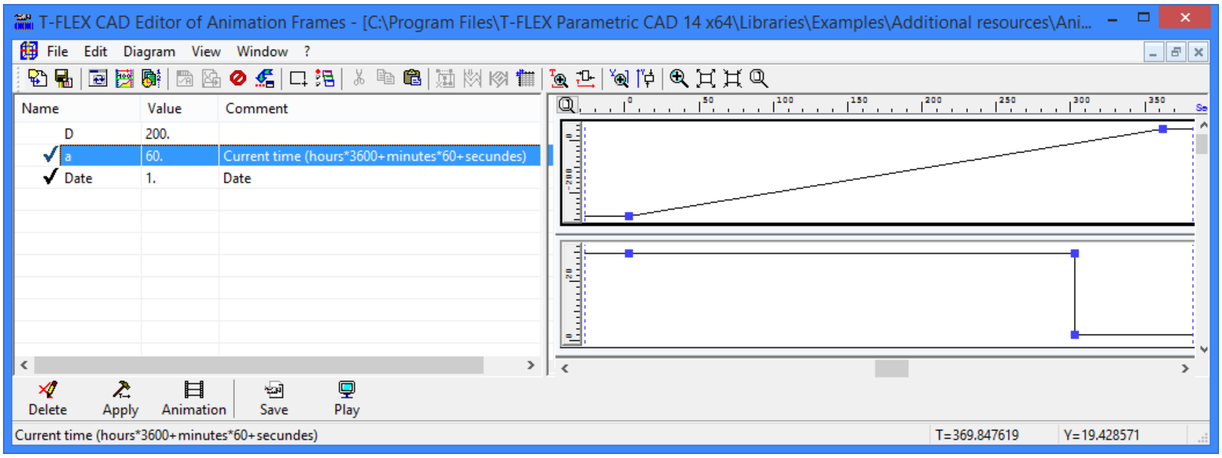
1. Forming diagrams of each variable's time dependency (defining the minimum and maximum values of the respective variable; setting points on the diagrams defining the variable values at specific moments in time).

2. Setting animation options and creating a multimedia file (*.avi).
3. Starting the animation and viewing the results. When forming an animation, the system defines the number of animation steps per the specified animation time range and increment. Then, the variable values are subsequently read from the created diagrams at each step, and the model is regenerated with the current set of parameters. The results of the animation can be recorded (if necessary) in a multimedia file (*.avi).

Note that the animation screenplay functionality is accessible only when the document has at least one variable that is not a textual (string) variable. A T-FLEX CAD document may contain several animation screenplays, defining the different ways of modifying variables. Upon calling the application, the dialog box is displayed on the screen for entering the name for a new screenplay (**[New]**) or selecting an existing one. If an existing screenplay is selected then the provided graphic buttons allow running, recording or reviewing the animation defined by this screenplay, or proceed with editing (**[Open]**) or deleting (**[Delete]**) the data.



Editing of an animation screenplay is done in the dialog box “T-FLEX CAD Editor of Animation Frames”. The dialog box is invoked by pressing the graphic button **[Open]**. The variations of each variable are defined as separate diagrams. The left pane of the dialog box keeps the list of the model variables, their values and comments. Driven variables (defined by a function or an expression) are displayed on a different background (particulars of handling those will be described below).



Creating New Diagram

Click the desired line in the variable list with the mouse, and then select the following option in the context menu or under the “Diagram” menu entry:

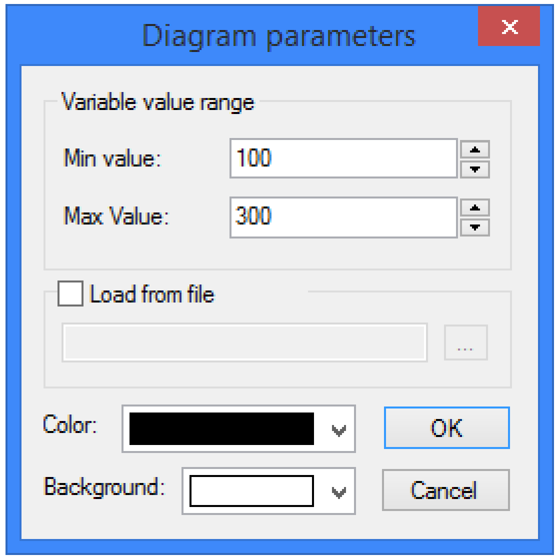
	<Ins>	Add Window With Diagram For Selected Variable

A dialog box will appear next for defining the diagram parameters. The default range of the variable values is set from ½ to 1½ of its current value.

In future, the specified values can be modified using the option:

	<C>	Set Diagram And Its Window Properties For Selected Variable

The “Load from file” flag allows reading in an existing diagram saved in a file (*.tfg). The “Color” and “Background” define the color of the diagram and its background window. Upon confirming the diagram parameters, it's pane opens. The ruler on the left side of the diagram pane represents the scale for the variable values, while the one on the top side of the diagram pane reflects the time scale. Both scales reflect the defined range is of the values (the default time interval is from 0 to 60 seconds). At the beginning, the variable diagram is a constant equal to the current variable value. The user can now add an arbitrary number of points, defining the variation of the variable values along the specified time interval.



If the screenplay already has another variable diagram in the window, then upon selecting a variable, the following option becomes available:

	<G>	Add Diagram For Selected Variable To Active Window

The current window is outlined by a frame, whose color is defined by the parameter “**Active window color**” per the option:

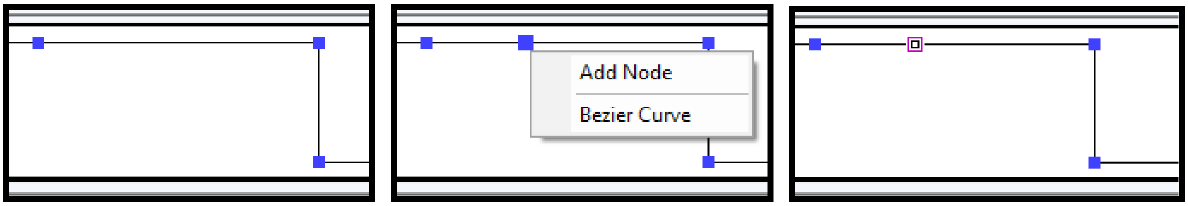
		Change Current Screenplay Name

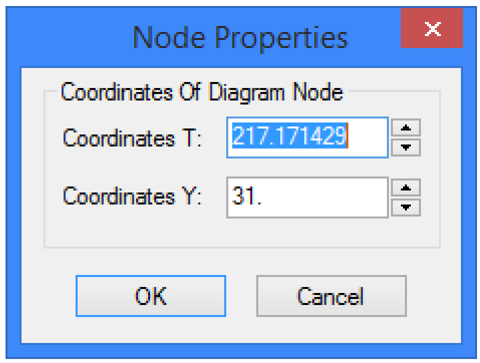
To select the current window, point the mouse to the desired diagram and click . The option of adding a variable to the current window allows overlapping several diagrams in one coordinate space. If several diagrams are present in one window, the last modified diagram is considered selected. Its line is thicker, and the respective entry is highlighted in the list of a variables. A diagram sharing the window with other diagrams can be moved to a separate window using the option:

		Move Active Diagram Into New Window

Adding/Modifying Diagram Points

As the pointer moves inside the diagram window, its current coordinates are displayed in the lower right corner. Upon clicking , a new node is created under the pointer position, and is connected by straight segments with the neighboring points in the diagram. To create a new node dividing an existing segment of the diagram in the two, point the mouse to the diagram line and select the item "**Add Node**" from the context menu.





To modify the node position, point the mouse to it (the mouse pointer will change its appearance). Then, dragging the mouse with the depressed, place this node of the diagram in a different position. To specify the exact node coordinates, right click , select in the coming up automenu the item "**Properties**" and enter the values for the T- and Y-coordinate.

In the case of multiple selection of the nodes in the diagram, the selected group of points can be translated:

	<M>	Transfer Selected Nodes

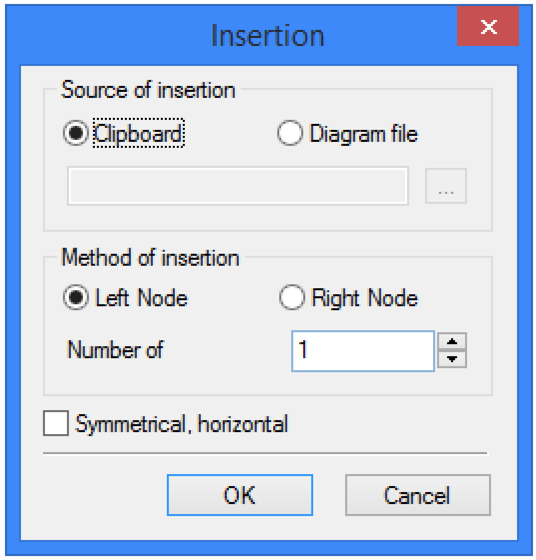
Upon selecting this option, a dialog box appears for inputting the shift values along the T- and Y-axes. As a result, the selected group of points will change its position per the specified values.

Adding points to a diagram can also be done using the option:

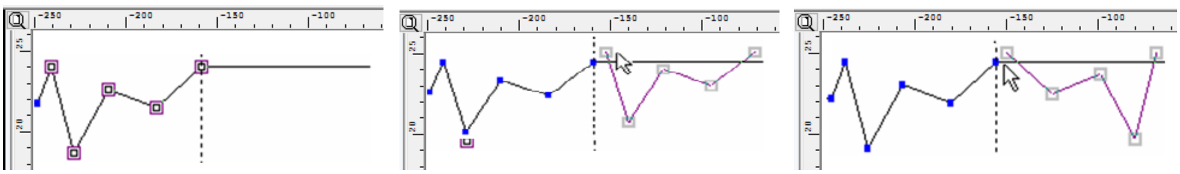
		Cut Selected Nodes Of Active Diagram And Put Them Into Clipboard Or File

		Copy Selected Nodes Of Active Diagram Into Clipboard Or File
		Paste Nodes With Positioning Into Active Diagram From Clipboard Or File

Use of these options implies existence of least one diagram with two nodes. Select several nodes (those will be highlighted) and pick one of the options, “Cut” or “Copy”, in the control toolbar. Then select the fixing node for inserting the selected fragment of the diagram and pick the option “Paste”. Thereafter, the “Insertion” dialog box appears.



Besides the Clipboard, a source for insertion can be a diagram saved in a file (*.tfg). The insertion particulars are defined by the fixing node of the diagram fragment being inserted (“Left Node”, “Right Node”) and by the number of insertions. If the flag “Symmetrical, horizontal” is set, then the insertion fragment will be a copy of the original mirrored about the vertical line passing through the fixing node.



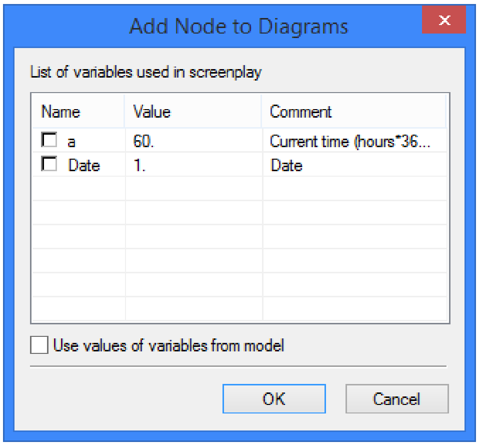
The leftmost picture shows the highlighted points of the diagram fragment being copied to the Clipboard. The second picture shows the insertion with the option **Left Node**, the third one - that with additional use of the flag **Symmetrical, horizontal**.

In the case when the time span of the diagram fragment being inserted is greater than the time interval of the remaining portion of the diagram to the right of the fixing node (when using left node fixing), or to the left of the fixing node (for right node fixing), the system aborts the insertion operation. This is done to prevent violation of the time segments order.

To add nodes to the selected diagrams at the point of the intersection with the *main ruler*, that defines the current time instant (highlighted in blue), use the option:

		Add Nodes To Selected Diagrams By Time Axis Of The Main Ruler

Upon calling the option, a window appears containing the list of variables for each of the existing diagrams.



The column **Value** contains the variable values at the current time instant. Check the variables in the list, whose diagrams are to be added nodes. Upon pressing the **[OK]** button, new points will be created in the respective diagrams at the point of the intersection with the time main ruler marker.

Setting the flag **Use values of variables from model** positions the new nodes along the time main ruler, using the variable values defined for the model. In this case, the new node is connected by straight segments to the neighboring points in the diagram.

This capability allows defining diagrams by modifying variables in the model and then adding nodes with their values at the certain time instant.

Selecting Diagram Nodes

Point the mouse at the desired node and click – the point on the diagram will be outlined by a box. Multiple selection is done by one of the following means:

- With the <Shift> key held down, a sequence of nodes is selected between the two specified ones;
- With the <Ctrl> key held down, the directly picked nodes are selected.

Selection of all nodes in the diagram is done by the option:

	<Ctrl><A>	Select All

The option is called from the context menu over the diagram area.

Box selection of a group of nodes is done with the option turned on:

	<S>	Select Group Of Diagram Nodes

Deleting Diagram Nodes

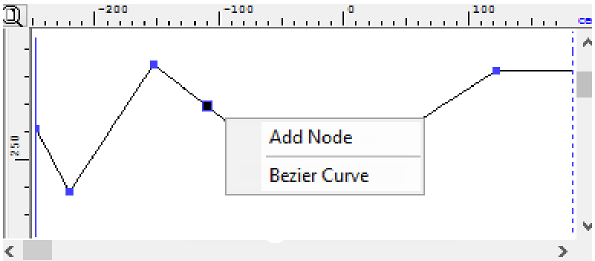
The following option is provided for deleting selected nodes:

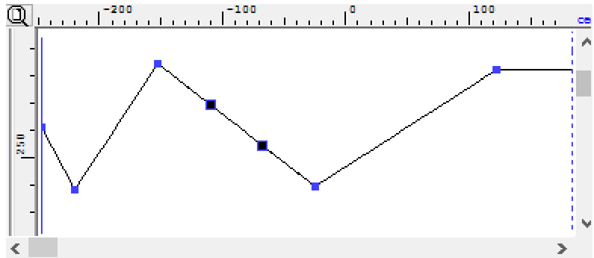
	<D>	Delete Selected Node Or Group Of Nodes

The option is available under the menu entry **Edit** or from the context menu. Once a node(s) is deleted, the neighboring points in the diagram are connected by straight segment.

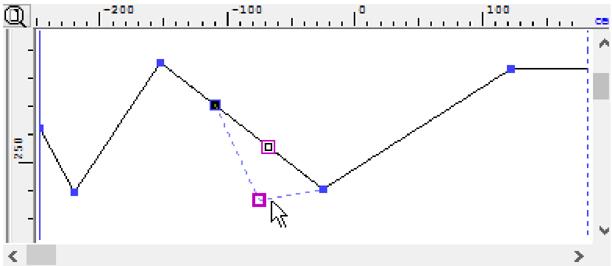
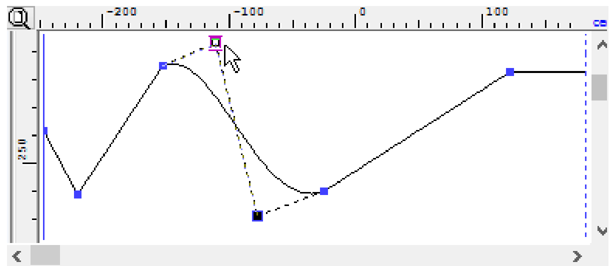
Smoothing Diagram

Upon adding new nodes, the diagram is formed as a polyline. If necessary, you can obtain a smooth curve by replacing the polyline segments by curved segments. To do this, point the mouse to the desired segment in the diagram and right click, and then select the item **Bezier Curve**. Two nodes will appear on the selected segments dividing it in equal parts.





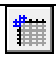
The segment can be transformed into a curve by modifying the position of each node.

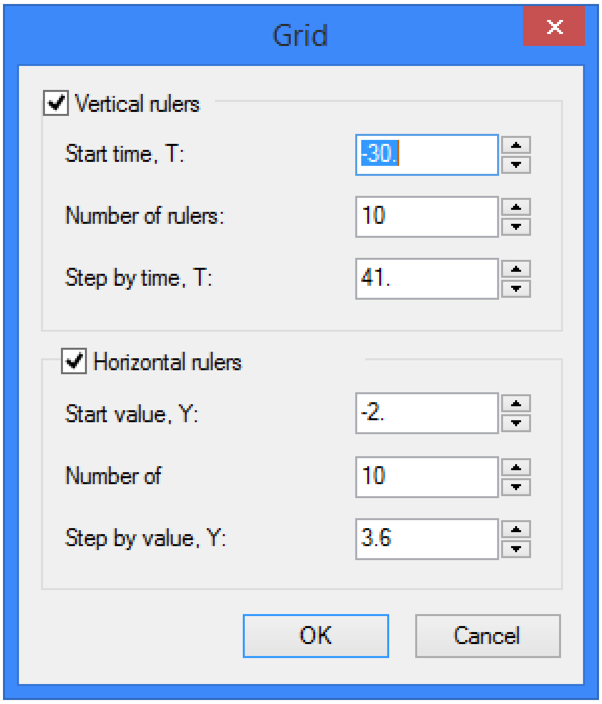
The reversed transformation of a curve segment into a straight segment is done by the command "**Straighten**", available in the context menu upon its selection.

Grid

To apply vertical and/or horizontal ruler lines in the diagram pane for easy orientation in the coordinate space when constructing diagram points, use the option:

		Construct Grid Of Rulers In Active Window

The respective flags in the **Grid** dialog box control the presence of vertical and horizontal rulers. For each type of ruler, the following parameters are defined:



Start Time, T. Defines the position of the first ruler;

Number of rulers. Defines the number of rulers;

Step by time, T. Defines the distance between rulers.

To define a ruler crossing at a point with specific coordinates, use the option:

	<R>	Define Ruler Crossing For Diagrams In Active Window

The option is provided under the menu **Edit**. To add an individual vertical of horizontal ruler, set the mouse pointer at the image of the scale (time or variable), and click . In the coming up menu select the item **Ruler** and specify the point on the scale, for the ruler being created to pass through.

Driven Diagrams

If the model has variables defined by functions or actions, then it is possible to obtain a special type of a diagram – "**driven**". Since the values of such variables are dependent values, the system creates their diagrams automatically. The lines corresponding to the dependent variables in the list of the animation screenplay dialog box are grayed out. A "driven" diagram cannot be edited; it merely illustrates the variation of the variable during an animation. Note that a driven diagram is regenerated upon starting an animation. This type of a diagram can be used for analyzing mutual positions of various model elements. A variable can be created for this purpose, whose value would be the function of the distance measured between two bodies, vertices, etc. When modifying the model parameters, the diagram of such a variable will reflect changes in the distance concerned.

General Animation Parameters

Besides the described above, a number of options are available in the toolbar and under the “**Edit**” menu entry.

	<T>	Set Limits For Time Scale

This option sets the limits of the Time axis portion visible in the diagram pane.

		Change Current Screenplay Name

This option allows saving the current screenplay under a different name and/or change the color of the outlining frame highlighting the active diagram window (the parameter **Active window color**).

		Save Active Diagram To Text File

This option allows saving the current diagram or the selected nodes in a text file (*.tfg). In future, the file contents can be inserted into another diagram or another screenplay.

	<F7>	Update List Of Variables For The Current Model

This option reloads the changes in the model variable list.

	<Alt>+<P>	Set Parameters For.avi File

This option allows defining the parameters for recording an .avi file. The parameters are the same as the respective ones in the command **AN: Animate Model**.

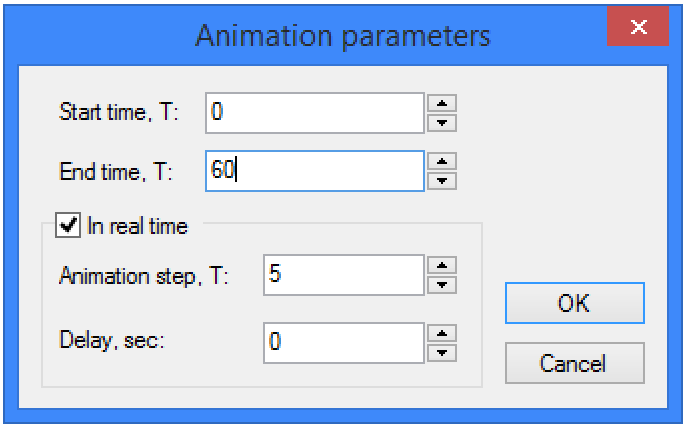
	<Alt>+<A>	Set Limits, Step And Delay For Playing Animation

This option defines the values of the following parameters:

Start time, T. Defines the time instant from which the animation should begin.

End time, T. Defines the time instant at which animation ends.

In real time. With this flag set, the animation is performed in real time. The latter option means the following. At the beginning, the model is regenerated with the variable values defined for the starting instant of the animation. The next set of values corresponds to the instant when the first model regeneration completes. Therefore, in this case, the animation step is actually the duration required for the model regeneration.



Animation step, T. Defines the time step for sampling the variable values.

Delay, sec. Defines delay after each step of the animation.

		Do Not Use Variable Values From Diagram In Animation

This option allows excluding a variable variation diagram when forming an animation (in this case, the above icon appears before the variable name in the variable list). A repeated call to the option removes the variable diagram block.

Zooming Diagram Window

When working with the diagram, it is often necessary to change the scale of the diagram image. The listed below options allow manipulating the drawing image in various ways.

	<Ctrl>+<T>	Zoom In By Time Scale

Two points specified by clicking , define the boundaries of the new interval on the Time scale.

	<Alt>+<T>	Zoom Out Active Window By Time

This option results in doubling the current interval on the Time scale.

	<Ctrl>+<Y>	Zoom In By Values Scale

Two points specified by clicking define the boundaries of the new interval on the Value scale.

	<Alt>+<Y>	Zoom Out Active Window By Values Scale

This option results in doubling the current interval on the Value scale.

	<Z>	Zoom Window

Two points specified by clicking , define the box diagonal. The vertical and horizontal size of the box area in the window defines the displayed intervals on the Time and Value scales.

	<Ctrl>+<PgUp>	Zoom In Active Window

Doubles the current intervals along the T and Y-axes.

	<Ctrl>+<PgDn>	Zoom Out Active Window

Halves the current intervals along the T and Y-axes.

	<Ctrl>+<G>	Zoom All Diagram Nodes For Active Window

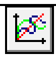
This option pans/zooms the current window in such a way to fit all the existing diagrams in this window.

Manipulating Animation

The following graphical buttons are located in the bottom portion of the animation screenplay dialog box (alternatively, those are accessible under the “Diagram” menu entry or from the context menu, invoked over the list of variables):

	<Del>	Delete Diagram For Selected Variable

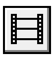
This option is available for variables with existing diagrams. It deletes the diagram of the current variable.

	<Ins>	Add Window With Diagram For Selected Variable

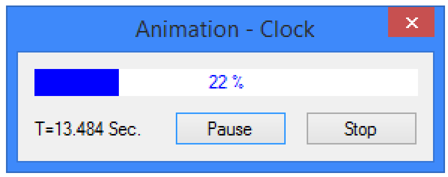
This option is available for variables without diagrams. It creates the variable diagram in a new window.

	<P>	Apply Current Variable Values For Animation

This option regenerates the T-FLEX CAD model with the variable values per the diagram readings at the current time instant. The current time instant is set using the time ruler (highlighted in blue on the Time scale). As the Time ruler is moved, the variable values are automatically updated in the list.

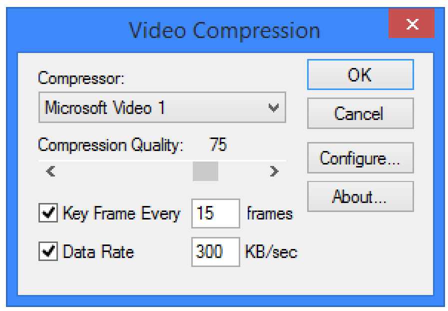
	<A>	Animation In T-FLEX CAD On The Current Values Of Variables

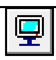
This option starts executing the animation. A sequence of the 3D model (drawing) modifications is displayed, reflecting the model regeneration per the variable values defined in the screenplay. The animation execution is controlled by the buttons **[Pause/Continue]** and **[Stop]** in the “Animation” window.



	<F>	Save Animation To File(*.avi)

This option is for recording an AVI-file. Upon starting the recording, the system offers to select a compressing application of the video image. The “Video compression” dialog box allows selecting one of the installed applications on your computer and setting its parameters.



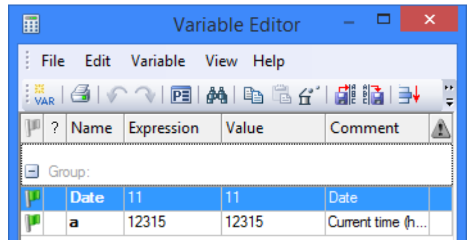
	<V>	Play Video (*.avi) File For The Current Screenplay

This option plays back the recorded AVI-file.

Animation helps visually illustrate the influence of parameter modifications on the shape and/or spatial situation of the objects, model the function of kinematic mechanisms and check permissible ranges of the variable values.

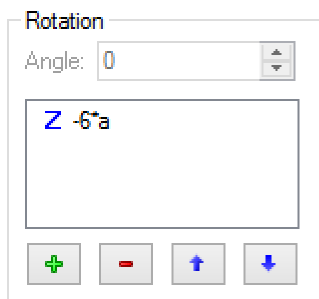
EXAMPLE: CLOCK TICKING ANIMATION

A simple and instructive example of using animation for simulating a mechanism motion can be a model of a clock. This model is provided with the library "`Examples`", the folder "`Additional resources\Animation\Watch`". The model contains an animation screenplay, modeling a working clock: moving arms and changing date.

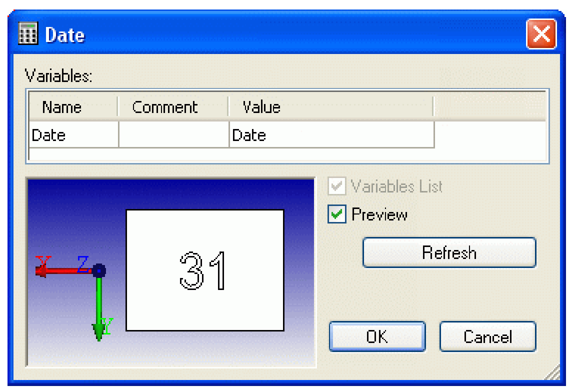


To animate the model, the variables "a" and "Date" are introduced. The variable "a" defines the current time in seconds (it will drive the position of the clock arms), while the "Date" variable – the figure displayed in the date dial of the clock.

The clock arms and the date dial are 3D fragments. Since the arms are supposed to rotate in the animation, each arm fragment has an angle parameter defining the angle of rotation with respect to the Z axis via an expression dependent on the variable "a": for the seconds arm it is "-6*a", for the minutes one - "-a/10", and for the hours – "-a/120".



The date dial fragment must update the displayed figures during the animation. For this purpose, an external variable "Date" is created in the fragment, defining the numeric figures in the fragment. Upon assembling the fragment, this variable is commanded by its namesake in the assembly.



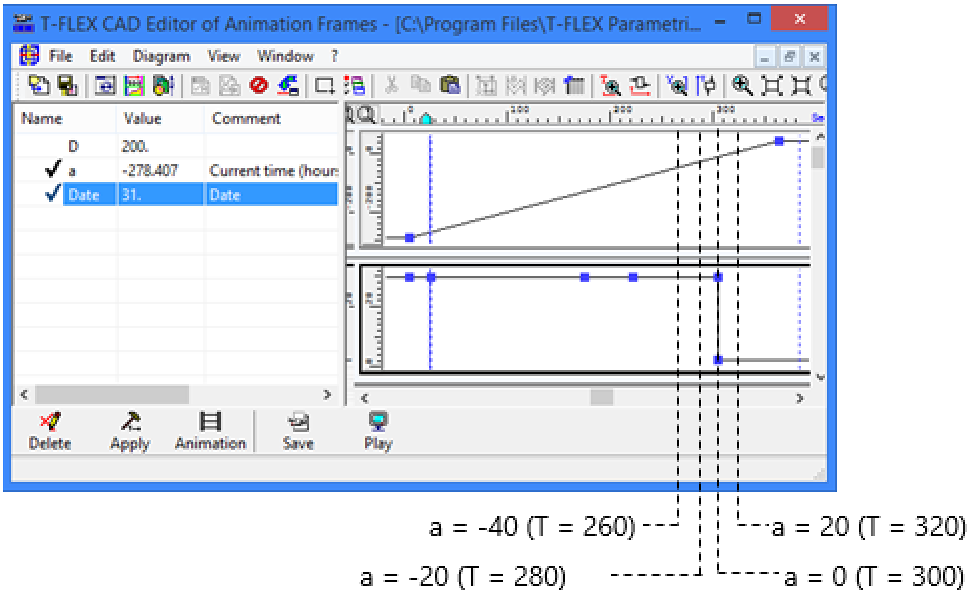
The animation is supposed to simulate the time from 23: 55 to 0:01. During the animation, besides the arms motion, the date shall change on the date dial (at the midnight).

Therefore, two independent variables, "a" and "Date", shall vary their values. The model is "animated" by the **Animation Screenplay** application, taking into the account this mentioned feature.

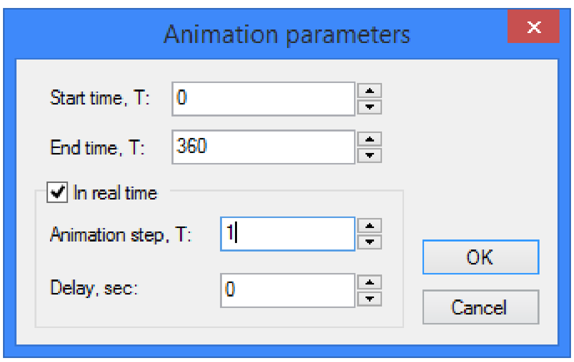
The "Clock" animation screenplay uses the variation diagrams for the variables "a" and "Date". The animation time interval T is assumed from 0 (5 minutes before midnight) to 360(1 minute of the midnight). The time T=300 corresponds to the midnight, that is the moment of changing date. The variable diagrams are created according to the following tabulated values:

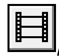
T	0	300	360
a, 0	-300	0	60

T	0	300	360
Date	31	31/1	1



The animation parameters define the necessary interval and step of the time variation. The step T=1 corresponds to the clock arms motion with the beat rate of one second.



Now, start running the animation using the option , and you will see a "live" model of the clock on the screen: the arms moving and the date changing from the 31st to the 1st at "midnight". The pictures on the right hand side show some snapshots of the animation corresponding to the time instants reflected on the diagrams.

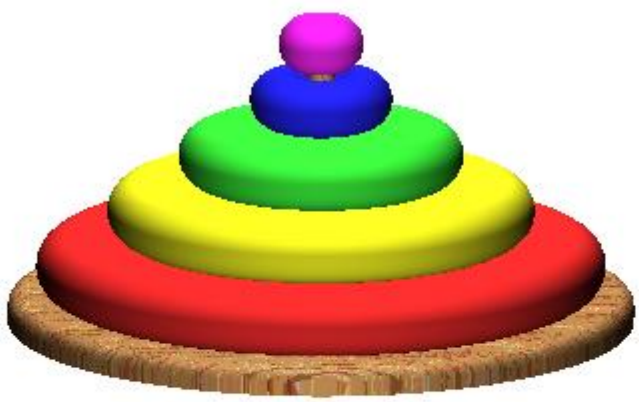
For viewing the animation "in the real time", check the respective flag among the animation parameters. The animation step will then be the time of the model regeneration. The arms movement in this case may not correspond to the normal clock arms movement.

When creating a video clip, the image playback speed depends on two parameters: the animation step and the frame frequency. In this particular example, the natural clock ticking can be obtained by specifying the animation step equal to T1/N, where T1 is a portion of the animation corresponding to the seconds arm advancing by one notch, and the N is the number of frames per second defined in the parameters of the video clip recording.

EXAMPLE: DISASSEMBLING A PYRAMID

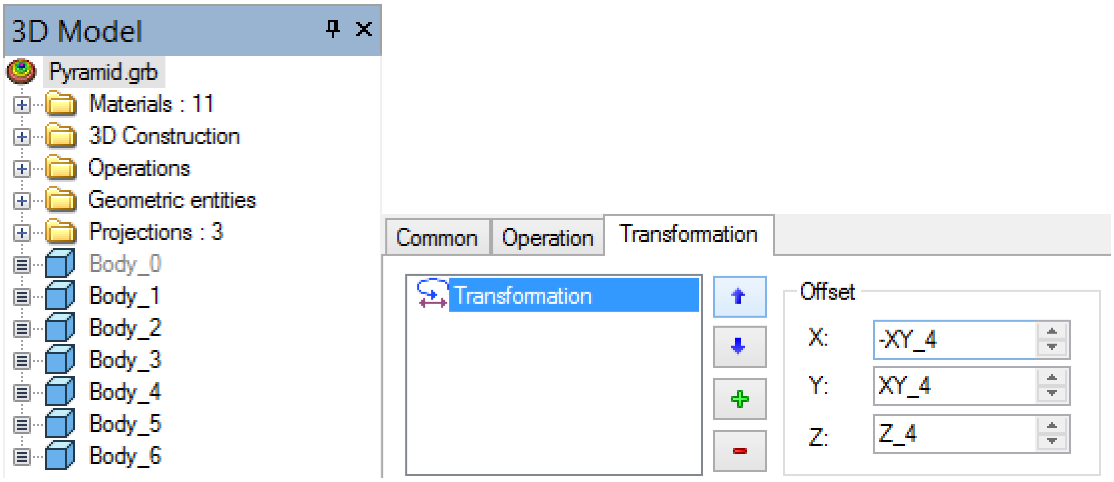
Another example of animation is the animation of taking apart a toy pyramid.

> The 3D model of the pyramid is located in the library "`Examples`", the folder "`Additional` resources\Animation\Pyramid".



The model of the pyramid is made of a base with a pole, stacked up four rings, and a topping. When disassembling, the base stays still, while the rings and the topping are "undone" one by one from the base and put around.

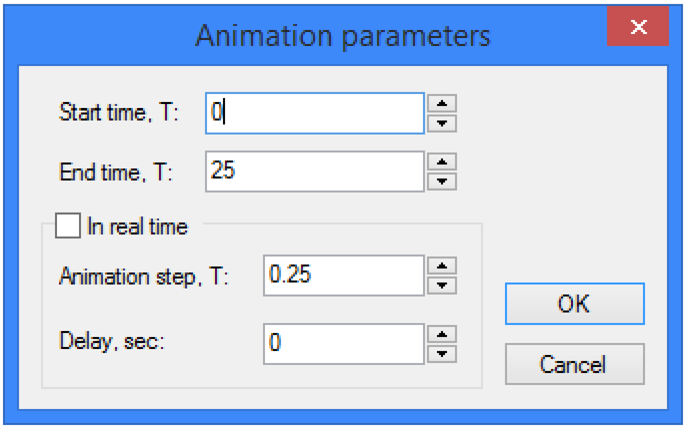
To achieve this, the shifts from the axes for the respective 3D operations are defined by variable-based parameters. The variables Z_1, Z_2, Z_3, Z_4, Z_5 will define the shifts along the Z-axis, corresponding to the vertical movement of the rings (along the axis of the pole). The variables XY_1, XY_2, XY_3, XY_4 and XY_5 describe the movement of the rings and the topping in the horizontal plane along the X- and Y-axes (removal of the rings off the base).



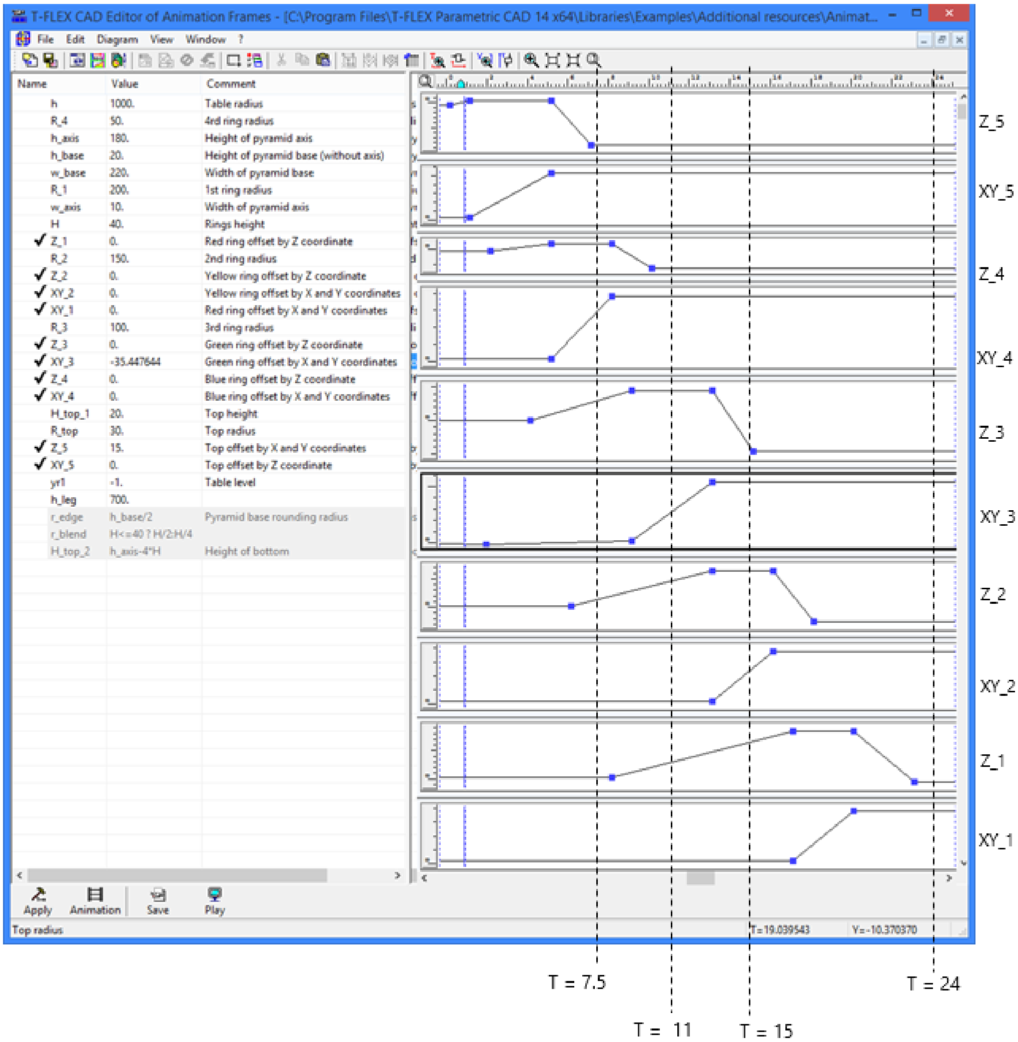
Such a large number of varied variables require use of the **Animation Screenplay** application.

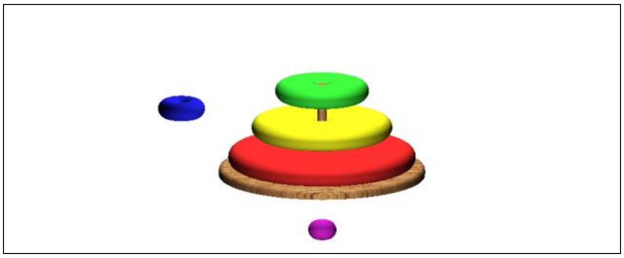
Let's define the diagrams for the model variables varying with time in the animation screenplay. Two diagrams correspond to each body being moved (for the topping, those are the diagrams of the variables Z_5 and XY_5, for the blue ring – the diagrams Z_4 and XY_4, etc.). One of those defines the object moving in the vertical plane, the other one - in the horizontal. The diagrams are designed in such a way, that each ring first moves up along the pole (the values of the respective variable Z in the diagram increase). At reaching the top of the pole (the flat portion of the Z variable diagram), a ring starts moving in the horizontal plane at some distance from the base (the growing portion of the respective variable XY diagram), and then goes down (the descending portion of the Z variable diagram). The rings start moving sequentially with the time delay interval of T=2.

Once all diagrams are created, let's define the animation parameters: the interval T from 0 to 25 (the last ring finishes its movement at the time instant T=23), and the step equal to 0.25.

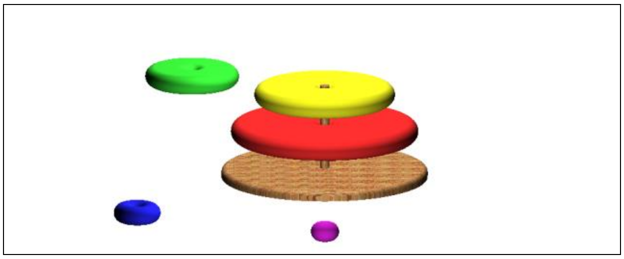


Upon starting the animation, the process of taking the pyramid apart will be simulated on the screen. The pictures below show certain animation stages corresponding to the time instants shown on the diagrams.

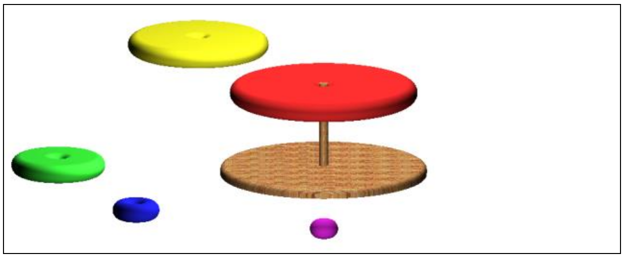




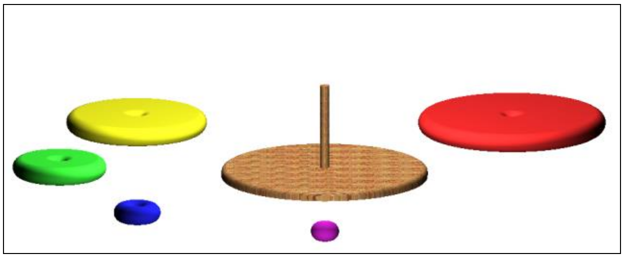
T = 7.25



T = 11



T = 15



T = 24

Preview/Slide

Preview is a vector graphic (T-FLEX CAD Metafile) or bitmap (Windows BMP) image of an arbitrary size saved together with the drawing file. It serves for quick previewing of the document contents in the preview window, when opening the document or inserting it as a fragment, a picture or a 3D picture.

> You can find out whether the preview is available within the given drawing file, when the document appears in the window "Library Explorer", by picking the item "Properties" from the context menu invoked by the .

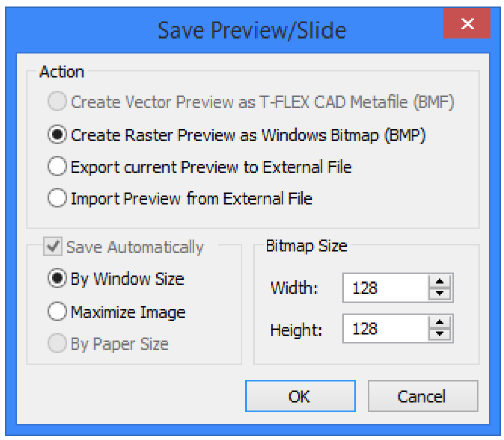
Document icon – small, raster picture of fixed size (16x16, 32x32, 48x48 or 64x64 pixels). It is displayed on the tab of the open document, in the list of documents found in the Library Explorer.

The mode of automatic preview and icon creation can be set in the command **ST: Set Document Parameters** on the tab **Save**.

Creating Preview

The command **PV: Save Preview** is provided for creating a preview. The command is called by one of the following means:

Icon	Ribbon
	Tools → Special Data → Preview
Keyboard	Textual Menu
<PV>	Tools > Special Data > Preview



Upon calling the command, a dialog box appears on the screen. This dialog box serves for creating a preview (in the BMF or BMP format), importing a preview from an external file and exporting a preview into an external file.

A preview (bitmap) stored in an external file can be edited using a graphic editor, such as "Paint".

Creating preview in the T-FLEX CAD Metafile (BMF) format is possible only upon invoking the command from 2D window. Such preview is created on the basis of the image of the active 2D page of the current document. The area of the page, the image of which will copied into the preview, can be determined by the system automatically or specified by the user.

For automatic determination of the area of the drawing for which preview is created, the flag **Save Automatically** should be enabled. The state of the toggle below the flag determines how the system will select the area:

By Window Size. The preview image is created by the image of the current 2D window;

Maximize Image (By image size). The preview image is generated by the whole image of the current page of the drawing;

By Paper Size. The preview image is created by the image of the drawing found itself within the drawing border of the given page.

After turning on the flag “Save automatically” and setting up the desired toggle state, it is enough to press **[OK]** and the preview will be created in the current document.

To manually specify the area of the drawing, the flag **Save Automatically** should be turned off. In this case, after pressing**[OK]** and closing the dialog window “Preview”, in the 2D window it is necessary to specify the rectangular domain of arbitrary size. To do it, bring the cursor to one of the corners of the intended rectangle and press , then bring the cursor to another corner and press again. After that, the preview creation will be completed.

Preview in the raster Windows Bitmap (BMP) format can be created when invoking the command both from the 2D or 3D window. As in the previous case, the preview is created on the basis of the whole image or part of the image of the active 2D page. The used area of the page can be determined by the system automatically or specified by the user.

The size of the created raster picture (in pixels) is specified via the group of parameters **Bitmap Size**. The same parameters determine the dimensions of the rectangular box which is used for specifying manually the area of the drawing for creating the preview.

Creating preview with automatic determination of the area of the drawing is carried out in the same way as for the BMF format: it is necessary to set on the flag **Save Automatically**, select the desired approach for determination of the area of the drawing and press**[OK]**.

For manual specification of the area of the drawing, the flag **Save Automatically** should be turned off. After pressing **[OK]** and closing the dialog window of the command, a rectangular box attached to the cursor will appear on the screen (its size will coincide with the size of the picture being created). To create preview, bring the box to the desired area of the 2D window and press .

> The settings of the command **ST: Set Document Parameters** can be disabled in favor of the described command settings. This will happen if you use the value “None” or “Auto” in the **Save** combo box on the tab **Save** of the command **ST: Set Document Parameters**. Upon confirming the parameter settings, a message would be displayed that the preview would be created manually.

The preview created in the current document can be exported to the external file of the BMF or BMP format (according to in which format the preview of the current document was created). And vice versa, any picture from the external file of the BMF or BMP format can be imported to the current document. To do it, after invoking the command, select in the list the required operation (**PV: Save preview**), and then in the appeared standard window for saving/opening files specify the name of the external file.

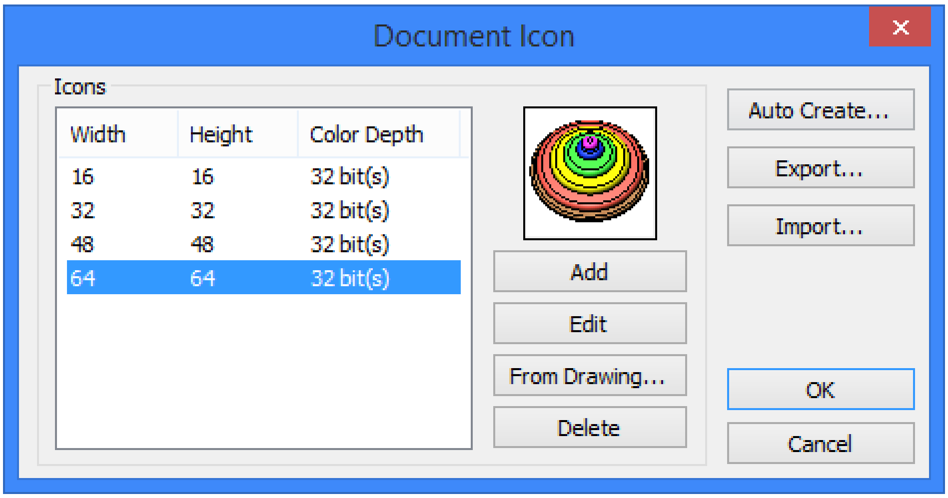
The preview saved in the external file of the BMP format can be edited by using tools of any graphics editor.

Creating Icons

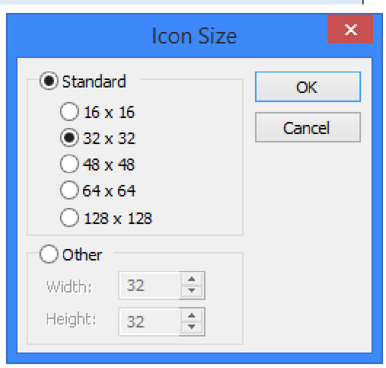
To create the document icons, there is a command **IC: Create/Edit Document Icon**:

Icon	Ribbon
	Tools → Special Data → Icon
Keyboard	Textual Menu
<IC>	Tools > Special Data > Icon

After invoking this command, the dialog window **Document Icon** is opened. In the field **Icons** of this dialog, there is a list of icons created in the current document. After selecting the icon in the list (with the help of), in the preview area (to the right of the list) the image of the selected icon appears. If there are no icons in the current document, the list will be empty.



> Recall that icons can be created automatically. The regime of automatic icon creation is set in the command "**ST: Set Document Parameters** (tab **Save**).



To create an icon in the current document, the buttons [Add] ("manual" creation of icon image) or [From Drawing...] (automatic creation of icon image) can be used. When pressing any of these two buttons, the dialog appears in which a user will be prompted to select the size of the created icon: one of the standard values (16x16, 32x32, 48x48, 64x64 pixels) or arbitrarily specified size.

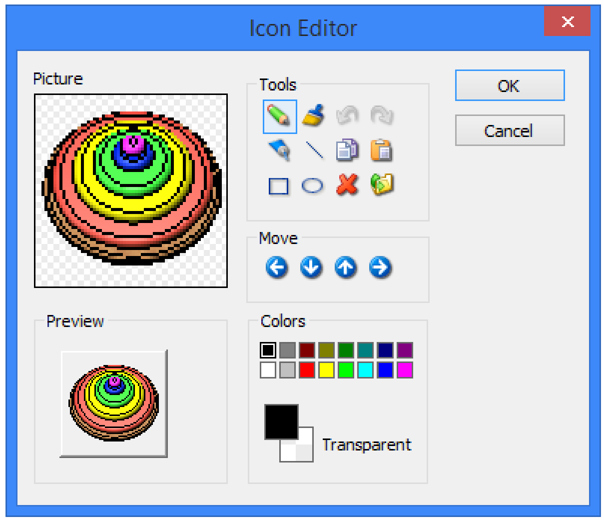
Large icons (of size 32x32, 48x48, 64x64 pixels) can be used in the windows of the T-FLEX CAD libraries.

After selection of the icon size with the use of the button [Add], the window of the icon editor (see below) is opened.

When the button **[From Drawing...]** is used, the command response after selection of the icon size depends on which way the command **IC: Create/Edit icon** was invoked. In case it was invoked while the 2D window was active, the window of the command gets closed temporarily, and the rectangular box snapped to the cursor appears on the screen. The box allows a user to select the area in the 2D drawing on the basis of which the icon image will be created. If the command was invoked from the 3D window of the system, then the icon image will be created on the basis of the contents of the entire 3D window of the current document.

There is one more button in the command dialog which allows a user to create icons – button **[Auto Create...]**. This button creates all four icons at once (of size 16x16, 32x32, 48x48, 64x64 pixels). The icon image is specified in the same way as when using the button **[From Drawing...]**.

In addition to creating icons, in the command **IC: Create/Edit Document Icon** it is possible to edit and remove already existing icons. To edit the icon, select it in the list (with the help of) and press the button **[Edit]**. As a result, the window of the icon editor will be opened in which the current icon image can be modified.



To remove already existing icon, it is enough to select it in the list and press the button **[Delete]**.

The button **[Export...]** allows a user to save all created in the current document icons in one external file of the "*.ico" format. With the help of the button **[Import...]**, it is possible to load the icons from the external file.

Exporting and Importing Documents

T-FLEX CAD allows exporting its 2D drawings and 3D models into other graphical system formats.

Exporting from T-FLEX CAD is supported for the following formats:

Only for **2D drawings:**

- Windows Metafiles (*.wmf);
- Enhanced Windows Metafiles (*.emf);
- AutoCAD files (*.dwg);
- AutoCAD DXF files (*.dxf);
- AutoCAD DXB files (*.dxb);
- T-FLEX CAD Metafiles (*.bmf);
- T-FLEX CAD Metafiles without hidden lines (*.bmf).

For **2D drawing** and **3D models:**

- Raster Image (*.bmp, *.jpg, *.gif, *.tif, *.tiff, *.png);
- Files in PDF format (*.pdf)

Only for **3D models:**

3D documents

- Parasolid files (*.xmt_txt);
- Parasolid Binary files (*.xmt_txt);
- STEP (*.stp, *.step);
- IGES 3D (*.igs, *.iges);
- JT (*.jt)
- ACIS (*.sat, *.sab)
- PRC (*prc);

Mesh geometry

- AutoCAD DXF 3D files (*.dxf);
- Stereo Lithography (*.stl);
- PLY files(*.ply);
- OBJ files(*.obj);
- VRML 2.0 files (*.wrl);
- U3D files(*.u3d);
- X3D files (*.x3d);
- POV-Ray (*.pov);

- Open Inventor files (*.iv);
- Rhino Model (*.3dm);
- IFC (*.ifc);
- 3MF (*.3mf);
- T-FLEX Scene files (*tf3d).

Importing in T-FLEX CAD is supported from the following formats:

Only for **2D drawings:**

- AutoCAD files (*.dwg);
- AutoCAD DXF files (*.dxf);
- AutoCAD DXB files (*.dxb).

Only for **3D models:**

- Parasolid (*.xmt_txt, *.x_t, *.x_b, *.xmt_bin);
- AutoCAD DXF 3D/DWG 3D (*.dxf, *.dwg);
- STEP (*.stp, *.step);
- IGES 3D (*.igs, *.iges);
- ACIS (*.sat, *.sab);
- SolidWorks (*.sldprt, *.sldasm, *.sldlfp, *.asm);
- Autodesk Inventor (*.ipt, *.iam);
- Siemens NX (Unigraphics) (*.prt);
- Creo (ProE) (*.prt, (*.prt.*, *.neu, *.neu.*, *.asm, *.asm.*, *.xas, *.xpr);
- Catia V5 (*.CATPart, *.CATProduct, *.CATShape);
- Catia V4 (*.model, *.dlv, *.exp, *.session);
- Solid Edge (*.asm, *.par, *.psm, *.pwd);
- Rhino (*.3dm);
- I-Deas (*.arc, *.unv, *.mf1, *.prt, *.pkg);
- VDA-FS (*.vda);
- JT (*.jt);
- PRC (*.prc);
- 3dxml (*.3dxml);
- CGR (*.cgr);
- U3D (*.u3d);
- IFC (*.ifc).

3D Images

- TF3D (*.tf3d);

- Open Inventor (*.iv);
- AutoCAD (*.dwg, *dxf, *.dxb),
- VRML 2.0 (*.wrl);
- X3D (*.x3d);
- 3DS (*.3ds);
- PLY (*.ply);
- OBJ (*.obj);
- STL (*.stl).

When porting models from T-FLEX CAD to other systems and from other systems to T-FLEX CAD, you are strongly advised to prefer the STEP to the IGES format whenever possible as the more advanced one.

An additional license is required to import in the Creo/ProE, Catia V5, Catia V4, I-DEAS formats.

Besides the above-listed formats, the 3D configuration of the system supports import of *.x3d, *.ply, *.obj, *.3ds, *.dxf, *.tf3d, *.stl files, VRML 2.0 (*.wrl), AutoCAD (*.dwg, *dxf, *.dxb) and Open Inventor (*.iv) files by the command **3I: Insert 3D Picture**. This capability is described in details in the chapter “3D Pictures” of the Three-dimensional Modeling User Manual.

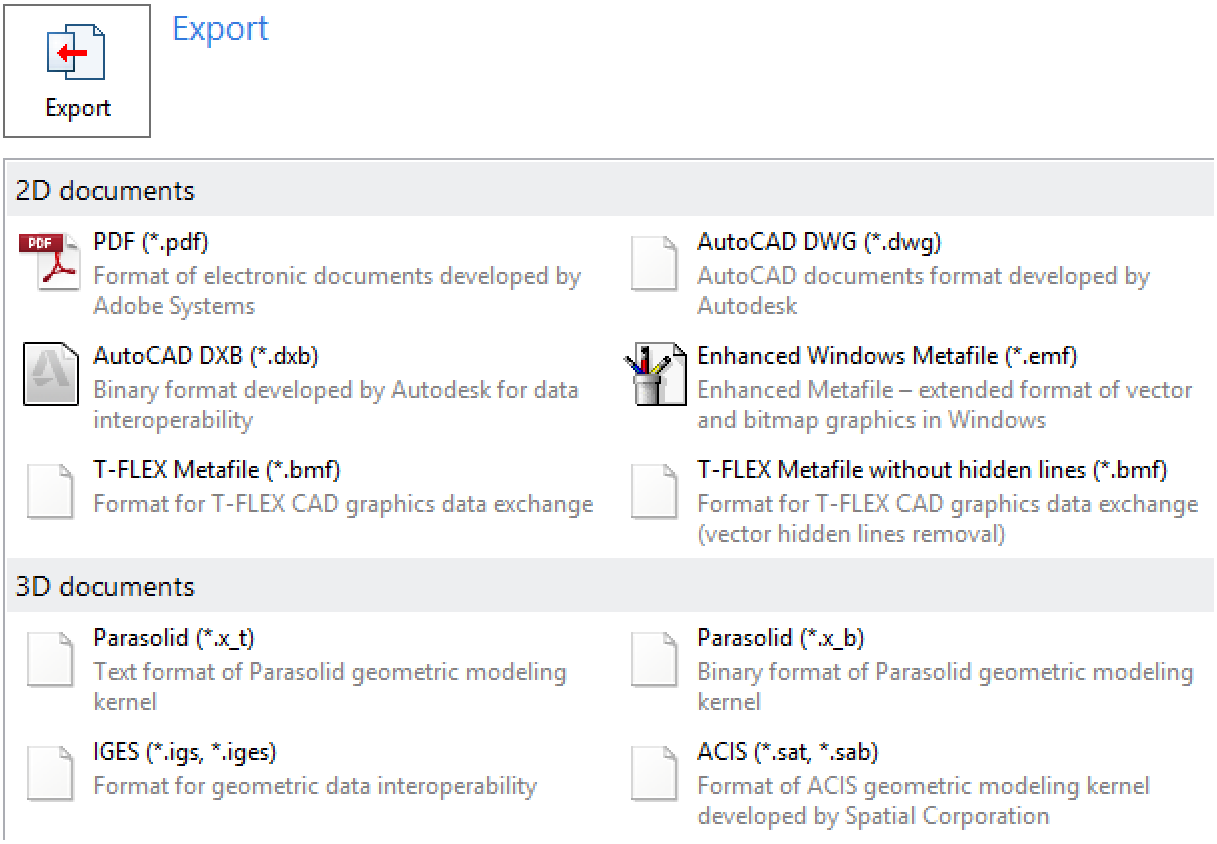
Documents of raster images in formats *.bmf, *.bmp, *.jpg, *.jpeg, *.gif, *.dib, *.pcx, *.tga, *.tif, *.tiff, *.png, .grb drawings, Windows metafiles (*.wmf and *.emf) may be inserted into a 2D drawing with the help of **IP: Picture** command.

Exporting Documents

Exporting T-FLEX CAD documents into other system formats is done by the command **EX: Export Drawing Or Model**:

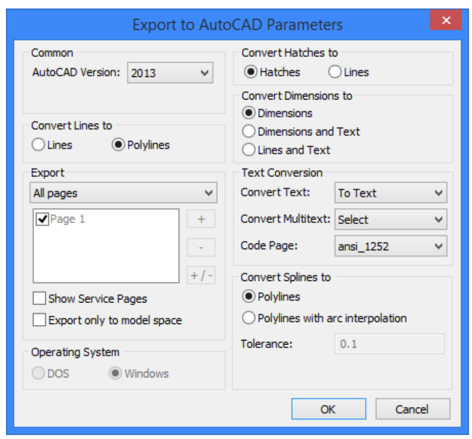
Icon	Ribbon
	File → Export
Keyboard	Textual Menu
<EX> or <Ctrl><W>	File > Export

After calling the command, the dialog with formats available for export appears. You should select one and press **[Export]** button on the top of the screen.



Next, depending on the selected export format, either the file will be saved at once (as in the formats: Windows Metafiles, Enhanced Windows Metafiles, T-FLEX CAD Metafiles, Metafiles without invisible lines), or additional parameters, described below, must be specified (for all other formats).

AutoCAD System Formats: DXF, DXB, DWG



When exporting 2D drawing into one of the AutoCAD system format (DXF, DXB or DWG) specify the following parameters:

AutoCAD version;

Convert Lines to, **Convert hatches to**. These parameters determine into what AutoCAD system elements the T-FLEX CAD lines and hatches will be converted;

Convert dimensions to. For T-FLEX CAD dimensions, one can select one of the following export options:

Dimensions. In this way of converting dimensions, T-FLEX CAD dimension parameters are translated into AutoCAD dimension parameters. This may suffer some loss of dimension text data that is not supported by AutoCAD (for example, tolerances containing letters in their notation). Converting some dimensions not supported by AutoCAD could also create additional strokes, notations and text unrelated to the AutoCAD dimension, used for accurate representation of the respective T-FLEX CAD dimension. For example, this is possible when exporting certain dimensions with the text put on a bent leader or on a leader extension.

Dimensions and Text. Selecting this option also makes AutoCAD dimensions (sometimes, with additional lines). However, the textual fields of the resulting dimensions get hidden by AutoCAD. Instead, those get objects of the type “Text” inserted that appear as T-FLEX CAD dimensions yet are not related to the AutoCAD dimension.

Lines and Text. In this case, AutoCAD dimensions are not created in the conversion. The T-FLEX CAD dimension images are translated as lines and text.

Text Conversion. This is the group of parameters defining the method of exporting text:

Convert Text. This parameter defines the method of converting elements of the type “String text”: **to text** or **to line** strokes in AutoCAD;

Convert Multitext. This parameter defines the method of converting “Multiline text”, “Paragraph text” and “Table” text types. Three options are available in the combo box:

To Multitext. T-FLEX CAD text of the specified types is converted to AutoCAD multi-text (mtext). Subscripts, superscripts, GD&T symbols which are part of T-FLEX CAD text content, are not translated to AutoCAD. The contents of T-FLEX CAD tables is written in one column. The fractions contained in T-FLEX CAD text are converted correctly only upon the condition that those do not use nested fractions;

To Text and Lines. T-FLEX CAD text is exported to ordinary AutoCAD text and lines;

Select. During the conversion, T-FLEX CAD text is checked for presence of tables, fractions, subscripts, superscript, GD&T symbols, roughnesses, etc. If any of those entities is found, then the text is exported into ordinary AutoCAD text and lines. Otherwise, the text is converted to AutoCAD multi-text.

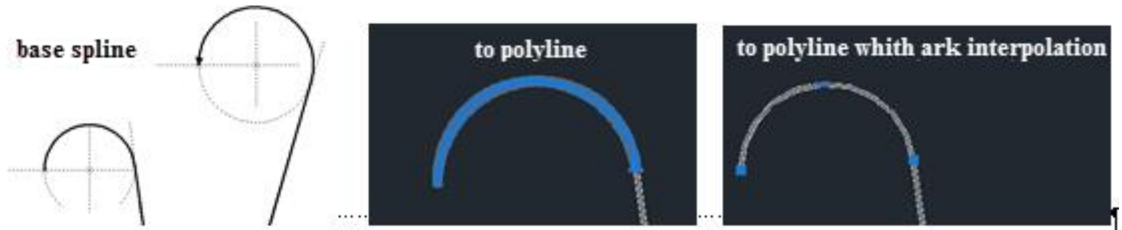
Code Page. This parameter allows selecting the desired text encoding. The default encoding corresponds to the language used in T-FLEX CAD. When exporting a file that contains characters of a language other than that used in T-FLEX CAD, you may need to change encoding.

Convert splines into. Parameter that controls the transfer of graphic lines-splines from GRB file to the AutoCad file:

Polylines. Spline is created as a polyline with division into segments.

Polylines with interpolation by arcs. Segments of the spline are replaced by arcs where it is possible.

Tolerance. This parameter specifies allowable offset of the location of polyline points from the original spline.



The “**Export**” group allows us to find out which pages of the original T-FLEX CAD drawing must be exported into AutoCAD document. The drop down list of export modes offers the following options:

All pages;

Current page;

Selected objects. In this case only the objects which were selected on the drawing are exported;

Selected pages. Only the pages selected in the list below will be exported. Page selection is carried out in the same way as upon printing out a document.

Export only to the space of the model. When this parameter is enabled, the export is carried out to the tab “Model”, but the pages are not created in the file (layouts).

> If the drawing being exported uses SHX fonts missing in AutoCAD, you can add those to AutoCAD by copying those fonts into its working directory.

The earlier versions of T-FLEX CAD (8.0 and prior), a different export/import mechanism was used for exchanging data with the AutoCAD formats. The old mechanism did not support AutoCAD 2004/2005 files format and the binary *.DXB format. When exporting a T-FLEX CAD drawing into AutoCAD, one could not select specific pages of a drawing that needed to be exported. Besides, the conversion method of T-FLEX CAD elements into AutoCAD elements that was used in the old mechanism might not always deliver correct translation of all drawing elements. Nevertheless, the old data exchange mechanism had its own advantages. For example, it supported selection of the operating system for exporting (for data exchange with old AutoCAD versions), as well as a text conversion method.

To use the old mechanism of data exchange with AutoCAD format for exporting, you would need to check an additional flag **Use Version 8 Algorithm**. This mechanism can be used only in the 32-bit version of T-FLEX CAD. Activating the flag enables the parameters of the old mechanism and blocks the parameters inherent solely to the new mechanism T-FLEX CAD. Described below are the parameters pertaining solely to the old export mechanism:

Convert Dimensions to. T-FLEX CAD dimensions conversion into AutoCAD elements in the old mechanism occurs differently. The following options are available:

Dimensions. When exporting a dimension, an AutoCAD block is created in which all strokes and text are added that were related to the dimension. This block “links” to the AutoCAD dimension. A shortcoming of such approach is the fact that the dimension appearance may change significantly when regenerating in AutoCAD.

Lines and Text. In this method, no AutoCAD dimensions are created; instead, all T-FLEX CAD dimension data translates into lines and text.

Text Conversion\Convert Text. This parameter determines into what AutoCAD system elements the T-FLEX CAD text will be converted:

To Text. The original drawing text will be translated into AutoCAD text. The **Code Page** parameter allows selecting the desired text encoding;

To Lines. The original T-FLEX CAD text are converted into a set of line strokes.

Operating System. This parameter allows selecting the type of an operating system: DOS or Windows. Default is Windows; however, if the file is being exported into an older AutoCAD version running DOS, you should change the radio group setting.

> When using the old data exchange mechanism, you cannot select an AutoCAD version higher than 2000.

Raster Image (*.bmp, *.jpg, *.gif, *.tif, *.tiff, *.png)

The following parameters are defined in the raster image export dialog:

"**File**" group of parameters displays and allows to edit the name and format of the output file defined in the previous step.

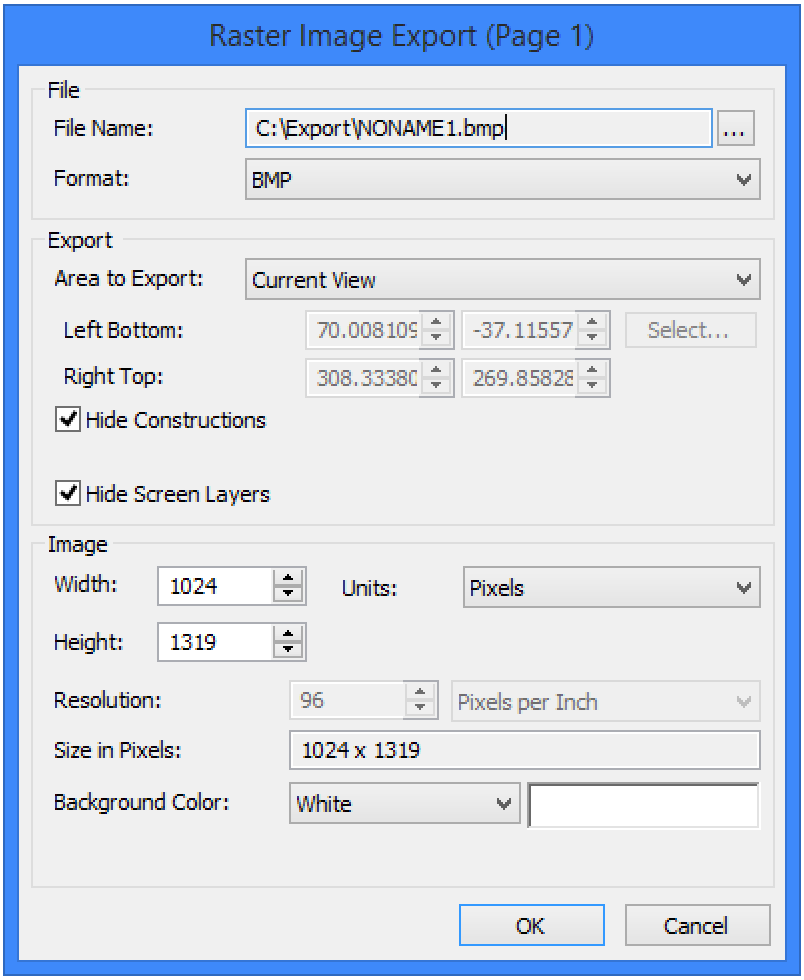
"**Export**" group of parameters defines the export image:

Current View. This setting means exporting the current window image;

Current Page. This setting means the current 2D page image will be exported;

Rectangular Area. This setting serves to manually define a rectangular area of the 2D page, whose content will be recorded in the resulting file. There are two ways to specify the area:

1. By defining the **lower-left and upper-right corner** coordinates of the area (**Left Bottom** and **Right Top**). The coordinates are entered in the respective fields;



2. By selecting a rectangular area in the 2D window. To select an area, click the button **[Select...]**. The dialog window will temporarily disappear from the screen, and you will be able to specify the desired area with the cursor. To return to the dialog, click  in the automenu.

An additional **Hide Constructions** flag (set by default) has an effect on exporting 2D and 3D construction elements. When this flag is set, the construction elements are not exported.

Please note that the **Area to Export** parameter, as well as the fields to define a rectangular area, are available only when calling the export command from a 2D window. If calling the command from a 3D window, then the **Current View** setting is forced. Besides that, when called from a 3D window, the command dialog gains another flag – **Hide Annotation Elements**. When this flag is set, the 3D annotation elements are not exported.

The **Hide screen layers** flag is also available only when exporting from a 2D window. This parameter determines whether the drawing elements from the layers for which the "Screen" option is set will be exported.

Image group of parameters serves to define parameters of the file being created: the image dimensions, resolution and background color:

Width, **Height**. These are the width and height of the image in the resulting file. If these parameters are set, the system will force saving the image aspect ratio.

Units. This parameter defines the units in which the height and width of the image are specified: **Pixels**, **Millimeters**, **Centimeters**, **Inches**.

Resolution. This parameter defines the resolution of the file being created. You can additionally select the resolution units: **Pixels per Inch** or **Pixels per Centimeter**.

An info parameter "**Size in Pixels**" serves to assess the size of the resulting file.

Background Color. This parameter defines the background color of the picture being created: **Auto** (same as the active window background), **White**, **Black**, **Other** (any color from the drop-down list at the right).

If you export a 3D scene, the option **Transparent** is available for PNG format. The option allows you to set a transparent background for the image.

Quality. This parameter is available only when exporting the *"*.jpg"* format. It defines the rate of compression for the file. The higher the compression quality setting, the less image loss and the larger the file size will be.

PDF Format

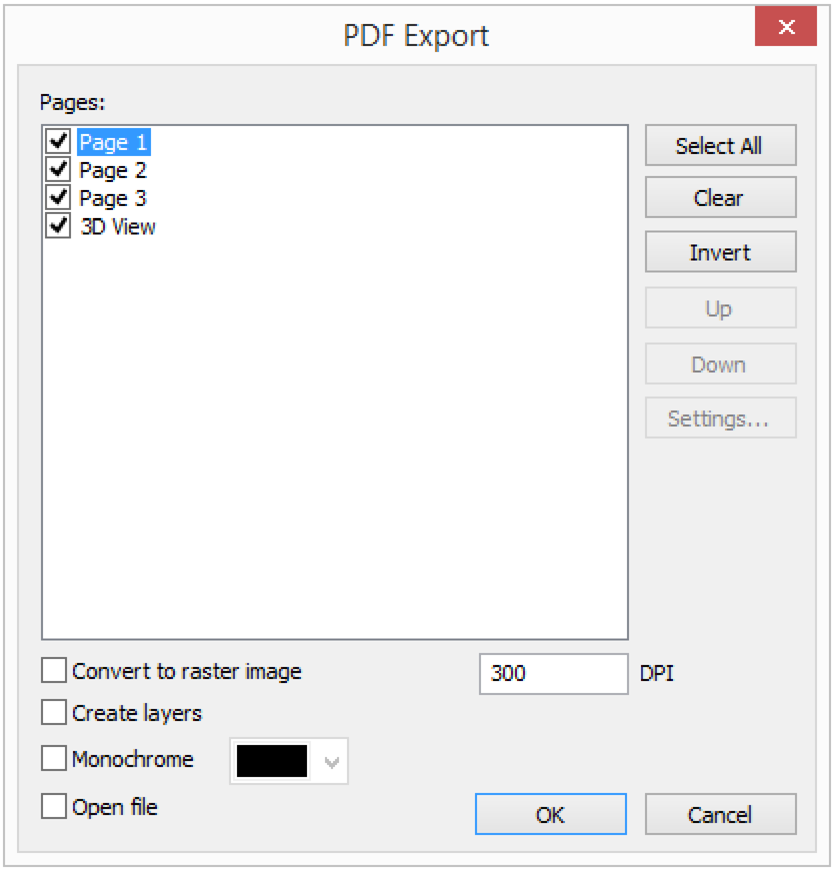
You should select a path for saving the exported file after calling the command for exporting to PDF. A dialog window appears after that. Here user can specify export parameters.

In the **Pages** field user should select pages for export.

Buttons to the right of the field allows to control the selection of pages for export.

Buttons **Up** and **Down** allows to position the 3D view at the beginning of the PDF document or at the end.

The **Settings** button is available for the 3D view. The button calls the 3D PDF export dialog.



Convert to raster image. All pages will be converted to raster images. You can specify image resolution in the field to the right.

Create layers. If the flag is set, layers will be created in the PDF document. The layers will be similar to the layers in the T-FLEX CAD document.

Monochtome. Allows to specify single color for the drawings on all pages. By default the color is black.

Open file. If the flag is set, the PDF document will be opened when the export is complete.

Parasolid (*.xmt_txt, *.xmt_bin) Formats

When exporting into the Parasolid format, you need to specify the Parasolid **Version**.

> By default, the Parasolid version number is set to the one that T-FLEX CAD system is running (the maximum number).

Data organization. Specifies export type: **Assembly** or **Set of Bodies**.

Assembly. The model is exported with assembly hierarchy, if it exists.

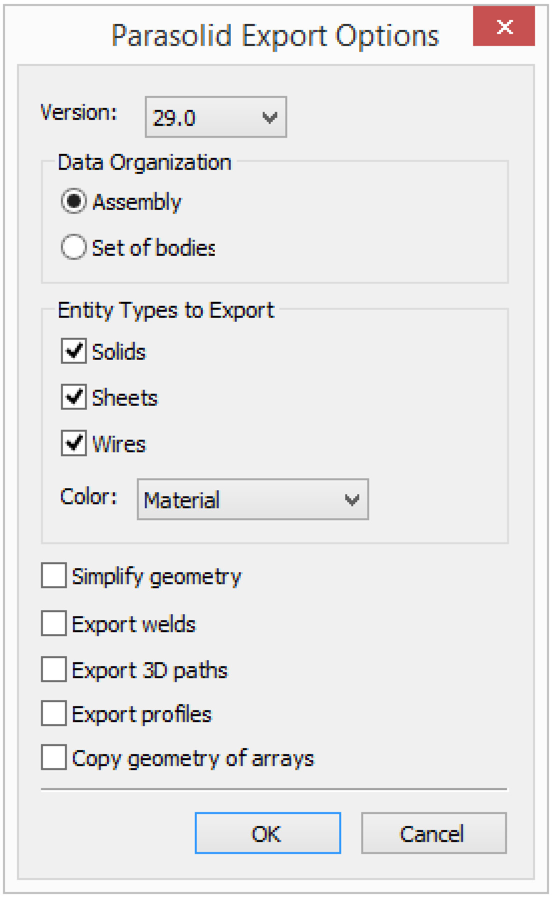
Set of Bodies. The model is exported without assembly hierarchy.

In the **Entity Types to Export** group, you can specify which objects will be exported:

- **Solids**;

- **Sheets**;
- **Wires**.

Color. Allows to select one of the following color sources: **Material** or **Shading.** In the first case the scattering color of material (is specified in the material parameters in T-FLEX CAD) is exported. Otherwise, shading is exported.



Simplify geometry. The option, if possible, removes the redundant topology of the exported model bodies. As a result the size of the output file is reduced. The option does not change the quality of the model.

For example, it can be "extra" ribs, breaking a cylindrical surface into segments. With this option, the export time and memory consumption are increased.

Export welds. Manages export of welded seams.

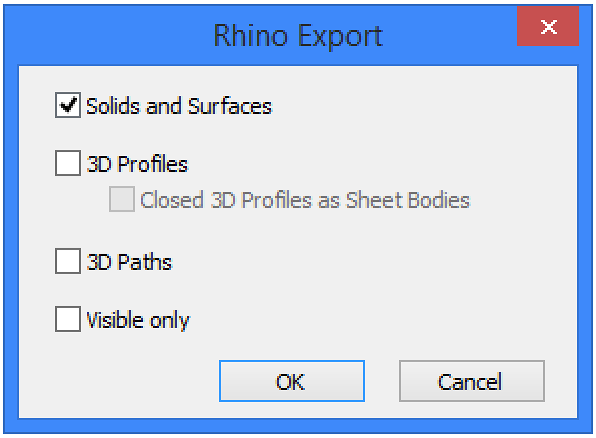
Export 3D paths. Manages export of 3D paths, routes.

Export profiles. Manages export of profiles.

Copy geometry of arrays. If the flag is set, copies of all of the elements of the array will be created. File size in this case increases.

Rhino Model (*.3dm)

Export options for this format specify types of T-FLEX CAD elements for exporting to Rhino model:



Solids and Surfaces. When this flag is set, all visible solid and sheet bodies will be exported.

3D Profiles. This flag enables 3D profiles for exporting as Rhino curves. If flag **Closed 3D Profiles as Sheet Bodies** is set, Rhino surfaces will be generated instead of curves-bounds for the closed 3D Profiles.

3D Paths. When this flag is set, 3D Paths will be exported as Rhino curves.

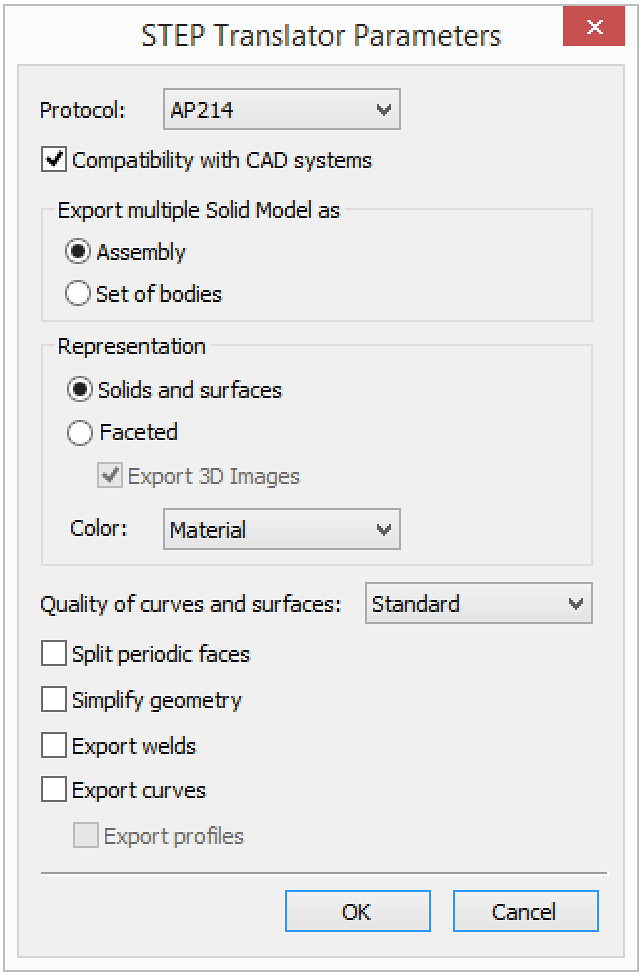
Visible only. This option controls 3D Profiles and 3D Paths to be exported. When this flag is active only visible objects will be exported. Solid and sheet bodies are not affected.

STEP (*.stp, *.step) Format

If STEP export format was selected, the **STEP translator parameters** dialog appears after entering the file name. Here you can specify the process options.

Protocol provides compatibility of STEP files with older versions of CAD systems. New protocol is enabled by default. Older protocol version AP203 doesn't support colors, so the **Color** option is unavailable for it.

Compatibility with CAD systems option provides possibility to read STEP file in other CAD systems.



Export multiple Solid Model as group:

Sets the mode of exporting a multiple-solid model:

Assembly. The model being exported is converted as an assembly with its hierarchy.

Set of Bodies. Geometry contained in the file being converted is exported as a plain set of bodies without creating an assembly structure. Each body contained in the model is exported separately.

Representation specifies type of bodies export: boundary representation based on geometry (**Solids and surfaces)** or mesh polygonal representation (**Faceted**).

Export 3D Images option is available for **Faceted** representation. If it is disabled, objects of 3D image type will not be exported into STEP file.

Color option specifies source of bodies color: basic material color or shading color.

Quality of curves and surfaces. Setting this option forces conversion of all non-analytical surfaces into B-spline surfaces. Accuracy of current geometry conversion into representation spline can be set using the drop-down list. The geometry is converted only if it is necessary. This might improve the resulting export in some systems, as, for example, in CATIA and SolidWorks, but increases the size of the resulting file. It is recommended to use standard quality by default.

Split periodic faces defines the method of converting periodic geometry present in the model.

By periodic geometry, we mean cylindrical, toroidal, spherical surfaces and their counterpart spline surfaces. When converting a model into the STEP format, such surfaces and curves can be broken into pieces for better export results.

Simplify geometry option removes excess topology from the model and reduces file size. It is disabled by default, as export time and memory consumption grow when it is used.

Export welds. When the option is enabled, welds are exported. Welds are transformed into bodies.

Export curves. When this parameter is set on, in addition to bodies of the model, the wire geometry will be exported as well.

Export profiles. When the flag is set, profiles are exported.

Curves and profiles are transformed into paths.

Convertation process is performed after pressing **[Ok]** button. The progress bar follows on the overall conversion process.

JT, ACIS, PRC Formats

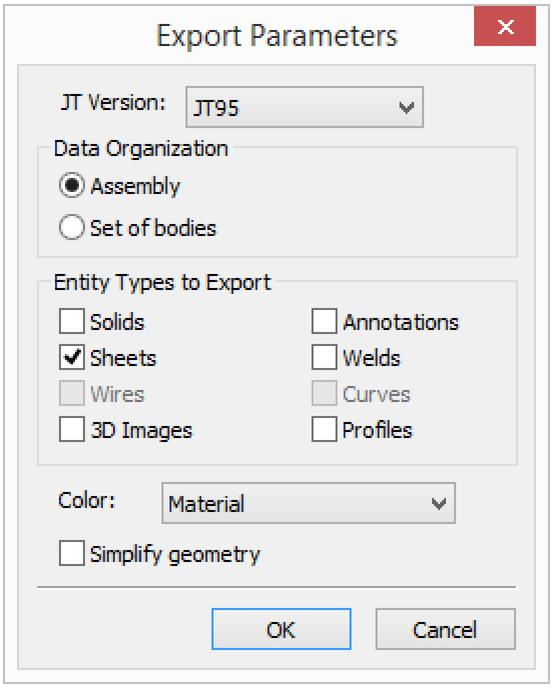
Similar dialogs are used for JT, ACIS and PRC formats.

For JT Format you can additionally specify **JT Version**.

Data organization. Specifyes the export type:

Assembly. The model is exported with assembly hierarchy, if it exists.

Set of Bodies. The model is exported without assembly hierarchy.



In the **Entity Types to Export** group you can specify objects to export: **Solids, Sheets, Wires**, **3D Images**, **Welds, Profiles**.

Option **Annotations** is available only for JT format. It allows to export dimensions and another annotations.

Wires and **Curves** export is available only for PRC format.

Color. Allows to select one of the following color sources: **Material** or **Shading.** In the first case the scattering color of material (is specified in the material parameters in T-FLEX CAD) is exported. Otherwise, shading is exported.

Simplify geometry. The option, if possible, removes the redundant topology of the exported model bodies. As a result the size of the output file is reduced. The option does not change the quality of the model.

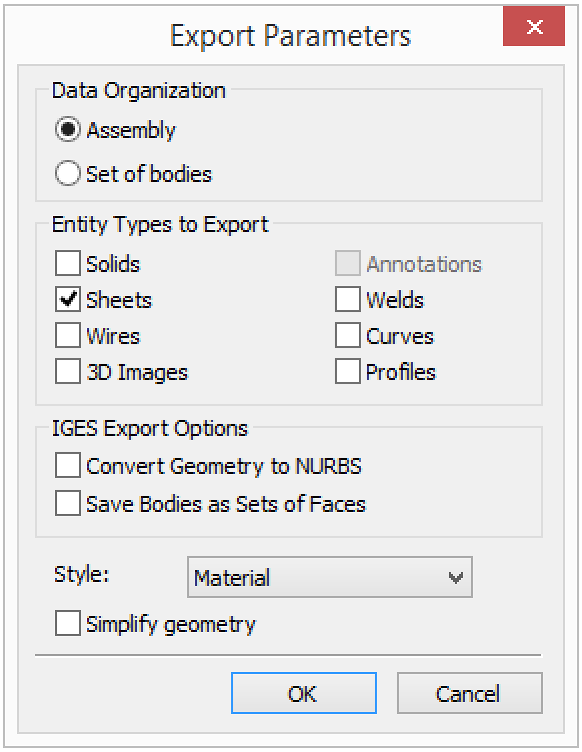
For example, it can be "extra" ribs, breaking a cylindrical surface into segments. With this option the export time and memory consumption are increased.

IGES (*.igs, *.iges) Formats

The IGES export dialog is similar to the dialog for JT, ACIS and PRC formats, but two additional parameters are available for the IGES format.

Using NURBS. Setting this flag allows use of rational splines (NURBS) when converting a model. If the flag is cleared, all exported geometry is described solely by polynomial splines.

Save Bodies as Sets of Faces. Instead of preserving a solid body, a set of its faces will be preserved, each face as a separate surface.



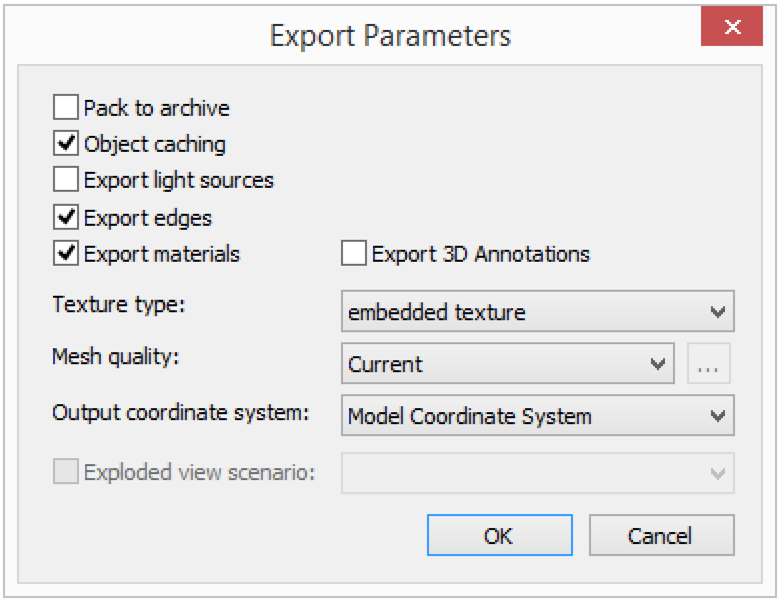
3D Mesh Formats Export Parameters

When models are exported to 3D mesh formats, their geometrical shape is converted into a set of triangular elements. You may control the accuracy of representation which determines the number of triangular elements and thus the export file size.

The same **Export Parameters** window is used for specifying export parameters for various 3D mesh formats. Depending on the format, the set of parameters may differ.

X3D, IV, VRML 2.0, TF3D Formats

You may define various export parameters for x3d, iv, vrml 2.0, tf3d formats (These parameters are also used in most of the formats described in the following sections):



Pack to archive - reduces the size of resulting file and stores all relevant data in one compressed file. For example, you can store main scene file and textures.

Object caching. If there are several identical elements in the document, and the flag is active, the elements will be saved in a single instance. Transformations are used to create all elements based on this instance. If the flag is disabled, a separate instance is created for each of the elements and the file size will be larger.

Export light sources - adds light sources of active view.

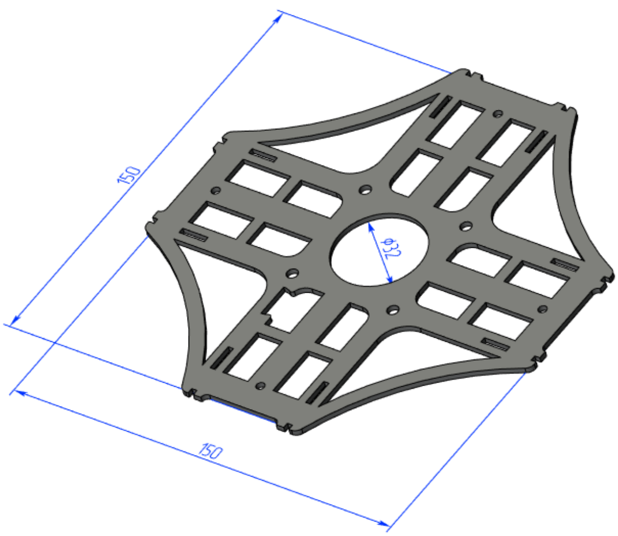
Export edges – adds model edges as separate graphical items.

T-FLEX CAD

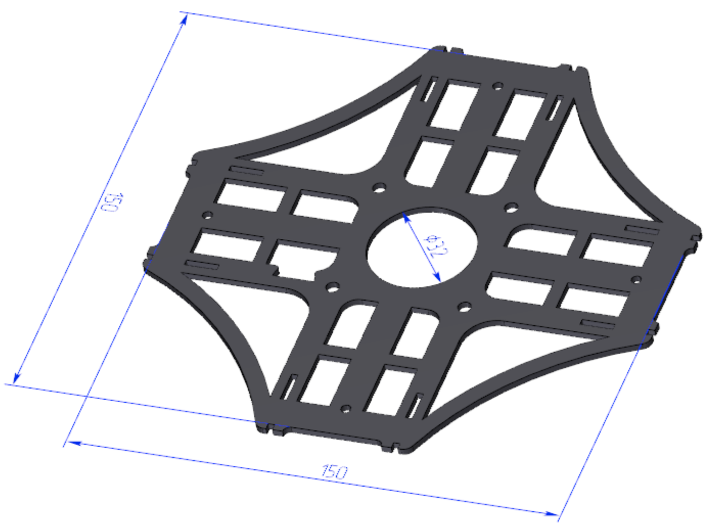
3D PDF

Export materials – when the flag is set, the materials will be exported. Otherwise, model colors will be exported.

Export 3D Annotations. Allows to export 3D annotations, such as dimensions and GD&T symbols.



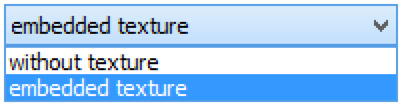
T-FLEX CAD



3D PDF

Texture usage:

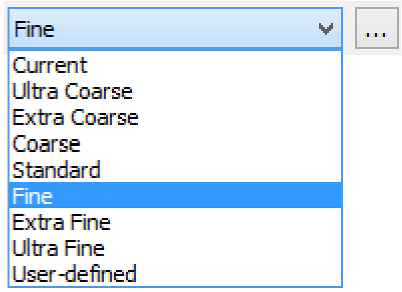
without texture - file is saved without data on textures;

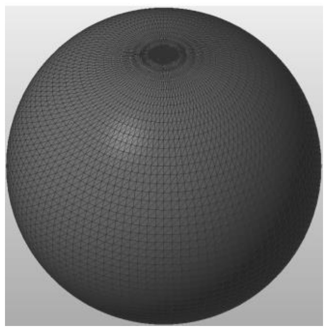
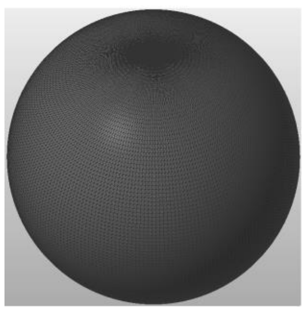


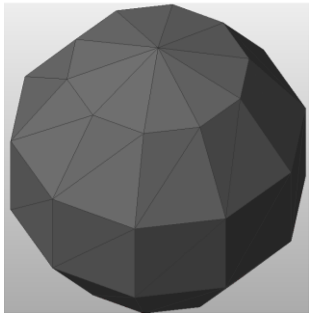
embedded texture – texture is stored inside the file.

external texture – adds only links to the texture files;

Mesh quality specifies image quality for the exported model. Higher quality produces bigger files.



Accuracy	Result	Size of STL file (ASCII format)
Default		3,27 MB
Increased accuracy		13,1 MB

Decreased accuracy		**21,7 KB**

To define mesh quality settings manually use option .

> More information about mesh quality settings can be found in "3D tab" section of "Drawing setup" chapter

Output coordinate system. Allows you to select the LCS that determines the initial orientation of the model when you open the exported file.

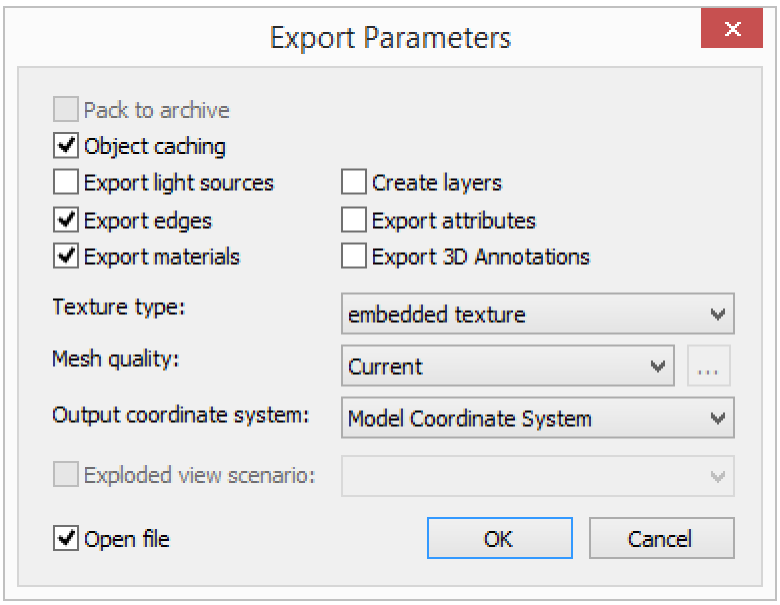
Exploded view scenario. Allows to select an exploded view scenario, which will be added to the exported file.

3D PDF, U3D Formats

These formats have the same options as described in the section "X3D, IV, VRML 2.0, TF3D Formats" and several additional options.

Create Layers. When it is enabled, all layers from T-FLEX CAD document will be created in the file.

Export attributes. Attributes may be set for 3D model objects. These attributes will be saved in the resulted file on export.

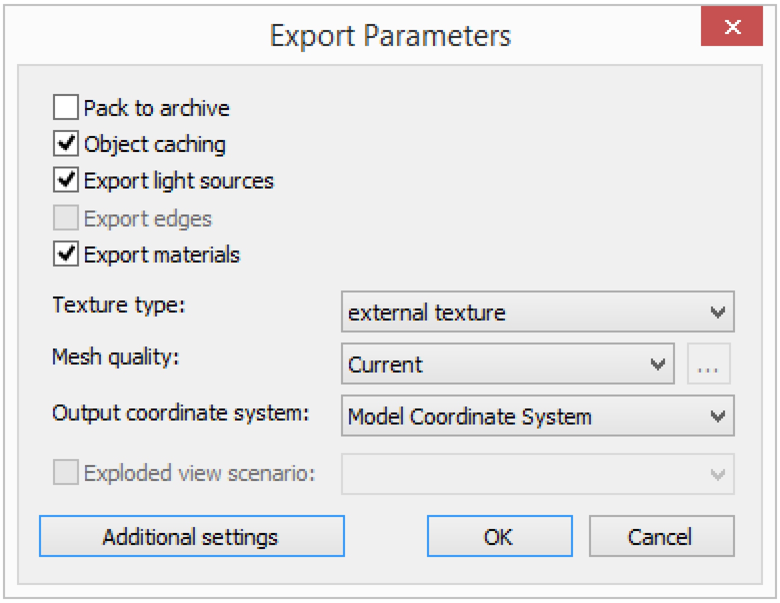


Threads can be exported into the 3D PDF format. Textures export should be enabled for this purpose.

Open file. Allows to open file in the program, associated with the format.

POV format

The **Export Parameters** window for the *.pov format has the same set of parameters plus button [Additional settings].

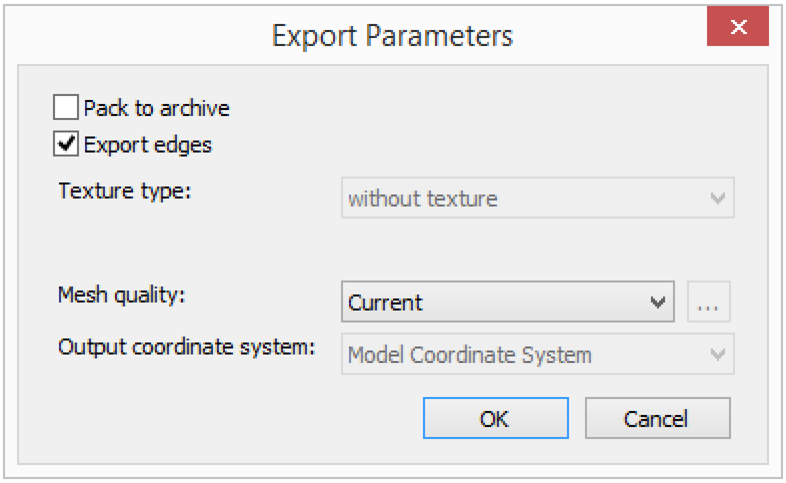


After pressing the button the **Pov-ray export** window appears.

More information about the window can be found in chapter “Photorealistic view”.

PLY, OBJ Formats

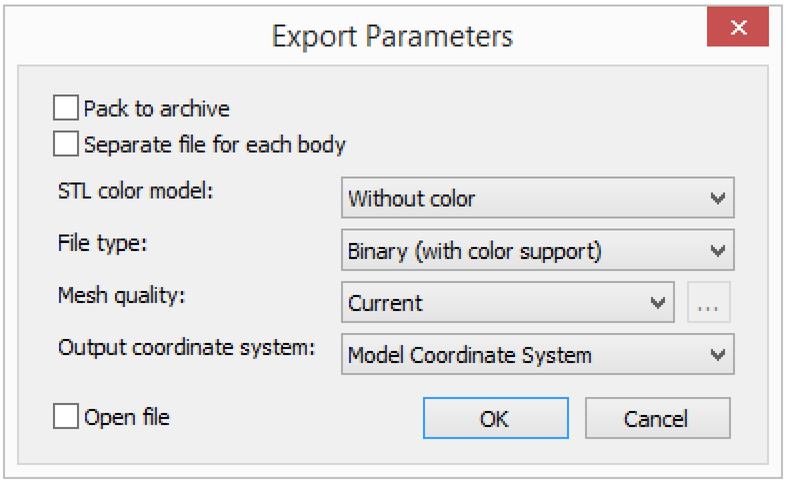
The following **Export Parameters** window has fewer parameters:



All options were already described above.

STL (Stereolitography) Format

There are some extra parameters in **Export Parameters** window specific to STL format:



Separate file for each body. Separate files will be created for each body in the assembly.

STL color model. You can select color from one of two standard formats**: VisCAM and SolidView format** or **Materialise Magics format** or select **Without color**.

File type:

Binary (with color support) - binary format with color support. Bodies are stored as a single large mesh.

ASCII (multibody) - text format without color support. The bodies are divided into individual meshes.

Open file. Allows to open file in the program associated with the format.

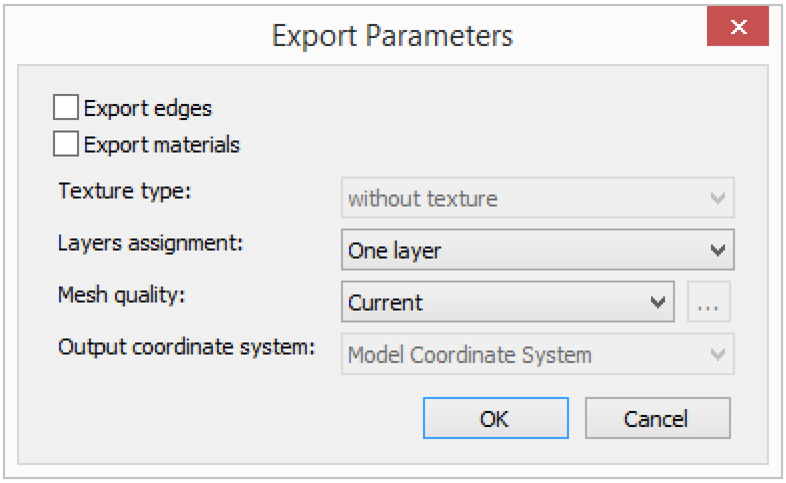
DXF 3D format

The only difference of **Export Parameters** window for DXF 3D format is a **Layers assignment** field. You can select one of the following variants:

One layer – all bodies will be placed on the same layer.

Layer for each body – bodies will be placed on different layers.

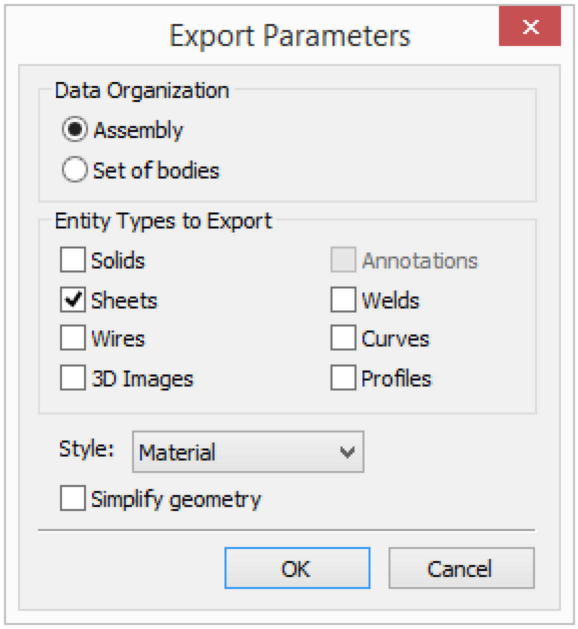
T-FLEX CAD layers – layers specified in T-FLEX will be used.



3MF Format

3MF is an XML-based mesh format created for use in additive production and 3D printing. The format includes additional information (for example, about the material and color of the model).

The export parameters dialog for the 3MF format is similar to the dialog described in the chapter "JT, ACIS, PRC Formats". The only difference is that the result of the export is a mesh geometry.



IFC Format

IFC is a file format that includes data of the construction industry.

Export to this format is started without selecting additional parameters.

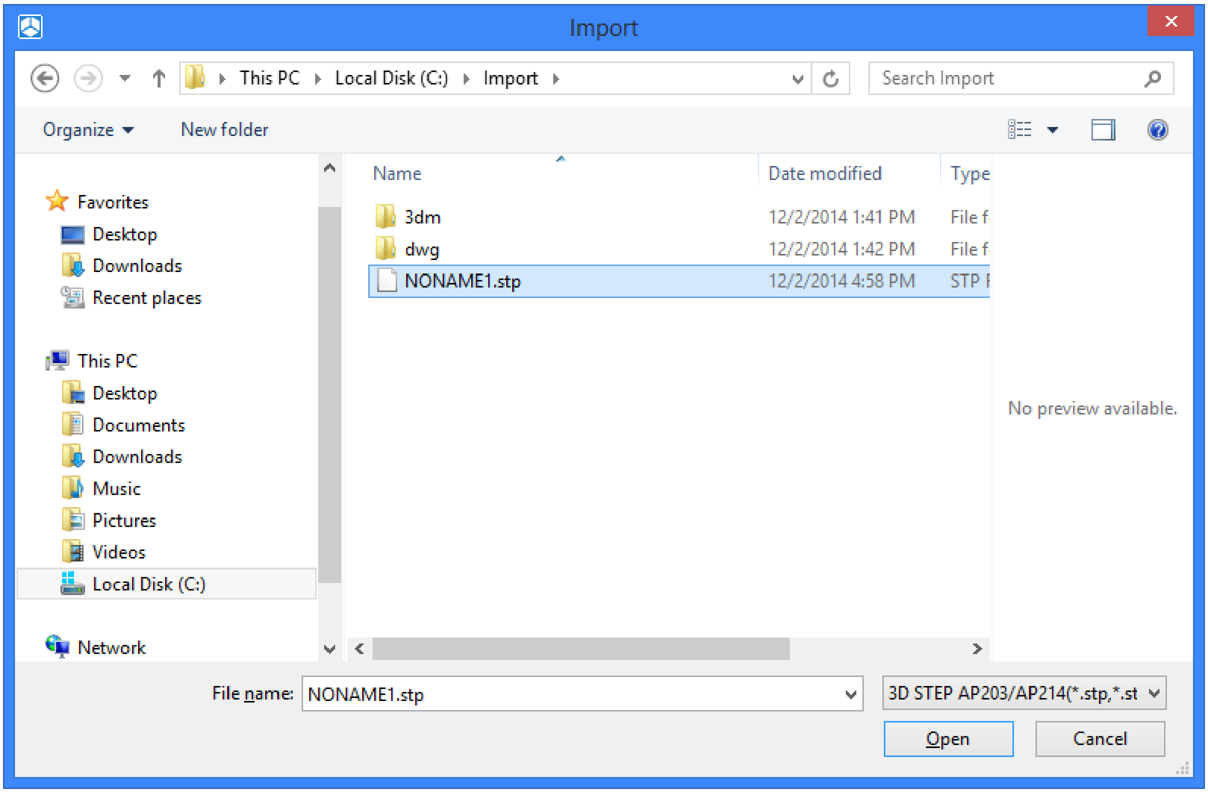
The result of the export is a mesh geometry.

Importing Documents

Importing documents in T-FLEX CAD from other systems is done by the command **IM: Import Drawing Or Model**:

Icon	Ribbon
	File → Import
Keyboard	Textual Menu
<IM> or <Ctrl><R>	File > Import

After calling the command, the standard File Open dialog appears on the screen. In this dialog specify the name of the file to import and its format. Then the dialog window for specifying additional import parameters appears (see below).

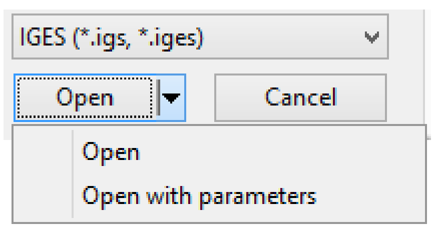


You can also import the file by moving it to the 3D scene of the opened file using drag'n'drop.

> To import files of various formats into the 3D version of the system, you can also use the command **3MO: Insert External Model.** More details on this are provided in the chapter "External Model" of the "Three-Dimensional Modeling User Manual".

In addition, you can open the imported model using the command **O: Open Model** or by dragging the model to the panel from Windows Explorer using drag'n'drop.

When you open a file using the command **O: Open Model**, you can select the **Open** or **Open with parameters** option.



If you select **Open with parameters**, a dialog similar to the import dialog for the specified format opens. In this way, you can open the model with the specified parameters.

> T-FLEX CAD can be set as the default program for formats that can be opened with the **O: Open Model** command.

AutoCAD Formats

When importing AutoCAD documents into T-FLEX CAD, you need to specify the following parameters:

Create new Document. This option directs where the result of the import shall be placed: in the current T-FLEX CAD or in a new document.

Convert Text to Windows coding. Setting this option converts all imported text to Windows encoding. This flag shall be used when importing an AutoCAD document of one of its earlier versions running under DOS.

Convert Blocks into Fragments. This option defines method of processing AutoCAD "block" entities. When the flag is on, blocks will be converted into T-FLEX CAD fragments. If this flag is cleared, blocks are converted directly into T-FLEX CAD entities.

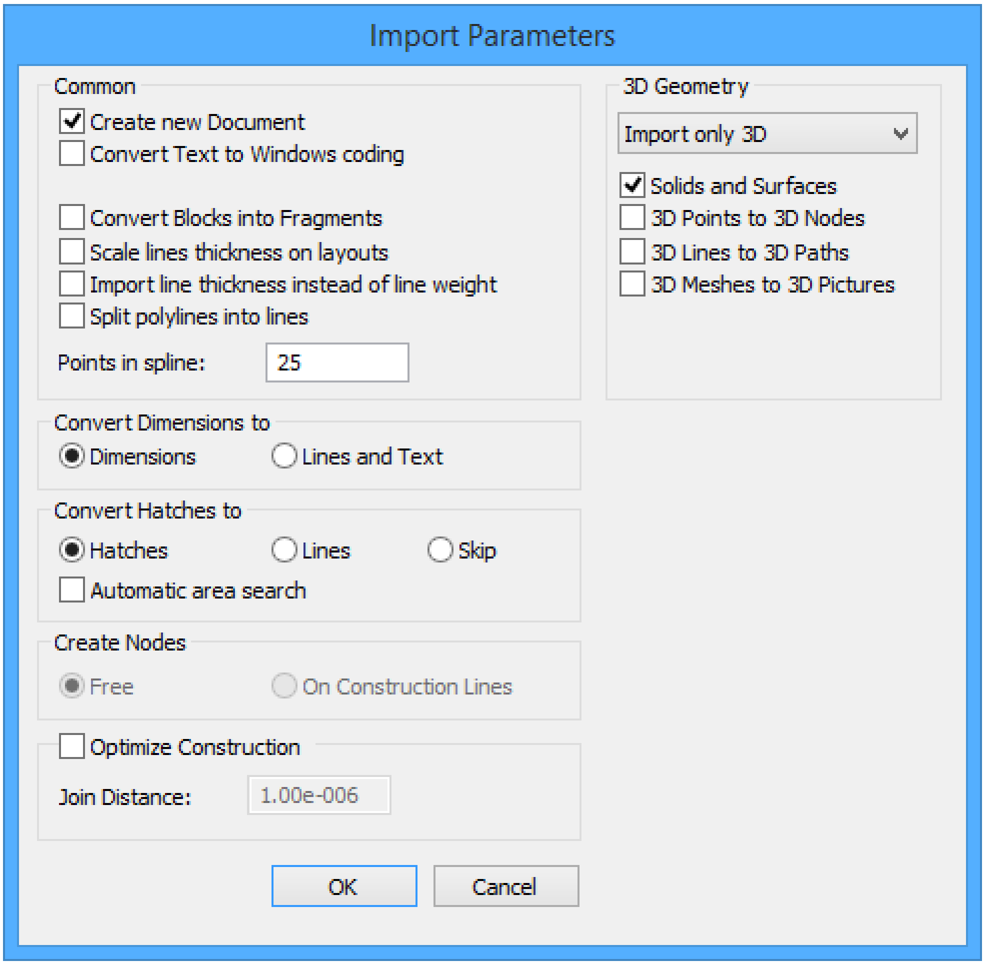
Converting blocks to T-FLEX CAD fragments may significantly slow down conversion if there are too many blocks and/or they have multilevel hierarchy.

Scale lines thickness on layouts. This parameter scales thickness of lines for drawings views and is helpful after importing from AutoCAD.

Import line thickness instead of line weight. The parameter defines the method of assigning thickness for imported lines.

Split polylines into lines. Imported polylines are divided, if possible, in several lines with simple geometry (segments, splines, arcs). On these lines you can create dimensions.

Points in spline. Specifies number of segments for imported splines representations. May be helpful in case of large number of imported splines. In this case reducing the number is recommended.



The next two groups of options (**Convert Dimensions to** and **Convert Hatches to**) define the method of converting AutoCAD elements (dimensions and hatches) into T-FLEX CAD elements. An additional flag for hatches **Automatic area search** enables the automatic hatch tracking (by the coordinates of points specified in AutoCAD at hatch creation - Pick points). Its use is recommended in the cases when hatch import fails in the normal mode (when the flag is cleared).

During the import, the points of the source AutoCAD drawing are converted (wherever possible) to free 2D nodes within T-FLEX CAD. A large number of such introduced nodes might interfere with further manipulations over the imported drawings. This can be avoided by setting the flag:

Optimize Construction. Upon setting this option, the nodes that are closer to each other than the specified **Join Distance**, will be merged in one node.

If the flag is cleared, then nodes coincidence is not tested. This helps speed up the import process, while could significantly clutter the drawing.

> Note that since the drawings in the DXF and DWG formats are not parametric, those remain non-parametric in T-FLEX.

3D Geometry. This parameter is used when the source AutoCAD file contains 3D solid geometry:

Ignore. When selecting this parameter, 3D geometry is ignored at conversion;

Ignore 3D as 2D. In this case, at conversion 3D bodies from the source AutoCAD model are projected onto 2D drawing, 3D geometry is absent in the resulting T-FLEX CAD file;

Import only 3D. 3D geometry of the source file is transformed into 3D geometry of the T-FLEX CAD.

- Solids and Surfaces – exact solid and surface geometry will be imported.
- 3D Lines to 3D Paths –3D AutoCAD lines will be imported as 3D paths.
- 3D Points to 3D Nodes - 3D AutoCAD points will be imported as 3D nodes.
- 3D Meshes to 3D Pictures – AutoCAD mesh models will be imported as 3D Pictures.

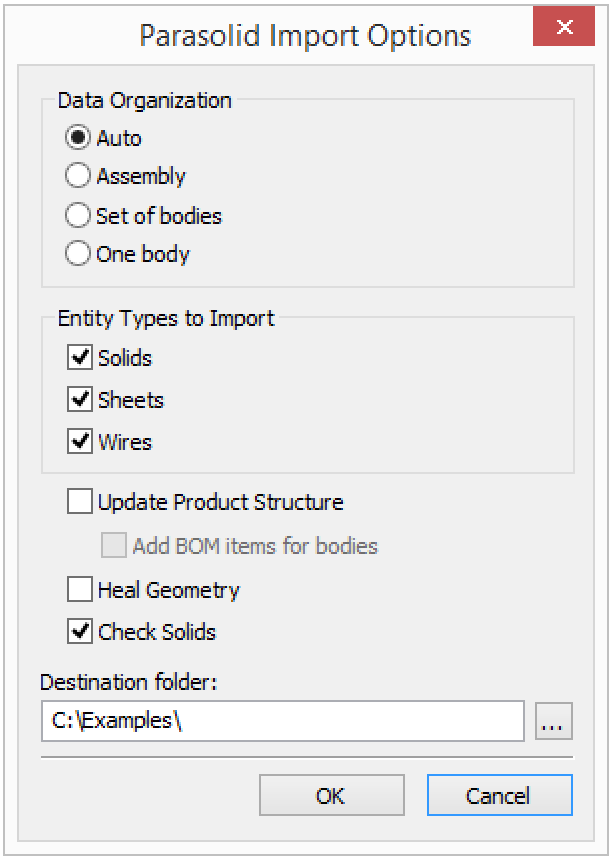
Just like when exporting, the old data exchange mechanism can be used when importing T-FLEX CAD version 8.0 and earlier data. For this, set the **Use Version 8 Algorithm** flag. The flag is available only for 32-bit version of CAD. Once this flag is set, the option "Automatic area search" becomes inaccessible, while the group of options activates that pertains to the old mechanism:

Create nodes: Free, **On Construction Lines**. This group of options allows selecting the method of creating nodes in an imported drawing: as free nodes or as intersection points of additionally created horizontal and vertical lines.

Parasolid Format

When importing Parasolid 3D models you should define the import mode: **Assembly** or **Set of Bodies** or **One body**.

> This mode is important only when importing multibody models or assemblies. If original file is presented as one solid, the result will be the same in all modes.



When importing in **Assembly** mode the model will be imported into multiple files retaining assembly structure. Parts and subassemblies will be stored in the newly created documents. The root assembly file will be opened after import completion.

When importing in the mode **Set of Bodies**, the model will be imported as a set of independent external models corresponding to separate parts of the assembly. The imported model is added to the current T-FLEX CAD document.

One body mode will insert the resulting single body in the current document. The result is similar to the result of using the **3MO: External Model** command.

In the **Entity Types to Import** group you can select the objects to be imported into the document: **Solids**, **Sheets** and **Wires**.

Update Product Structure. Option manages the creation of product structure based on data from file: Description (Product name) and Part No (Product ID). It should be used only if records for the product structure exist in the imported file.

> **Add BOM items for bodies**. If the option is active, records about bodies will be added to the product structure.

Heal Geometry. If there are problems with imported geometry the system will try to fix them.

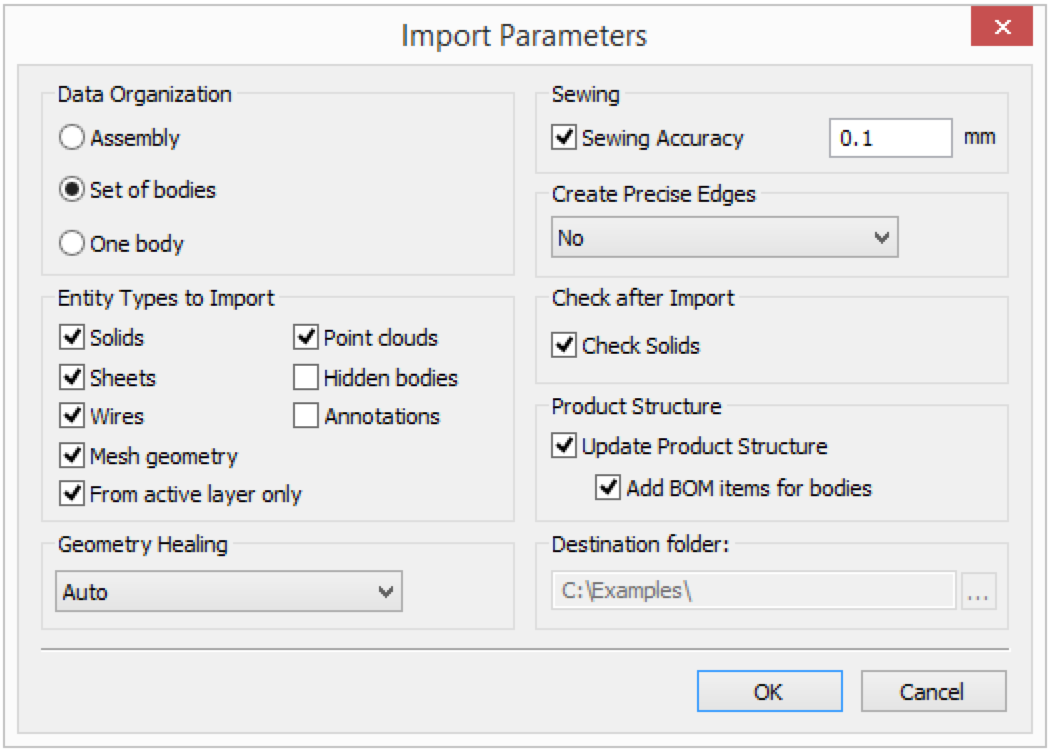
Check Solids. If the option is active, the import will check the correctness of the imported geometry. If errors are detected, messages will be displayed in the diagnostics window.

In **Destination folder** field you can specify folder for the created parts and assembly documents.

IGES, STEP, Solid Edge, SolidWorks, Rhino, Autodesk Inventor, ACIS, Siemens NX (Unigraphics), Creo/ProE, Catia V5, Catia V4, I-Deas, JT, PRC, 3DXML, CGR, U3D, VDA-FS Formats

Similar import parameters dialog is used for all of the formats.

In the **Assembly** mode, the model will be imported with the creation of the assembly structure. New documents are created on the disk for storing of the fragments and the assembly. After the import is completed, the created assembly file will be opened in the T-FLEX CAD window.



Set of Bodies. The model will be imported as a set of independent external models corresponding to the individual parts of the assembly. The imported model is added to the current T-FLEX CAD document.

If you select **One Body** the result of the import is single body. The result is similar to the result of using the **3MO: External Model** command.

In the **Entity Types to Import** group you can select the objects to be imported into the document: **Solids**, **Sheets**, **Wires, Mesh geometry, Annotations, Hidden bodies,** bodies **From active layer only**.

If **Point clouds** option is active, all points from the file are imported into T-FLEX CAD as 3D Nodes.

Geometry Healing. You can select one of the three variants: **Auto, Yes, No.**

If you select **Yes**, the system tries to correct the erroneous geometry in the imported model and receive a correct body. Potential errors may take place due to the presence of self-intersections or non- sewed surfaces in the original model. The resulting body after healing may differ from the original one.

> The **Geometry Healing** option does not guarantee obtaining absolutely correct data.

If you select **Auto,** the system decides whether it should try to heal it or not.

If you select **No**, the model is not healed, which accelerates the import process.

Sewing Accuracy. If the surfaces can form a solid body with a given accuracy, they will be sewn into a solid body. Otherwise, the surfaces will remain surfaces

Create Precise Edges. You can select: **Yes, No, Auto.** Enabling the option is recommended only for attempting to correct the geometry if errors occur in the model.

Check after import. If the option is active, the imported geometry will be checked and all found errors will be listed in the diagnostics window and all objects with errors will be marked in the model tree. Special warning will also be displayed at the top of the screen.

> To get a more detailed description of the problem, you can use the **QM: Check model** command.

> By default for options **Geometry healing**, **Create precise edges** and **Check solids** the optimal variants are set in terms of performance and result. It is recommended to change them only if there are problems in the imported model.

Update Product Structure. Option manages the creation of product structure based on data from file: Description (Product name) and Part No (Product ID). It should be used only if records for the product structure exist in the imported file.

Add BOM items for bodies. If the option is active, records about bodies will be added to the product structure.

Check Solids. If the option is active, the import will check the correctness of the imported geometry. If errors are detected, messages will be displayed in the diagnostics window.

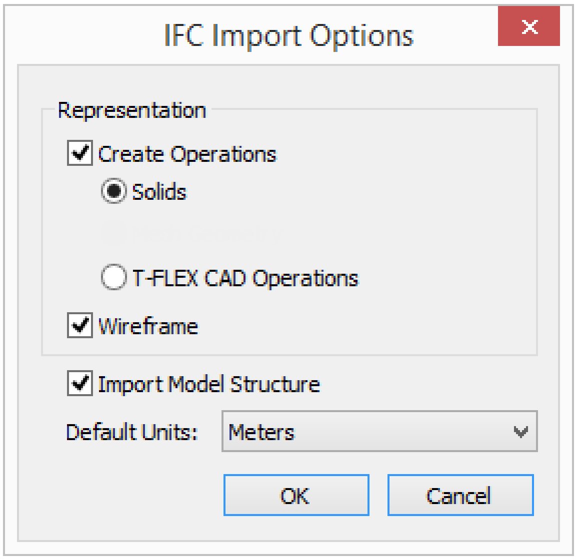
In **Destination folder** field you can specify folder for the created parts and assembly documents.

IFC format

The geometry from the file of IFC format can be imported as: **Solids** and **T-FLEX CAD Operations**. When you select **T-FLEX CAD Operations** the system tries to create T-FLEX CAD operations and the import will be slower.

Wireframe. Wireframe data will be imported.

Import model structure. Hierarchy of the source IFC file will be shown in the 3D Model tree.



Default units. Allows to specify units of the imported geometry in case that they were not specified in the source file.

ANNOTATIONS CREATION

Annotation is a set of notes, which overlaid T-FLEX CAD drawing. Annotation is saved in separate GRN file which content is displayed in annotated drawing. The original drawing itself is not changed. Creation and edition of annotations is performed using annotations editor.

Annotating allows the head of department, verifier or any other employee to add comments to the document without editing the original document. This functionality may be helpful for team development, especially when working with document management system.

There can be several annotations in one document. The annotations can be created by different people. A list of all document annotations is stored in GRI file. The file name is the same as for document.

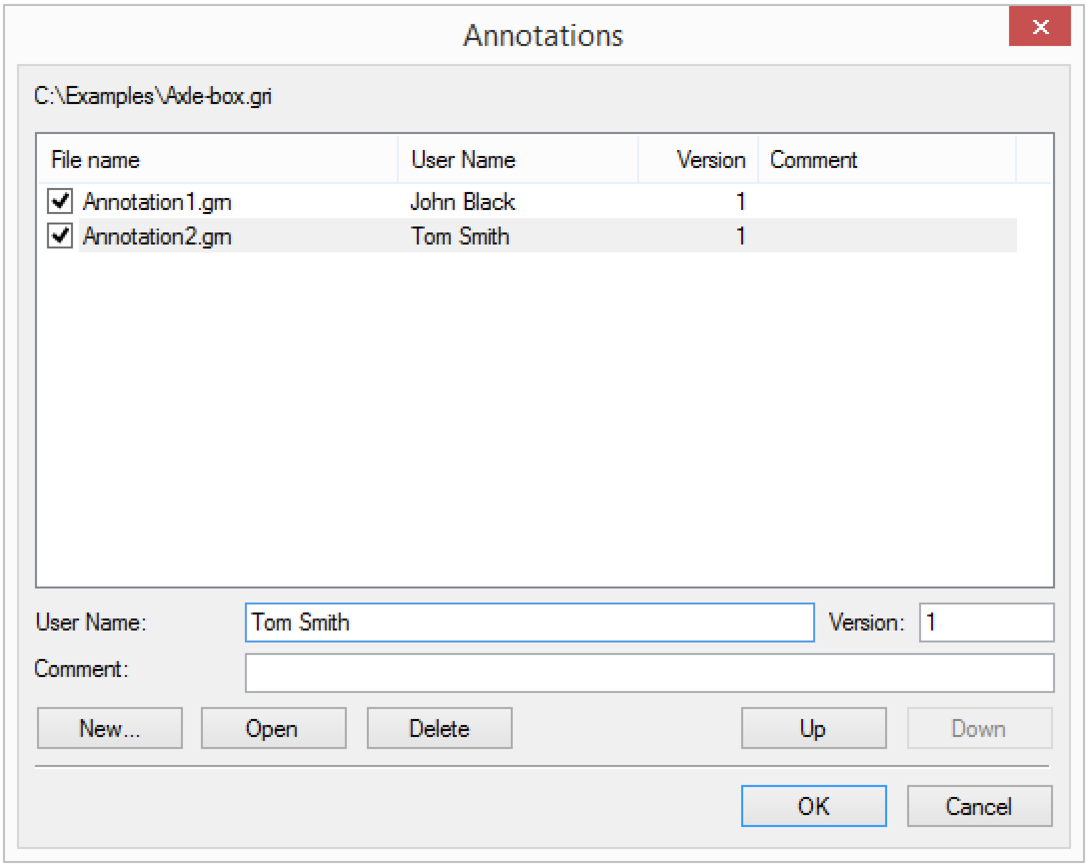
You should use **AT: Annotations** command to manage annotations. It shows a list of annotations for the current document, allows to show/hide annotations and to activate annotations editor.

ANNOTATIONS COMMAND

You should use AT: Annotations command to manage annotations. The command is called via:

Icon	Ribbon
	Tools → Tools → Annotations
Keyboard	Textual Menu
<AT>	Tools > Annotation

The command brings up a dialog that contains a list of annotations created for the current document.



Annotations with active flag are displayed in the drawing window. If you want to hide an annotation, you can disable the flag.

Priority of annotations on the drawing is determined by their order in the list. Use buttons [Up] and **[Down]** to change annotations order in the list. For example, you can display the text and dimensions from one annotation above the fill from another using the buttons.

Information about annotations is displayed in "User name", "Version" and "Comment" columns. The information can be edited.

To create a new annotation use **[New...]** button. The dialog window for the filename (By default "Annotation №.grn") and directory selection appears. Annotation files are stored in GRB file directory by default.

You can open the selected annotation for editing using **[Open]** button.

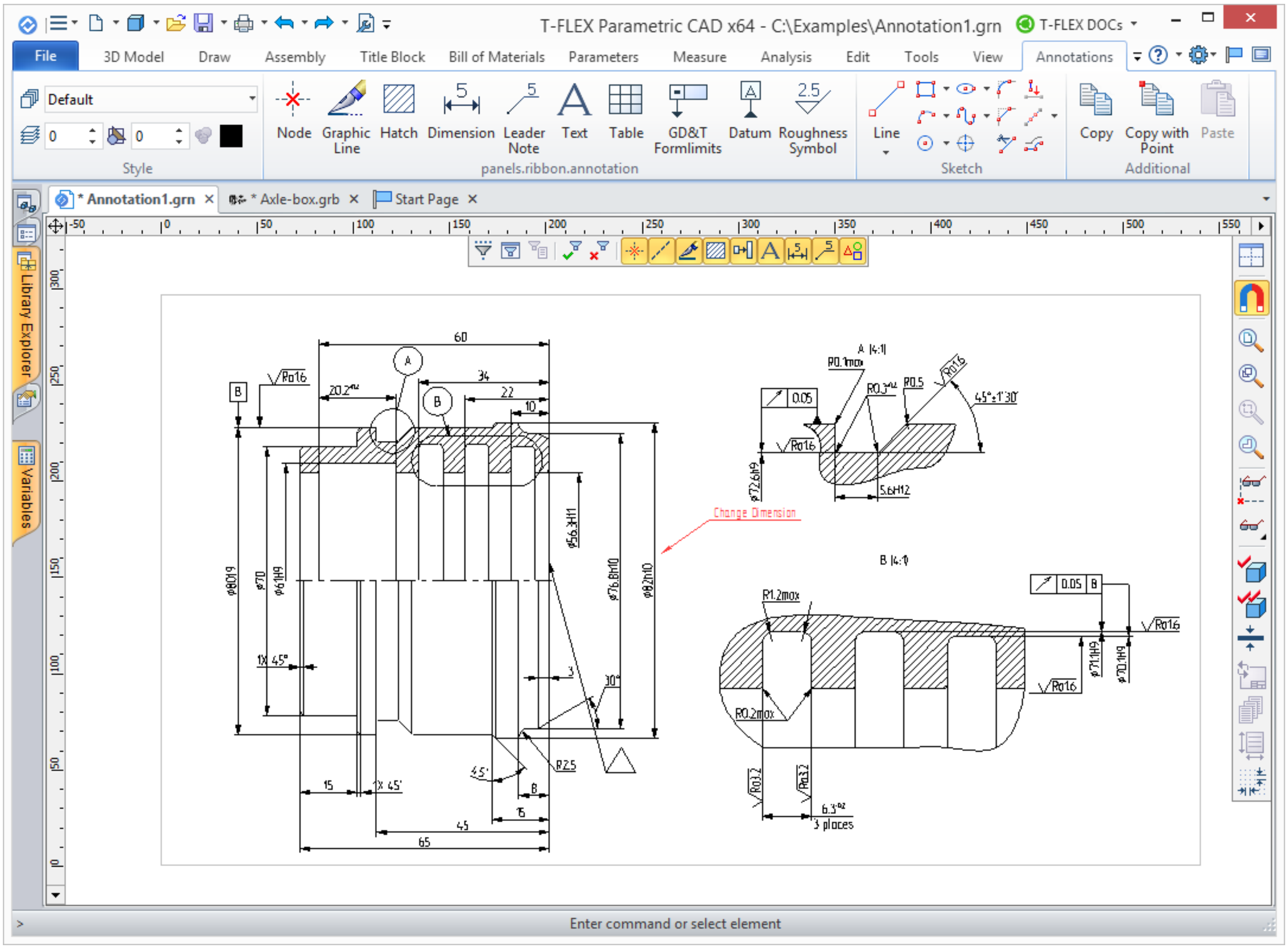
You can delete the selected annotation using **[Delete]** button.

Annotations Editing

When you open an annotation, system activates annotation edit mode. There are fewer elements available for creation/edition in the edit mode. Nodes, graphic lines, sketches, dimensions, text, hatch, leader notes and some other options are available for annotations.

All created elements are colored red by default.

Special tab with available commands appears after annotation opening.



Commands for system and drawing setup, toolbars and windows management are available.

Annotation creation is similar to creation of T-FLEX CAD fragment in the assembly mode. You can snap elements to nodes and graphic lines of the original document. If the drawing will be changed cause of model parameters changes, the snapped elements will also be changed.

After annotation creation it is necessary to save its file using **SA: Save Model** or **SL: Save All Modified Models** command.

Links. Managing Composite Documents

A T-FLEX CAD document can have links to other files: T-FLEX CAD documents (fragments), graphics files (pictures), database files, etc. Working with such composite document, one can face some difficulties, for example, when moving it to another computer.

To ease working with composite documents, T-FLEX CAD provides a mechanism of links management. A link is a T-FLEX CAD object containing the path to an external file (the target of the link). Links are used when creating fragments, pictures and other T-FLEX CAD elements for specifying the source of external data. The same link can be used by different elements; for example, several fragments based on one file will refer to one link.

The links management mechanism allows managing the way of storing the link targets. T-FLEX CAD allows storing the link targets either outside a T-FLEX CAD document as a conventional external file ("external link"), or directly within the file of the composite document ("internal link"). The internal storage of the link increases the size of the composite T-FLEX CAD document, but allows dealing with it as with one file. Links are managed by the commands located in the submenu **File** > **Assembly**.

The links management mechanism solves the problem of moving large assembly documents. By using it, you do not have to search for all fragments files that may be located in different folders, on different disks, in libraries, etc. All you need to do is to "zip" the assembly model into one file with a provision of future unzipping, and move it to another place in the file system or into a storage of a document management system (for example, T-FLEX DOCs).

Links Management

Links window serves for managing links in a composite document. To call the window use **Customize** > **Tool Windows** > **External Links** command.

> More information about links management can be found in "Main operation principles" chapter.

The command **UL: Update all reference documents (Fragments)** allows to reload the data from all external files contributing to the composite document:

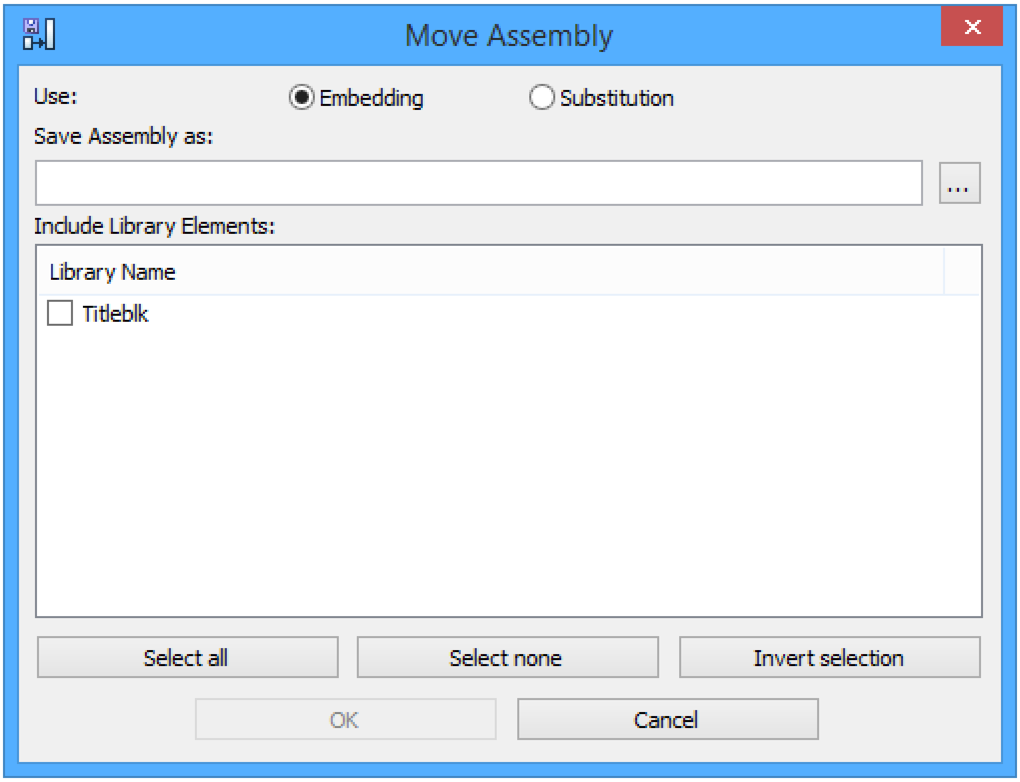
Icon	Ribbon
	Assembly → Component Links→ Update Links
Keyboard	Textual Menu
<UL>	File > Assembly > Update Links

MOVING ASSEMBLIES

The command **AM: Move Assembly** serves for moving a composite document to another location in the filesystem or for zipping the assembly into one file.

Icon	Ribbon
	Assembly → Component Links→ Move Assembly
Keyboard	Textual Menu
<AM>	File > Assembly > Move Assembly

After calling the command, the “Move Assembly” dialog box will be displayed. In it, you need to specify the new location and name of the moved document and the moving method:



Embedding. In the embedded moving, all linked files are stored within the assembly file. This method is convenient when you need to move a composite file to another computer, since it resolves in one common file.

Substitution. When moving by substitution, all linked files are gathered into one folder named after the file of the main document and located in the same folder.

In the lower pane of the **Include Library Elements** dialog, specify the list of libraries whose elements are used in the current document. The flags before the names of each library allowed specifying whether to move the library elements, to which there are links in the current document.

UPDATE ASSEMBLY

The command **UA: Update Assembly** starts converter that converts and saves all the fragment files that are included in the assembly.

Icon	Ribbon
	Assembly → Component Links→ Update Assembly
Keyboard	Textual Menu
<UA>	File > Assembly > Update Assembly

Creating Custom Lines and Hatches

T-FLEX CAD supports creation and use of user-defined custom graphic lines and hatches along with standard types. This chapter describes methods of creating and using those entities.

T-FLEX CAD standard distribution includes several examples of custom lines and hatches that can be used as a sample for creating your own ones.

Graphic Lines

To add your custom line type to standard types of graphic lines, all you need is to place the descriptor file of the new line type in the folder ...*Program**LinePatterns*.

The descriptor file of a custom line type – the line pattern – is a T-FLEX CAD drawing created to satisfy certain required rules. The file name coincides with the name of the line type. Upon restarting T-FLEX CAD, the new line type automatically appears at the end of the list of graphic line types.

Creating Line Pattern

Image of a line is created in general case from several elements described in the line pattern according to the special rules.

Description of a line element constitutes a drawing of the corresponding segment of a line executed with the use of construction lines, nodes, graphic lines, text and hatches. Description of each element of the line must be located on a separate page of the pattern. The sequence order of pages in the pattern's document is not important.

In general case the image of the line is created from the following elements: **Symbol**, **Space**, **Line**, **Start** and **End**.

Element "Symbol" – "main" element of the line. It can be repeated several times on the line. The number of repetitions is specified in the pattern of the line with the help of the variable CenterMaxCount. If this variable is not created in the pattern, the "Symbol" element is drawn only once on the line. The number of repetitions of the "Symbol" element can be decreased by the system if the number indicated in the pattern can not be fitted to the specified line length. The "Symbol" element is not drawn at all if the variable CenterMaxCount was assigned the value of "0" or if the line length is too small.

"Start" and *"End"* are the elements that determine the line endings. They are placed at the beginning and the end of the line being created without changes. If in the line pattern the "Start" or "End" elements are absent, it will be possible to specify standard endings for a line on a 2D drawing upon using the given type of a line.

"Space" and *"Line"* – auxiliary elements of the line that are repeated many times along its length, if required. When creating a line the system draws the "Start" and "End" elements at the line ends, then uniformly places along the line the required number of the elements "Symbol" (as many times as it can be fitted along the line length but no more than it is specified by the CenterMaxCount variable). The

remaining intervals between “Symbols” are filled with the “Space” and “Line” elements. Filling in is carried out in the following way: into each interval the maximum possible number of the “Space” elements is inserted. The spaces remaining between them “are covered” automatically by the “Line” elements scaled to the required proportion in such a way that the “Space” and “Line” elements alternate.

> Any elements can be absent in the line pattern except the “Space” element.

The required attribute of description of each element of the line is a pair of nodes named in a special way. The nodes determine the points of joining the given element of the line with the neighboring elements (characteristic points). The nodes names are precisely defined for each element of the line:

- *The “Symbol” element* – specified by nodes CenterStart, CenterEnd;
- *The “Space” element* – specified by nodes Start, End;
- *The “Line” element* – specified by nodes LineStart, LineEnd;
- *The “Line start” element* – specified by nodes TailStart, TailEnd;
- *The “Line end” element* – specified by nodes HeadStart, HeadEnd.

> Start and end are connected to the main part of the line by TailEnd and HeadEnd nodes, respectively.

When creating a pattern, one can use graphic elements of different color and different width. However, the color and width of a graphic line created from a custom type can be modified only if the whole pattern uses the same color and line width.

The line elements may be of a quite complex shape. When a hatch is created based on graphic lines of a custom type, the hatch contour will not follow the line shape by default; instead, it will be composed of line segments passing through the characteristic points of the line elements. To make the hatch contour exactly follow the line or assume other arbitrary shape, you would need to specify additional 2D paths with special names in the line pattern drawing defining the hatch contour route.

The paths are created separately for each line element. Each path must lie on the same page as the respective line element. The path must start and end at characteristic nodes of its element and be named as one of the following: “Polyline” – the path of the periodical part, “TailPolyline” – the start path, “HeadPolyline” – the end path, and “CenterPolyline” – the path of the center part.

Working with Custom-type Line

To start using a created line type, simply place the line pattern file in the folder ...*Program\LinePatterns*. Upon restarting T-FLEX CAD, the line types found in this folder will be automatically added to the lists of line types in 2D commands. To delete a custom line type, simply delete the pattern drawing file from ...*Program\LinePatterns* and restart T-FLEX CAD.

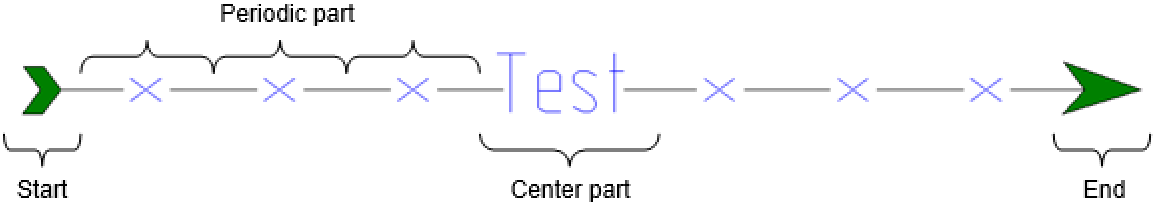
When using a custom line type in a document, the line pattern is saved with that document. Relation with the source file is lost at this point. Therefore, when porting a document file to a computer without the given line type, the image does not get lost.

When porting a document containing custom line types to a computer with existing line type pattern under the same name, the line image stays unchanged in the form of the image saved in the document.

To update a line image, you would need to redefine its type over again. To make an update, all you need is to bring up the parameters dialog box for one of the lines and confirm the existing value. New graphic lines of the same type that will be created in this document will take on the current pattern.

Example of Creating Custom Line

Let's review custom line creation on a simple example. We will create the line pattern shown on the figure. First of all, let's decide on the elements to be included in this line. The figure features: the periodical part – interlaced crosses and segments; center part entered just once – the text "Test"; start and end as special symbols.



To create the pattern, let's open a new 2D drawing (the command **FN: Create New Model**).

Description of each line element shall be placed on a separate page of the document. We will create four pages in the pattern document. For working convenience, each page can be renamed according to its purpose.

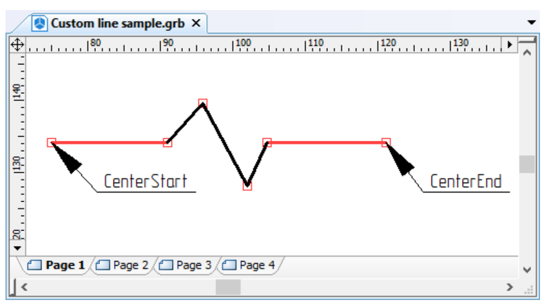
> To create or rename document pages, use the command **PG: Pages**.

When creating descriptor drawings of line types, one can use either use simple sketched lines, or graphic lines snapped to construction lines. For simplicity, we will use free-hand sketching.

We will begin creation of a line pattern from the main element – the periodic part. The descriptor of the periodic part will be the drawing depicting the cross with two strokes, on either side. The points at which this part connects to its neighboring line pieces are marked by the named nodes, "Start" and "End", as shown on the diagram.

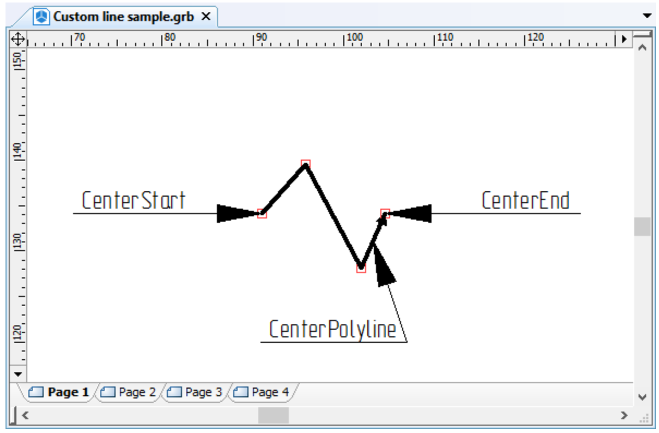
> A node name is defined by the command **EN: Edit Node**, the path name – by the command **EC: Edit Construction**.

The lines constituting the periodical part can be of various types, color and width, depending on the desired appearance of the custom line. In this example, the crosses on the line ought to be blue. Therefore, we will make the cross strokes blue, while the lateral strokes - black.

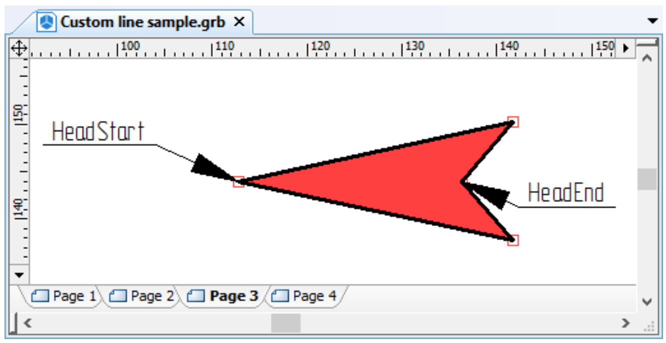


Next, we need to decide whether to prescribe the path for hatches. Suppose, this line may be used in the future for creating hatches, so that the hatch or fill lines are not allowed to overlap the cross strokes. For this purpose, we will create a named path, "Polyline", starting and ending in the named nodes "Start" and "End" and passing as shown on the figure.

After describing the periodical part, let's proceed to creating the next line element – the center part. To create it, we need to switch to another page of the pattern template.

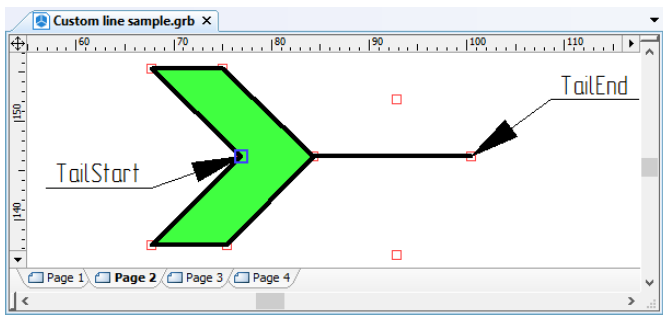


The pattern's center part will include the text "Test" with two strokes on either side. The text can be created by any text entity type (the command **TE: Create Text**). The start and end of the center part are defined by the named nodes "CenterStart" and "CenterEnd". Additionally, we will create a named path "CenterPolyline" for hatches based on this line, as shown on the figure.

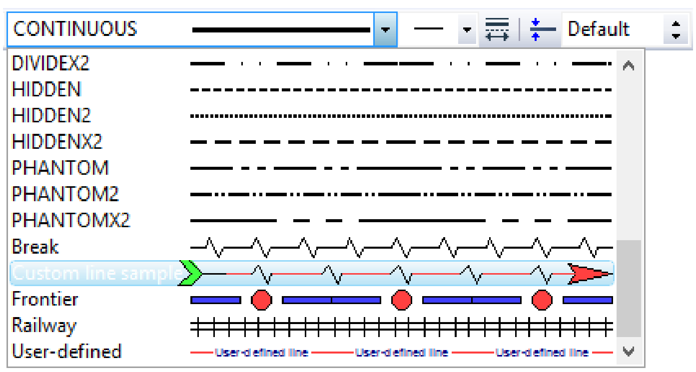


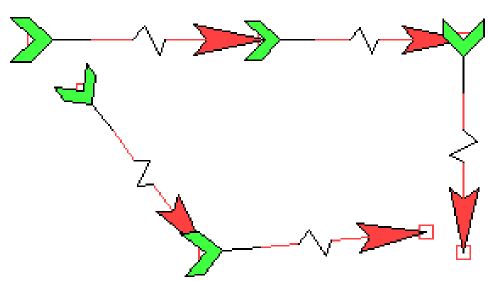
Similarly, we will create description of the tail and head of the line on separate pages, as shown on the figure. We will have to use a hatch to create the filling.

No additional paths for hatches will be created. In this case, the contour of a hatch bounded by this line type will follow the lines connecting the border nodes of these line elements.



What is left to do before using the created template is to save it in the folder ...*Program*\ *LinePatterns* under the name *"Custom line sample.grb"* and restart T-FLEX CAD. After restarting, the new line with this name will appear in the list of line types. It can be used just like a standard line type.





HATCHES

T-FLEX CAD allows creating custom hatch types along with custom graphic line types. To add a custom hatch type to the system, place the file with a hatch pattern in the folder …*Program**HatchPatterns*. Upon restarting T-FLEX CAD, the new type will appear in the list of hatches "by pattern". To remove a custom hatch type, simply delete its pattern file from …*Program**HatchPatterns* and restart T-FLEX CAD.

Creating Hatch Template

A pattern file is a T-FLEX CAD drawing complying with certain rules. It must contain the image of the hatch pattern composed of nodes, graphic lines, text and hatches, as well as construction elements. The start point of the hatch is defined by a special node named "Center". This node is mandatory in the hatch pattern.

To make a given pattern repeat multiple times when filling the hatch contour, additional named nodes must be defined in the pattern that would define the direction and step of the pattern repetition. The node defining the first hatch direction must be named «StepX»; the node defining the second direction – «StepY». The direction-defining nodes are optional. In the case either of them is not defined, the pattern will be drawn just once in the respective direction.

Working with Custom Hatches

When using a custom hatch type in a document, the hatch pattern is saved with that document. Relation with the source file is lost at this point. Therefore, when porting a document file to a computer without the given hatch type, the image does not get lost.

When porting a document containing custom hatch types to a computer with existing hatch type pattern under the same name, the hatch image stays unchanged in the form of the image saved in the document. To update a hatch image, you would need to redefine its type over again. To make an update, all you need is to bring up the parameters dialog box for one of the hatches and confirm the existing value. New hatches of the same type that will be created in this document will take on the current pattern.

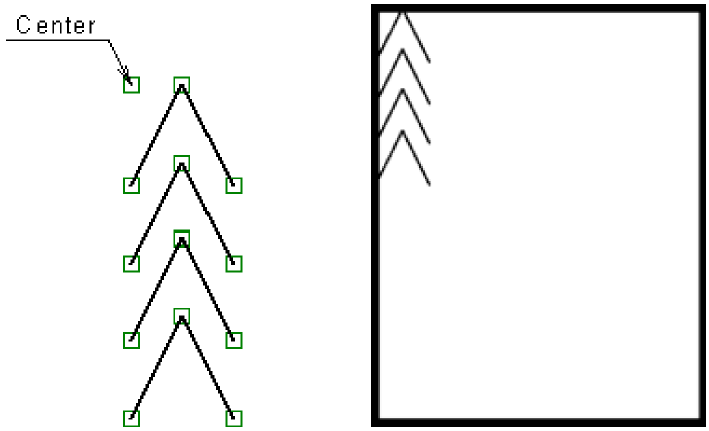
Examples of Creating Simple Hatches

To create a custom hatch pattern, let's open a new document (the command **FN: Create New Model**).

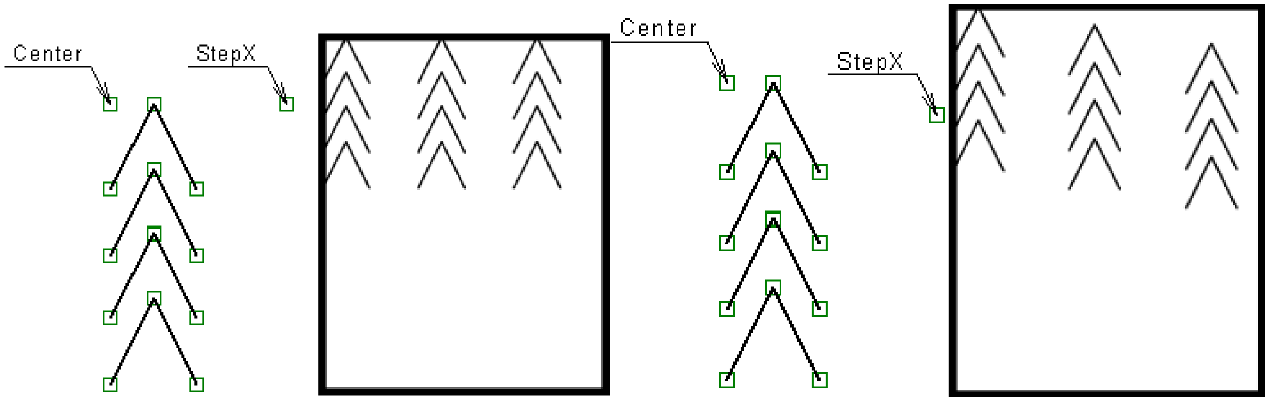
In the opened document, create the image of a hatch pattern as shown on the figure. Make sure to put an additional node on the drawing, named "Center", which must be present on a hatch pattern.

The drawing created in this way can already be identified by the system as a custom hatch template. All that needs to be done is saving the file in the folder …*Program**HatchPatterns* (for example, under the name *"Custom hatch sample.grb"*) and restart T-FLEX CAD.

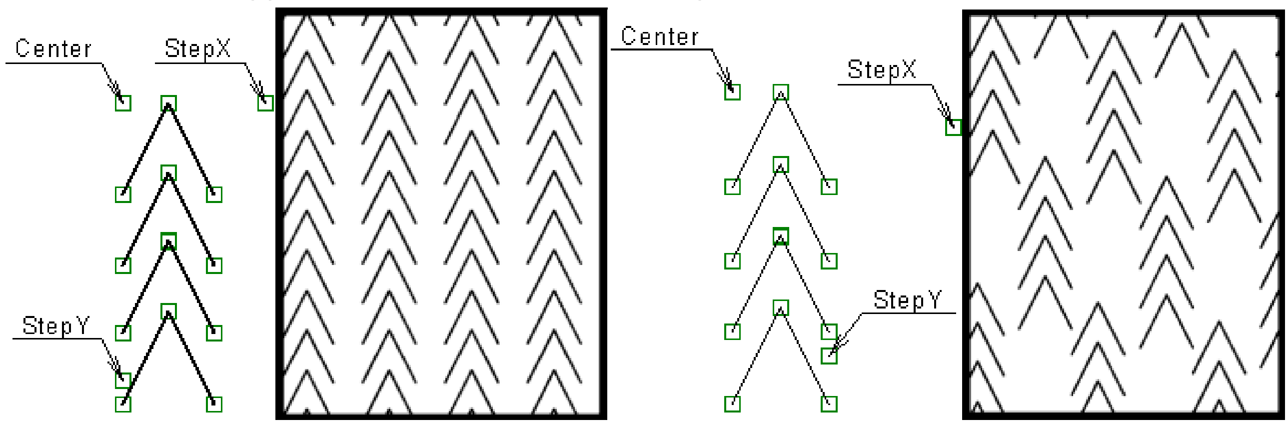
However, since no direction nodes are defined in the pattern, an actual hatch created from this pattern will contain just a single instance of the hatch.



To make the sample defined in the hatch pattern repeat in one direction, you would need to create the respective named node in the pattern, as, for example, “StepX”. The position and step of the node will define the step and repetition direction of the pattern sample.

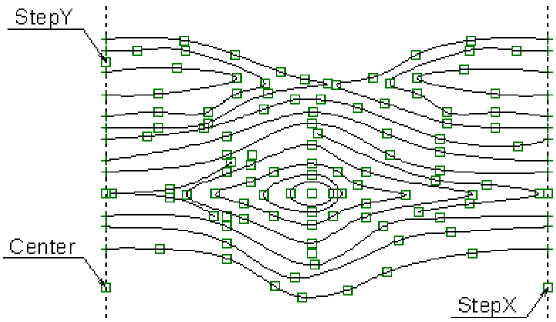


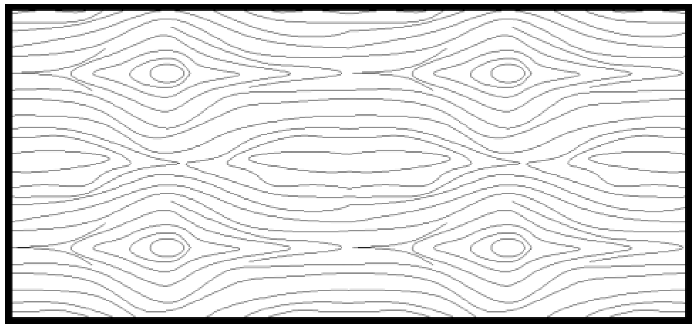
Two named nodes – “StepX” and “StepY” – would allow defining repetition of the pattern sample in both directions. By varying the position of the direction nodes with respect to the “Center” node, one can obtain various hatch appearances all based on the same pattern.



Example of Creating «Woody» Hatch

The custom hatch creation mechanism allows creating even more complex hatches, for example, a "woody" hatch. To create this type of a hatch, a pattern was created as shown on the figure. An appropriately arranged placement of direction nodes yields a hatch imitating wood texture.





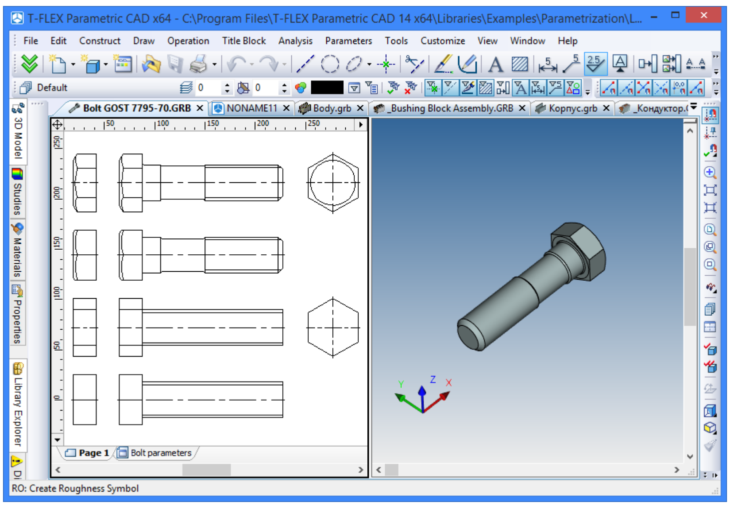
CREATING LIBRARIES OF PARAMETRIC ELEMENTS

T-FLEX CAD offers a rich library of standard elements included in the application installation package. The libraries of standard elements speed up the design process and allow the designer to concentrate on the actual development, rather than on drawing bolts and nuts alike.

Besides, T-FLEX system provides tools for the user to define one's own standard elements, that support wide range of productivity improvements. Designers often use families of elements whose members are mostly identical, often with the only difference in the dimension values, having to spend time just to draw such an element. This issue can be resolved by creating a standard parametric element in T-FLEX CAD system.

A big advantage of T-FLEX CAD compared to other systems is the provision for the user to create own standard parametric elements. This does not require knowledge of any programming language. The user can create any objects, from drawings and 3D models to dialog boxes for fragment insertion. T-FLEX CAD provides special commands for this purpose. Another positive feature is that any library element is an ordinary T-FLEX parametric drawing.

CREATING PARAMETRIC LIBRARY ELEMENTS



Parametric library element creation can be divided into several steps:

Creating databases (if necessary).

Creating variables, including those relying on databases.

Creating a parametric drawing and/or 3D model.

Creating fixing vectors and fixing LCS;

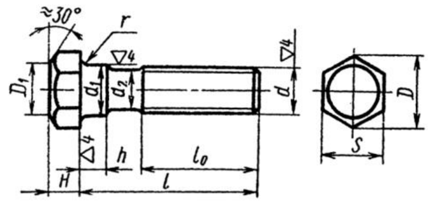
Creating 2D and/or 3D connectors;

Creating a dialog box of fragment's parameters.

Adding the element to the library.

Some of these steps can be skipped. For example, you may not need a set of values from a database, or you're not creating a 3D model, 2D or 3D connectors.

Let's review each step by an example. Consider a bolt GOST 7795-70.



The file for this example can be found in the library "Examples" in the folder "T-FLEX Parametric CAD 15 \Libraries\Examples\Parametrization\Library of parametric parts creation\ Bolt GOST 7795-70".

The parameters of this bolt should update from the database depending on the input diameter and length. The bolt has several implementations. Both the drawing (the three views: front, left and top) and the 3D model should adjust to each implementation. The bolt will be used as a fragment in other drawings.

Creating a Database

At the step, you need to create an internal database to pick the values from.

First, you need to decide, which will be the driving parameter. In our case, the candidate parameter is the bolt thread diameter. Therefore, the bolt diameter is entered in the first column, followed by all the rest of the parameters (except the length). The names of the database columns (and the database itself) should be made descriptive, so that later, when creating variables, you could easily remember which column relates to which parameter. However, too long column names are not recommended either. A good strategy is to name the columns according to engineering standard notations adopted in your industry. In our example, the thread diameter is named d, the bolt head size - S, etc.

b

№	d	d1	h	S	H	H1	D	r	d3	d4	l2
1	6	6	3	10	4.2	4.2	10.9	0.25	1.6	2	2
2	8	8	4	12	5.5	5.5	13.1	0.4	2	2.5	2.8
3	10	10	5	14	7	7	15.3	0.4	2.5	2.5	3.5
4	12	12	6	17	8	8	18.7	0.6	3.2	3.2	4
5	16	16	8	22	10	10	24.3	0.6	15	4	5
6	20	20	10	27	13	13	29.9	0.8	4	4	6.5
7	24	24	12	32	15	15	35	0.8	5	4	7.5
8	30	30	15	41	19	19	45.2	1	6.3	4	9.5
9	36	36	18	50	23	23	55.4	1	6.3	4	11.5
10	42	42	21	60	26	26	66.4	1.2	8	5	13
11	48	48	24	70	30	30	77.7	1.6	8	5	15

In our example, the input bolt length must be rounded to the nearest standard length. We should also account for the fact that the set of standard lengths is different for each diameter. For example, the bolt diameter of 6 mm corresponds with the length of 30 mm, while the bolts 12 mm in diameter have minimum length of 45 mm. Besides, depending on the selected bolt length, two additional parameters are defined: l1 and l0. We recommend creating separate databases of lengths for each diameter. The suggested database names are l6, l8, etc. The letter l, as in "length", refers to the databases of lengths, while the number stands for the respective diameter.

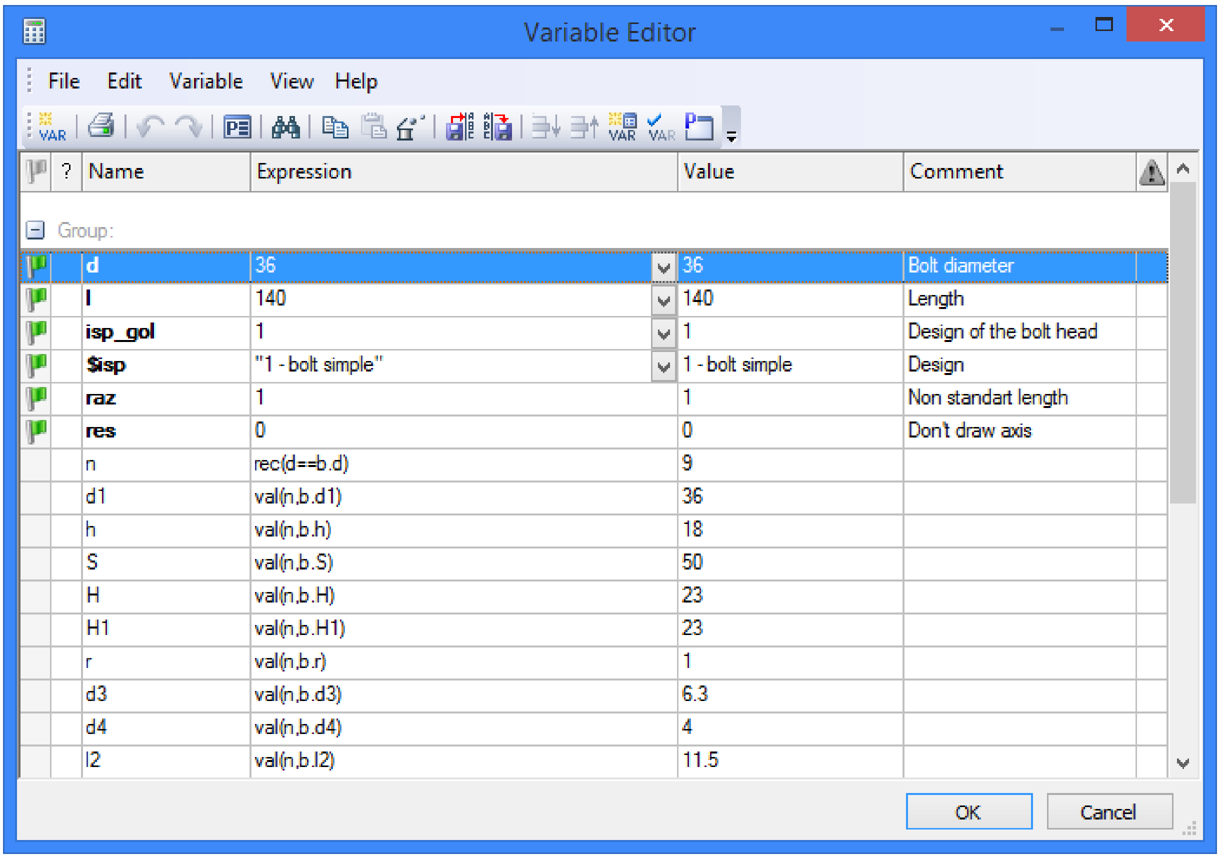
Once the databases are created, proceed to the next step.

l8

№	l	l1	l0
1	35	31	22
2	40	36	22
3	45	41	22
4	50	46	22
5	55	51	22
6	60	56	22
7	65	61	22
8	70	66	22
9	75	71	22
10	80	76	22
11	90	86	22

Creating Variables Relying on Databases

At this step, we need to create variables that would be used for building the parametric drawing and the 3D model. We recommend using descriptive names for the variables.



Name	Expression	Value	Comment
d	36	36	Bolt diameter
l	140	140	Length
isp_gol	1	1	Design of the bolt head
$isp	"1 - bolt simple"	1 - bolt simple	Design
raz	1	1	Non standart length
res	0	0	Don't draw axis
n	rec(d==b.d)	9	
d1	val(n,b.d1)	36	
h	val(n,b.h)	18	
S	val(n,b.S)	50	
H	val(n,b.H)	23	
H1	val(n,b.H1)	23	
r	val(n,b.r)	1	
d3	val(n,b.d3)	6.3	
d4	val(n,b.d4)	4	
l2	val(n,b.l2)	11.5	

First, define the driving variables for the rest of dimensions, whose values need to be defined when assembling the part. In this example, those are the diameter, the length and the implementation (see below). If a variable has a set of standard values, it is helpful to create the predefined list of values for convenient and quick input. The list of values can be created based on an existing field in the database. Upon creating a variable, it is a good practice to put a description in the comment field, so that another person can easily decide on what data to input, when working with this document.

Next, we need to create variables that keep track of the database record number from which the values are taken. Such variable value can be read by calling the function **rec** or **frec**. For the functions description, refer to the chapter "Databases". In this example, we need two such variables: one (the variable **n**) for accessing the values dependent on the bolt diameter from the respective database, and the other (the variable **nl**) for maintaining the value of the length. The challenge here is that the second variable should be assigned an appropriate record number across different databases (depending on the diameter). Besides, one should keep in mind, that the input length might not be always correct, as the designer of the part could enter a nonstandard bolt length. Therefore, the value of the variable **nl** can be described by the expression: *d==6 ? frec(6.l,l) : (d==8 ? frec(l8.l,l) : d==...),* that means, if the variable d (diameter) is equal to 6, then do the search for the record number in the database number 6, otherwise, if the variable d is equal to 8 - then do the search for the record number in the database named l8, and so on over all databases.

Next, create the rest of the variables. Their values are accessed by calling the function **val** based on the value of the variable that keeps the record number of the database (those are - **n** and **nl**). The value of the bolt length (**ll**) will be defined by the expression: *d==6 ? val(nl,l6) : (d==8 ? val(l8.l,l) :d==...)*. The meaning of this expression is similar to that of the variable **nl** described above.

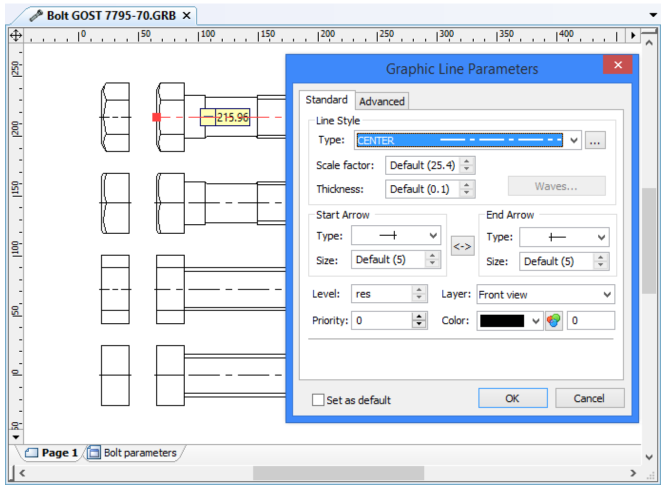
The bolt has several implementations. The drawing should adjust, depending on the implementation. This can be fulfilled by setting, when necessary, the level of certain drawing elements below the displayable threshold. We need to create special variables for this purpose (separate for each implementation). By default, the displayable elements have level from 0 to 127. If the level of an element is below zero, it is not displayed. Consider, for example, the variable **imp25** that defines the level of the elements visible only in the implementations number two and five. It is equal to 0 when the implementation (defined by the string variable *$imp*) is equal to "2 - with pin hole" or "5 **-** with pin hole and groove", and is equal to -10 in all other cases: *$imp=="2 - with pin hole"*||*$imp=="5 - with pin hole and groove"?0:-10*.

Special functions (WARN and ERROR) are provided in T-FLEX CAD for prompting the user about errors when variables take certain inadmissible values, and request inputting different value. For example, if a non-specified bolt length is encountered, you can force the message: "The input bolt length is not standard". This is done by calling the function: *WARN("The input bolt length is not standard")*.

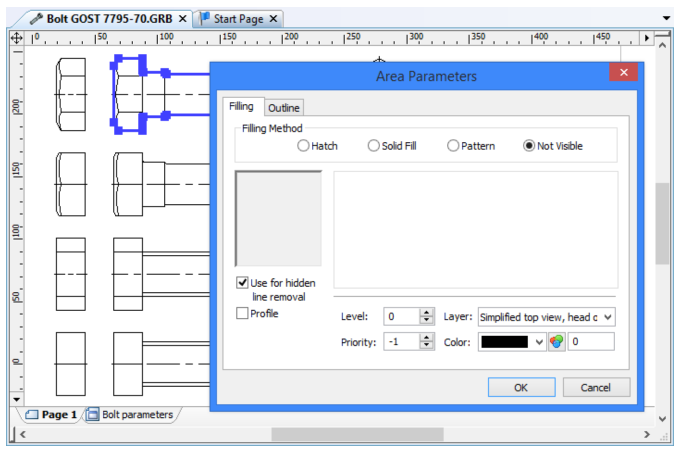
Once the necessary variables are created, you can proceed with creating the drawing and the 3D model.

Creating Parametric Drawing and 3D Model

When using a library element is a fragment, it is often necessary to insert just a portion of the drawing. Anticipating that, you can predefine several new layers. Use descriptive layer names to make it easy to understand which portion of the drawing belongs to which layer. In our example, we need to create three new layers. Let's name the layers according to their purpose: front view, left view and top view. During creation, all elements can be placed on any layer, for example, on the layer “Main”. Once the drawing is ready, simply select the desired portions by box and modify the layer of those elements as desired.



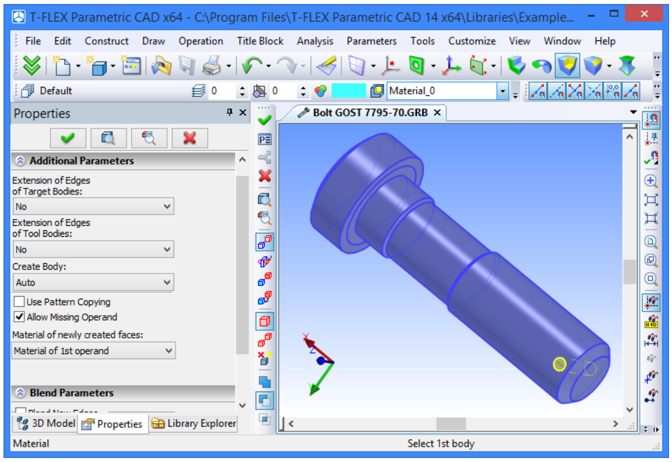
Some elements of the bolt (for example, the pin hole) may or may not be displayed, depending on the implementation. At the previous step, we created special variables for this purpose. The variable **imp25** is equal to 0 whenever the implementation is equal to “2 - with pin hole” or “5 - with pin hole and groove”. In any other implementation, the variable is equal to -10. If an element is to be displayed only in the implementation 2 or 5, its level must be defined by the variable **imp25**.



To make sure that in the assembly drawing the bolt is properly displayed in the bolted joints, to each view of the drawing let us add hidden hatches with the activated option of suppressing the hidden lines.

Upon completing the 2D drawing, proceed with the 3D model. Suppose, your 3D model is required to undergo changes depending on some parameter. It is not solely the model dimensions that could change. Some features may appear or disappear in the model.

In our example, such a feature is the pin hole. The easiest way to achieve this is by suppressing a certain operation. This can be done using the same variables as when defining the level of the respective features on the drawing. What exactly can be done in this case? The pin hole is created as follows. First, a rotation was made, creating a cylinder. Next, a Boolean subtraction was done of the cylinder from the bolt stem. The rotation operation can be suppressed as follows. Among the “Rotation” operation properties, assign the parameter “Suppress” to the variable **imp25**. The rotation operation will be suppressed when the variable is not equal zero (that is, when the implementation is not equal 2. To avoid a system error when creating a Boolean with an empty member, set the flag “Allow missing operand” among the Boolean parameters.



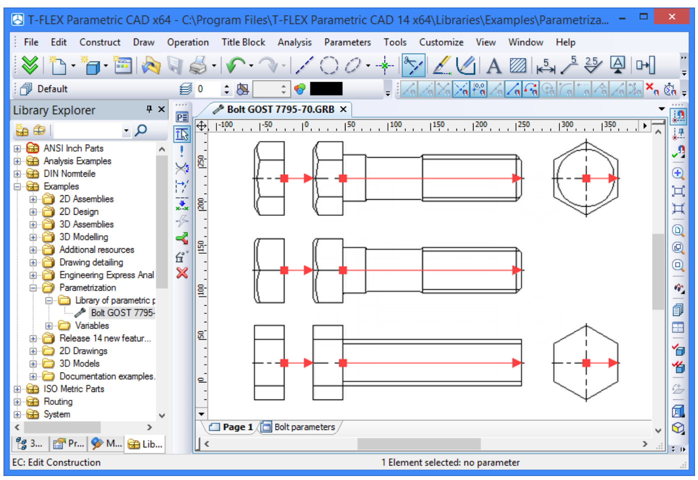
On the resulting body of the bolt the user should put a thread. To achieve this, let us use the command **“3AT: Create thread”**.

The drawing and the 3D model of the future library element are completed. Next, let us create fixing vectors and the special coordinate system for fixing the created document in the assembly drawing. They will be used for inserting the given document as a fragment into the assembly.

Creating Fixing Vectors and Coordinate System for Fixing Fragment in Assembly

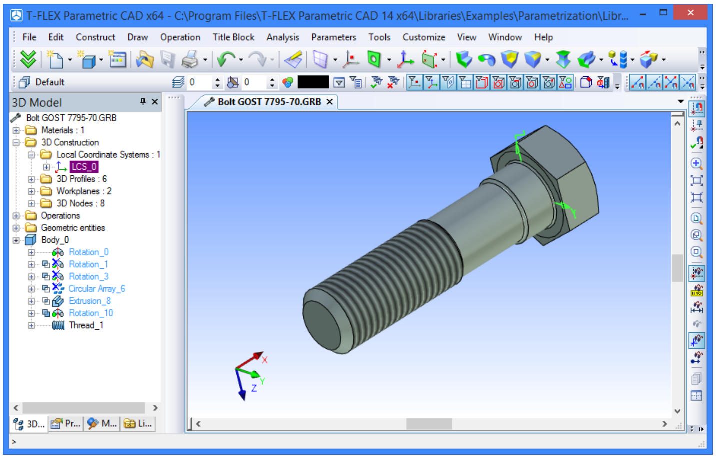
In our case, the fixing vector will be used when the library element is inserted as a 2D fragment into the assembly drawing. To use it as a 3D fragment, it is necessary to create a special local coordinate system (3D analog of the fixing vector) which will be used for fixing the fragment in the 3D assembly model.

For our drawing of the bolt let us create several fixing vectors, each for inserting a certain layer. Among the created vector properties, the flag is checked “Draw only marked”, and the layer is marked that should be displayed upon the insertion. The same vector names as the layer names are used for clarity.



On the 3D model of the bolt, let us create LCS for fixing a 3D fragment.

The position of the fixing vectors and the coordinate system is determined by the main surface (the contact surface between the bolt and other parts). This is the surface of the bolt head base.



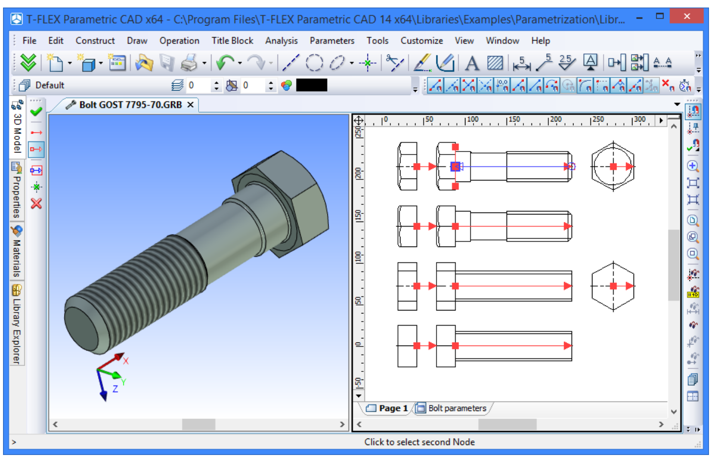
When inserting a fragment, the contents of the fragment folder are displayed in the left portion of the coming up dialog box. If the flag is set “Show icons”, then the drawing slides (icons) are displayed next to the file names. We recommend creating slides for the drawings, to insure easy selection for assembling.

The slide can be created using the command **Tools > Special Data > Preview**. Upon entering the command, select the item “Create Raster Preview as Windows Bitmap (BMP)”, press the graphic button **[OK]** and select the portion of the drawing that you want to put on the slide.

After this, proceed to the next step.

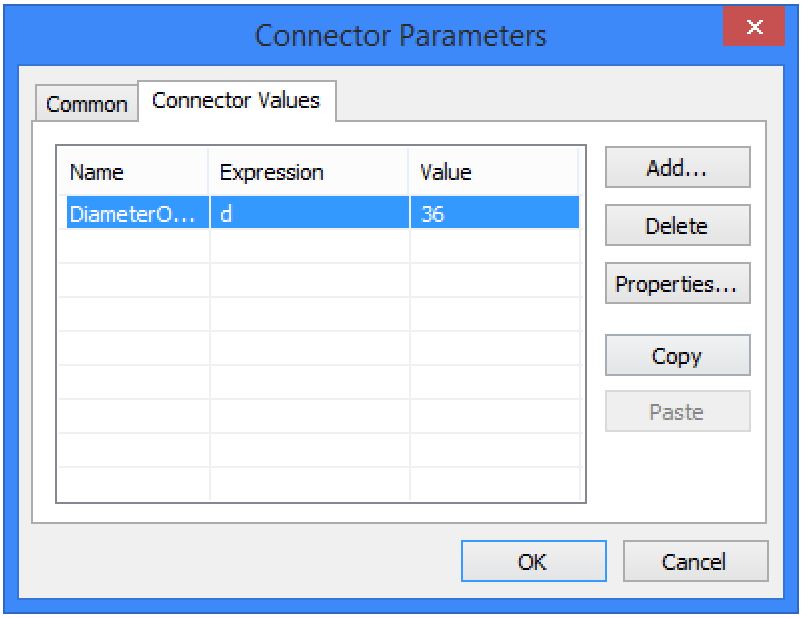
Creating Connectors

In the assembly drawings, other fragments, for example, a fragment-nut, will be attached to our library bolt element. Let us create conditions for a quick “connection” of other fragments: let us add connectors to created drawing and 3D model, and also assign connector’s values to external bolt variables (for a quick specification of the values of the fragment’s variables upon inserting a fragment into the assembly).



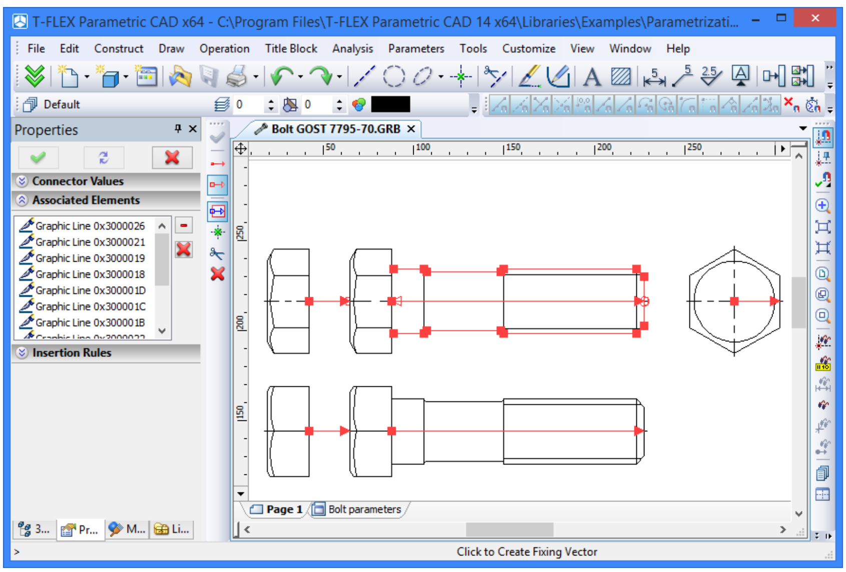
For the drawing, let us create four 2D connectors – for two front views (standard and simplified) and for two top views. For creating a connector, the command **FV: Construct Fixing Vector** can be used.

Let us specify the named value *"DiameterOuterMetricThread"* for all 2D connectors. This named value will pass the magnitude of the bolt's diameter to the fragment (nut) being attached. As an expression for the named values of all connectors, indicate the variable *"d"*.



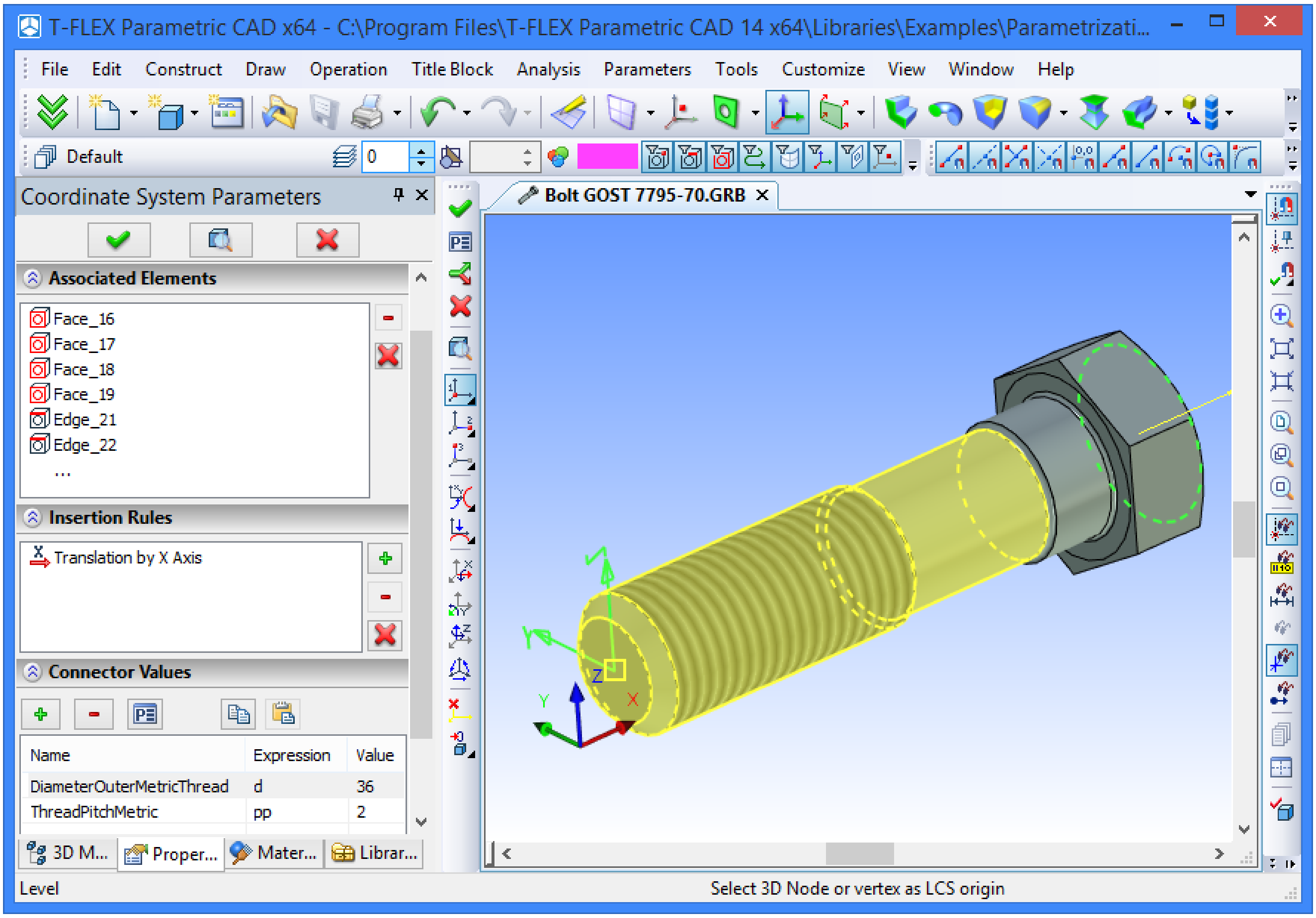
> Note that for the successful use of the connectors, upon creating the fragment of the nut it is required to indicate the connector's named value *"Diameter-OuterMetricThread"* in the properties of the variable controlling the nut's diameter.

For created connectors, let us indicate graphic lines, as associated elements, on the corresponding views of the bolt – this will simplify fixing to the given connector.

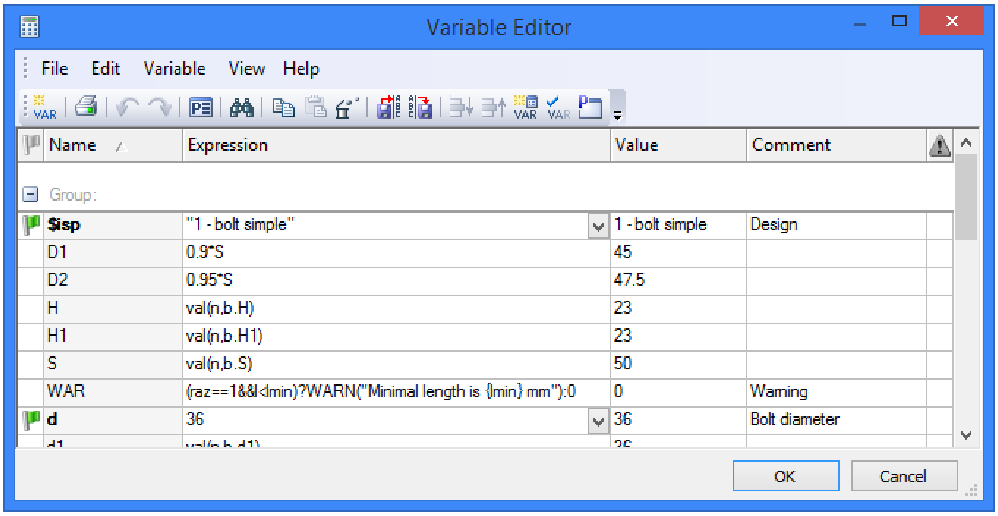


Also, for 2D connectors it is required to specify «insertion rules». When fixing a nut to the bolt's connector, it is usually required to specify additional translation along the X-axis of the connector.

By the same principle, let us create 3D connector on the 3D model. For the 3D connector let us create two named values: *"DiameterOuterMetricThread"* (bolt diameter) and *"ThreadPitchMetric"* (pitch of the bolt's thread).

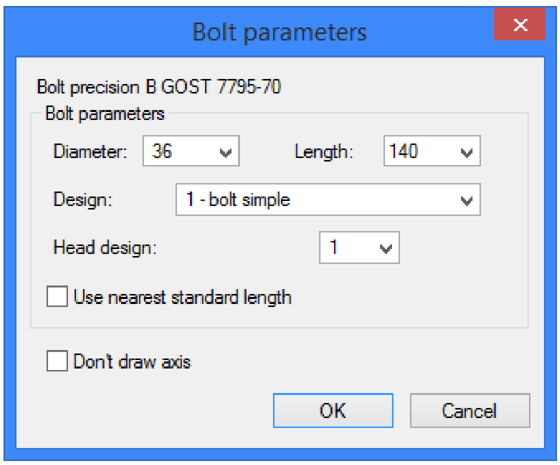


Having created connectors, we prepared a basis for convenient fixing of other fragments of the assembly to our fragment. Next, let us return to the variables editor of our model and specify the following value of the connector: *"DiameterInnerMetric"* for the variable *"d"*. The same named value was specified for the connectors entering the library of openings. As a result, when the fragment of the bolt is inserted into the assembly by means of fixing to an opening's connector, the actual bolt diameter will be defined automatically.



Creating Dialog Box

At this step, we will compose the dialog box that will be used for inserting the current drawing as a fragment. Each time when assembling a drawing (model), the user will be getting a dialog box containing standard Windows elements (input boxes, pull-down lists, etc.), helping easily and quickly defining the parameters of the fragment being assembled.



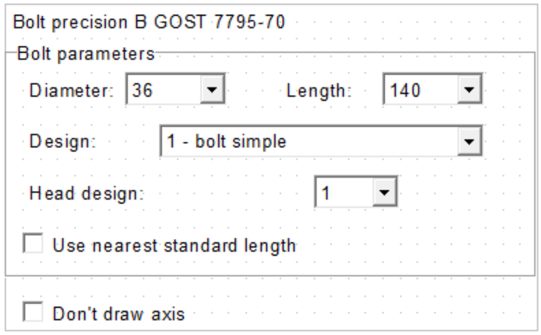
Creating such a dialog box is relatively simple. First, create a new page to hold the control elements. To do this, lunch the command **Draw** > **Control** and select the automenu option  "Create Dialog Page".

Some of the controls ("Static Text", "Group Box") do not depend on variables. When creating such elements, you define their position and then enter their name. Such elements help creating a clear dialog structure with necessary descriptions.

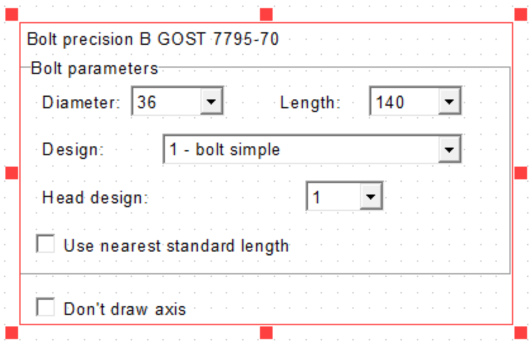
Other elements depend on variables. At creation, besides defining their position, you need to specify the variable related to such element. To input the parameters directly, you can use the items "Edit Box" and "Combo Box". The "Combo Box" is used when the variable has a predefined list of values. In our example, all variables that we input have lists of values; therefore, "Combo Box" should be used for them.

Some variable can have only two determined values. To work with these variables, it is convenient to use a "Yes/No" switch element. When creating such an element, you define the variable values for the cases when the switch is on and off.

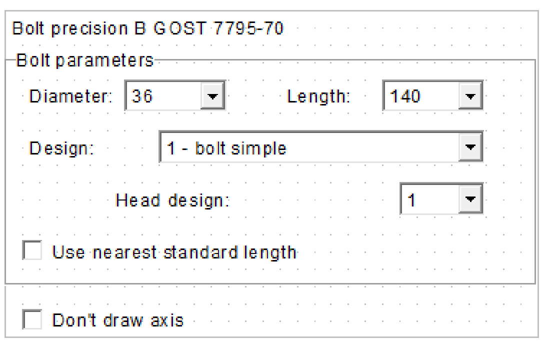
When specifying the text displayed on controls, it is possible to use variables. This will allow for changes in the text contents of the dialog according to selected bolt's parameters. For example, in the dialog of our bolt, the text for one of the controls of the type "Static text" is set in the following way: "Bolt {$ispsp}M{dd}{$polesp}%%042{ll}{$klprsp}{$mmat}{$Pocrnom} GOST 7795-70".



Next, change the page size (this can be conveniently done by the command **Customize** > **Page size**) to make its size equal to the size of the dialog box. After that, start the editing mode of the control elements and define the order of the control elements.



The order of the control elements determines their appearance in the dialog box and their activation order when using the <Tab> key. It is recommended to place static text first, followed by the control elements bound to the variables, in the order that provides easy switching between them.

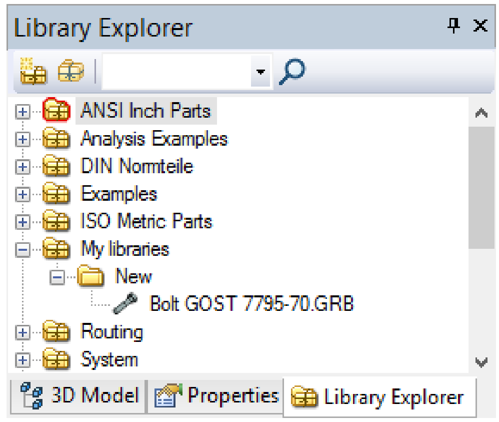


ADDING AN ELEMENT TO LIBRARY

The last step to be made is adding the element you've created to the library, and, if necessary, creating a new library.

To add an element to an existing library, copy the created element to the folder on your hard disk, where other library bolts are stored. In this case, the element will automatically appear in the library after restarting the application. You can also create a new library. When doing this, carefully check every step, as in the following example. Suppose, you need to move a library to another file system, where the root folder name is *D:* rather than *C:*. If the full path was specified when adding the element to the library ("C:\..."), the application will not find the file, and therefore, you will not be able to insert the fragment using the library.

To avoid this, use the relative rather than the full path when adding an element to a library. In this way, the path will be defined with respect to the configuration file that determines library contents. Besides, your created element should be placed "deeper" then the configuration file.



Suppose, we are to create a new library for our newly created bolt. The folder of this drawing will be *"C:\Program Files\T-FLEX Parametric CAD 15 \Libraries\My libraries\New"*, while the configuration file will be located in the folder *"C:\Program Files\T-FLEX Parametric CAD 15 \Libraries"*. When creating the library, specify the relative path with respect to the configuration file rather than the full path: *"New"*.

Now, we need to fit the drawing with the data necessary for supporting the BOM creation mechanism in T-FLEX. In **Product structure** window**, select the group to include the part into**. Define the name, material, weight and other parameters. Since the bolt name is supposed to change depending on its parameters, let's use variables for defining the name. The bolt name will be: Bolt{imp}M{d}*{ll} GOST 7795-70. In the formatted BOM, the actual values will replace the variable notations {imp}, {d} and {ll}. The weight and material can be defined in the same way.

Finally, we would like to mention that the described procedure is just a recommended technique, rather than a strictly determined sequence of actions.

Document Protection in T-FLEX CAD

This command serves for setting limitations on working with T-FLEX CAD documents. The developer of a document or a set of documents (library) can set a protection on those by using this command, and modify parameters of this protection in the future as necessary.

Working with protected documents requires the user to possess HASP protection key (the "plug"). The command has the capability of setting multiple protections on one document. Each protection requires selection of one of the **protection types.** The developer of the document can set different access rights to this document for different users. For this purpose, each protection has a set of **access types** (for editing, for opening, etc.). A protected document can have an unlimited number of protections, any of which can have its own set of access types, which allows using this document by different users with different access rights. For example, one protection would provide only the capability of opening and viewing the document, being set for all users, while another protection would provide the capability of full access, being set only for a particular HASP protection key ID.

All information about the document protection is stored within the document.

Protection Parameters

Protection Types

The following types of document protection can be set:

Not using HASP Plug. This type of protection is set for any user. This type of protection allows, for example, setting the access only for "Preview" and "Edit variables". Then, any user working with this will be able to open it view-only, without being capable of other actions, such as editing or copying the information. This is convenient if a developer intends to pass the document to the user solely for getting acquainted with the contents. In this case, the user is not required to possess HASP protection key, since previewing a document can be performed by the T-FLEX Viewer application that does not require a HASP protection key.

Specific T-FLEX CAD HASP Plug or list of HASP Plugs. This type of protection is used in the cases when the developer knows in advance the IDs of protection keys for T-FLEX CAD users who will be working with the protected documents. Enter the IDs of the T-FLEX CAD protection keys in the document protection parameters. The access to the protected document will be granted on any computer with this particular T-FLEX CAD protection key installed. If using the network key, then the access can be gained on any computer on the network.

Password with T-FLEX CAD HASP Plug. This type of protection is convenient if the developer does not know in advance the IDs of T-FLEX CAD protection keys of the users who will be working with the protected document. This type of protection is generally used when distributing a document or library (a set of documents). By setting this type of protection on the document once, you do not have to change the document protection for each new user. After that, the user shall send his (her) T-FLEX CAD protection key ID to the developer of the document. The developer uses the currently described command to generate a password based on the protection key ID, and passes it to the user. When accessing a protected document, the user must enter this password for gaining the access to the document.

Another specific HASP Plug or list of HASP Plugs. A document protection can be set not only by T-FLEX CAD HASP protection key, but also by any other HASP protection key. This type of protection is similar to the type "Specific T-FLEX CAD HASP Plug or list of HASP Plugs"; however, in this case the developer needs to specify, among the parameters, a special password for the protection key in use, which shall be obtained from the manufacturer of this protection key.

Password with another HASP Plug. This protection type is similar to the type "Password with T-FLEX CAD HASP Plug", except that it can be set on any HASP protection key, provided that the specific password is known for accessing this protection key, as supplied by the key manufacturer.

Access Types

For each protection, its specific document access types can be set. The command supports the following Access types:

Change access. This access type is set for the user who will be allowed to modify or delete protection types and access types of this protected document. Normally, this access type is set by the document developer for himself.

Edit. The user with this access type can open, edit and save a given document. Editing access means full access except the ability to "Change access".

Preview. This access type implies permission for opening, previewing and printing a protected document. When opening such document in T-FLEX CAD, all commands that can modify it are blocked.

View as fragment. A document permitted to this access type can be opened from an assembly document if it is used in this assembly as a fragment, a picture or an OLE object.

Edit variables. This access type permits editing document variables and regenerating the document with the set values. When setting this access type, additionally, the "Preview" and "View as fragment" access is granted automatically.

Insert as Fragment. The document with this type of access granted can be inserted as a fragment, the chair or an OLE object into another document. If a document is assigned this access type, then the "View as fragment" access is assigned automatically as well.

Working with "Document Protection" Command

The document protection command can be called by one of the following means:

Icon	Ribbon
	File → Document Protection
Keyboard	**Textual Menu**
<AF>	File > Document Protection

Upon calling the command, a dialog will be displayed on the screen in which you can select one of the following actions:

To modify document protection, do the following steps:

1. By using the option **Set or Modify Access Rights for Documents**, protect the intended documents by the sets of access types.
2. In the case of "By password"-type protection, the developer shall use the option **Generate Password for Document or Library** and generate the password for a user to access a document by the key ID received from the user.

The option **Get Current HASP Plug ID** allows the user to check the key ID currently installed on his machine.

Setting Access to Documents

Upon selecting the option **Set or Modify Access Rights for Documents**, the dialog will be launched for selecting files subject to setting or modifying the access.

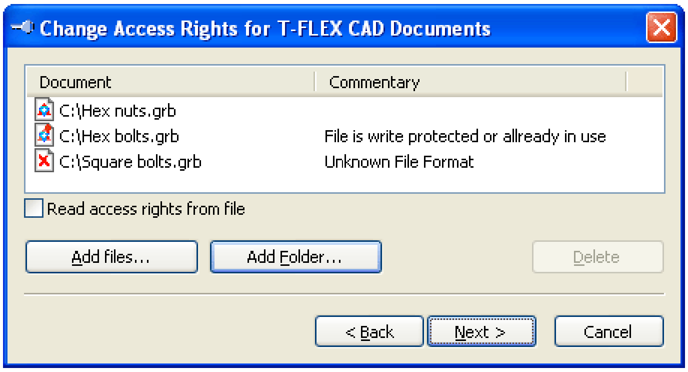
Use the buttons **[Add files...]** and **[Add Folder...]** provided in the dialog for selecting the separate files or specifying a folder whose files will be added to the list.

To set protection on files, their version must be the same as the current T-FLEX CAD version. The selected files will be added to the list. The icon before a file indicates its current state.

 – Indicates the file in the format corresponding to the current version of T-FLEX CAD. In this case, the file will be protected successfully.

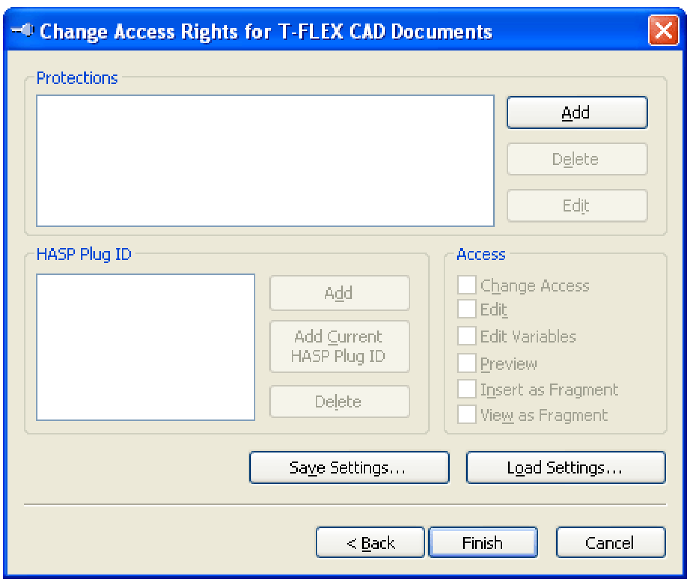
– Indicates the file in the format of one of the previous versions of T-FLEX CAD. In this case, the file needs to be resaved in the current version. This can be done by the converter of the older versions of files. Without that, the file protection will not be set.

– The file is write-protected or used by another application. In this case, either clear the file attribute “Read-only” or close this file in the other application, respectively. After that, retry protecting the document.

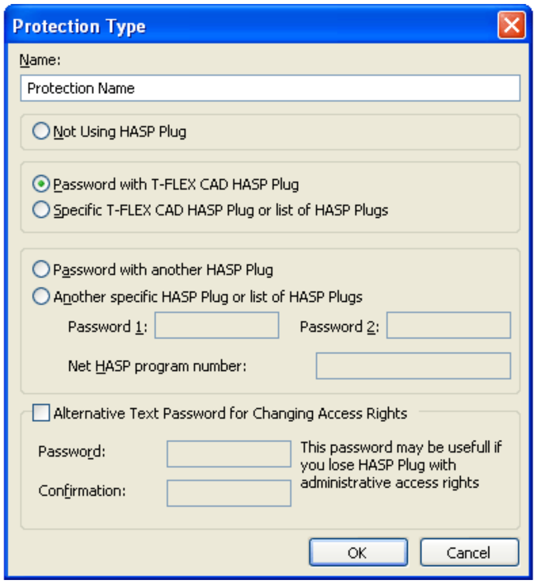


Node that in the cases when multiple documents (a library) need to be protected by the same password, preferably select all files of this library at once. This is because each newly created protection of the type “By password” is unique and requires generation and input of a unique password.

Once you selected the desired files and pressed the button **[Next]**, the dialog for modifying document access will appear.



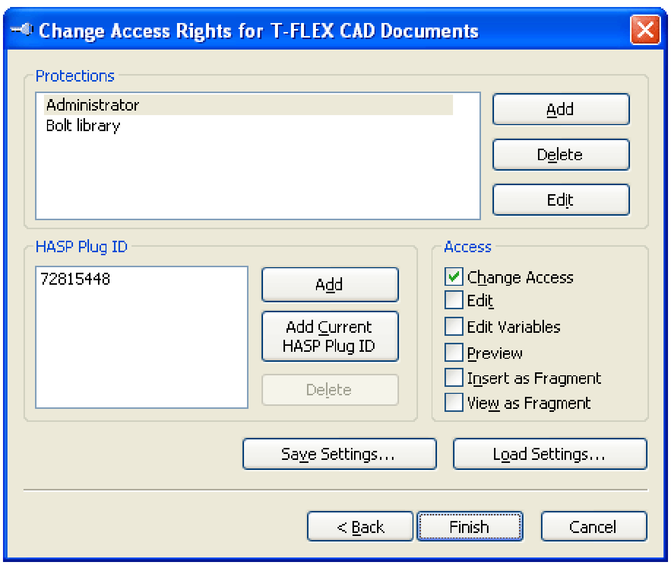
This dialog allows setting a new protection by the button **[Add]**. The button **[Edit]** serves for modifying the type of an existing protection. Upon pressing one of these buttons, a dialog will be displayed, offering selecting a protection type.

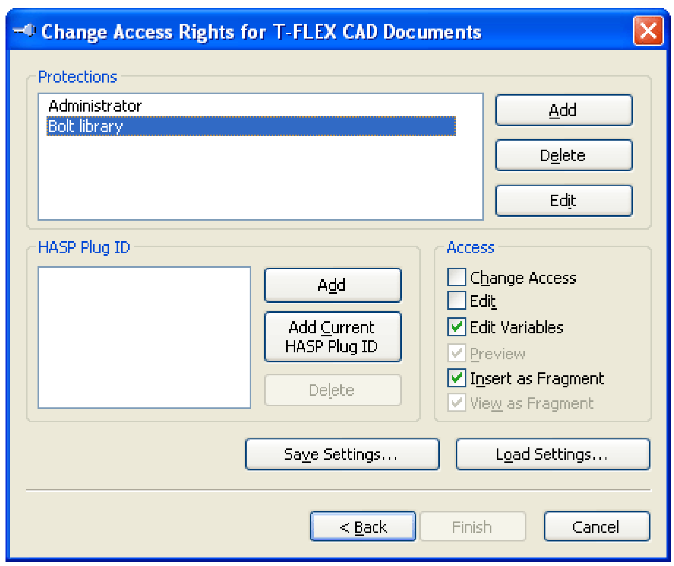


In this dialog, select a protection type and define its name. If the protection type “Another specific HASP Plug or list of HASP Plugs” or “Password with another HASP Plug” is being set, then for these types enter a special password for access to the HASP protection key in the “Password” and “Password 2” input boxes. This password is issued by the manufacturer of the HASP protection key. If this is a network key, then fill in the entry “Net HASP program number”. This information is also distributed by the manufacturer of the protection keys.

When setting up document protection by key, it is possible to enter an alternative text password. This password may be needed in the case when the protection key with administrator’s privileges is lost (that is, the protection key with the access type “Change access”).

Once the new protection is set, its name will be displayed in the list of protections in the “Change access rights for T-FLEX CAD documents” window.





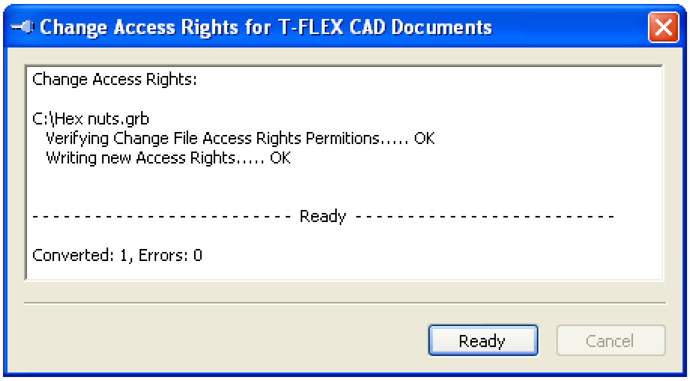
For the created protection, define access types. Using the buttons **[Add]** or **[Add Current HASP Plug ID]**, enter the key ID, if this type of protection relies on key protection.

Let's review in details one of the ways of setting document protection. The diagrams above illustrate two protections set to a document. The first protection, named “Administrator”, has the type “Specific T-FLEX CAD HASP Plug or list of HASP Plugs”. The key ID was entered for it, and the “Change access” access type set. This means that the user working with the protection key ID 72815448, can change protection parameters of this document. The second protection, named “Bolt library”, has the type “Password with T-FLEX CAD HASP Plug”, with the access types set for modifying variables and inserting fragment. For the user to gain access to a document with this protection, the following steps are required:

1. The user for whom this protection was set needs to send the ID of his T-FLEX CAD protection key to the developer.
2. The developer shall generate the password based on the received protection key ID, by using the command “Generate Password for Document or Library” (see the description below) and send this password back to the user.
3. When accessing the document, the user shall enter this password in order to gain the access.

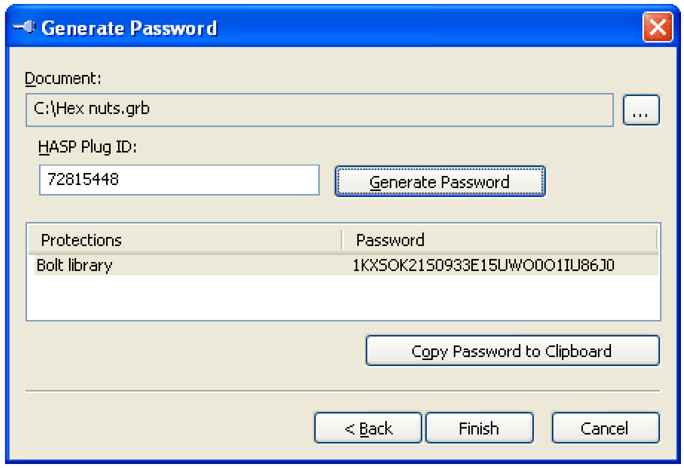
All protection settings can be saved in a file in a special format *.tfdat, using the button **[Save settings...]**. A protection can be loaded into document from a *.tfdat file, using the button **[Load settings...]**. This functionality may be helpful when one needs to add a document to an already protected library (a set of documents). To do this, one can save the protection parameters from a protected document, and then load those settings into the new document.

Upon pressing the **[Finish]** button, the protection will be set on the selected files.



Generating Password for Accessing Document or Library

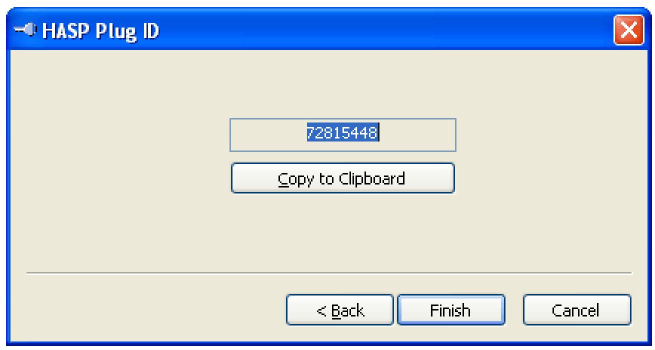
As was mentioned earlier, in the case of protecting a document by password with the protection key, the developer shall supply the password for accessing the document to the user, that was generated by the user's key ID. To do this, use the option “Generate Password for Document or Library”.



In this dialog, select a document that is already protected. After selecting the document, in the appropriate field enter the key ID, for which the document access password will be generated, and press the button **[Generate Password]**. Since a document might contain multiple protections simultaneously, the list of protections will be built, with an individual password generated for each of them. The button **[Copy Password to Clipboard]** copies the selected password to the clipboard.

Getting ID of the Current T-FLEX CAD Protection Key

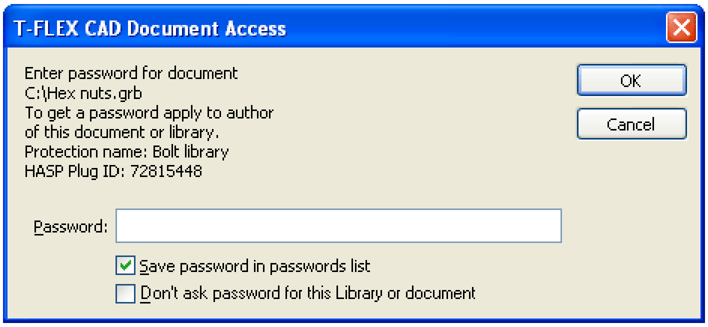
If this option is selected in the "Document Protection T-FLEX CAD" dialog, the information will be read from the current protection key. The acquired ID can be passed to the developer of the protected document for generating the password.



Working with Protected Document

If a document was protected by a specific HASP protection key, then, when the user works with this document (opening, inserting as a fragment, etc.), the current user's HASP Plug ID will be matched against the ID of the key set in the document protection parameters. In the case of matching IDs, this user will be allowed the access corresponding to the given protection. If the IDs do not match, the access to the document will be banned.

If a document was protected by password with access key, then, when the user attempts to open the document, one will be prompted to enter password.



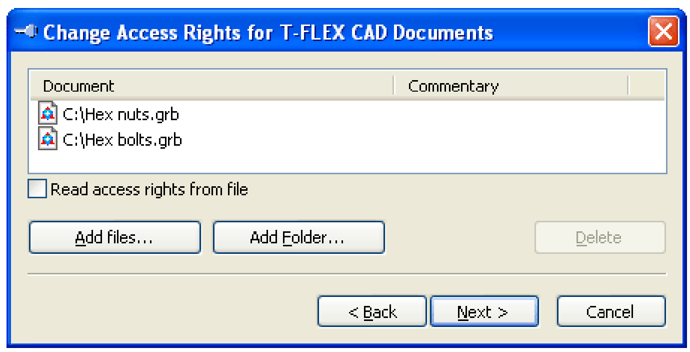
The user shall enter the password in this dialog, that he received from the document developer. (See "Generating password for access in document or Library").

Once the password is entered, it can be automatically saved in a special file DocAccess.ini, created into "Program" folder. When opening documents with this protection in the future, the password will be taken from the file DocAccess.ini.

If the password is not known at the time of opening the document, one can set the option "Don't ask password for this Library or document". Then, when using this document in the future, the password-querying dialog will not be displayed, and the access protection will be set to "Not using HASP Plug" (if this protection type was set in the document). Otherwise, if the protection type "Not using HASP Plug" was not assigned to the document, the access to the document will be banned.

When working with a protected document, the access is set that is the most of the allowed accesses. For example, if one protection grants document editing, and the other bans it, then editing document is considered allowed.

Modifying Document Protection

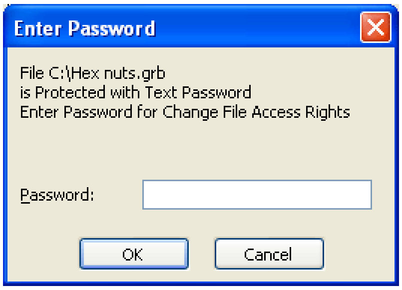


The procedure of modifying access to protected documents is similar to setting access to documents. First, by using the option **"Set or Modify Access Rights for Documents"**, select the protected documents or a folder with protected documents. In the list of files, select a file, from which the protection parameters will be read. If no such file is selected, then the protection parameters will be read from the first file in the list.

Depending on what type of "Change access" protection was used, the possibilities are as follows:

1. If the protection type was "Specific HASP Plug or list of HASP Plugs", then, upon matching Key IDs, the access for change will be granted.
2. If the protection type was "Password with HASP Plug", then, upon pressing the button "Next", a dialog will appear, requesting the password for this protection. Upon entering the password, the access for changing the protection will be also granted.

Otherwise, if the hardware key is missing that was used at the time of setting this protection, or the password for accessing this protection was not entered, then you can press the button “Cancel”. In this case, an alternative access password will be asked. Upon entering the alternative password, the access to changing the protection will be granted.



Next, a dialog will be displayed with the protection parameters set for the document. You can delete or change the parameters in this dialog.

Saving Textual Drawing Information

While working with T-FLEX CAD, you can create text documents (files) containing various information about a drawing or a model:

A system of drawing variables, with the respective values and defining expressions;

A report on variable modifications, including those used in animation;

Information about geometrical parameters of certain drawing elements;

Assembly structure.

The saved information about drawing parameters can be used for preparing accompanying documentation or as input data to other applications.

Saving Information about Drawing Variables to File

One way to store drawing variables in an external file is by using the command **WP: Save Model Parameters**:

Keyboard	Textual menu	Icon
<WP>		

This command allows saving to an external file the values, expressions and comments for all or specifically external drawing elements, at user's preference. The data is written using a specific format:

```
LD = 110 /* 110 ; Part length: from 60 to 100 1 hole, from 100 to 110 2 */
$vt1 = "Bushing" /* LD>100?"Bushing":"" ;  */
HD = 50 /* 50 ; Height of machined part (20-60) */
UGL = 10 /* 10 ; Surface slope angle (0-10) */
lo = 27.5 /* LD<=100?LD/3:LD/4 ; Distance to hole (do-LD/2) */
do = 7 /* 7 ; Diameter of hole (3-15) */
l01 = 97.5 /* lo<LD/2?(lo+15+LD/2):(2*lo+15) ;  */
l1 = 131 /* l01>LD+21||LD>100?(LD+21):l01 ;  */
ld = 35 /* l01>LD+21||LD>100?(LD-l1+56):LD+20 ;  */
$vt2 = "Bushing 2" /* LD>100?"Bushing 2":"" ;  */
$vint = "Screw" /* LD>100?"Screw":"" ;  */
```

The information saved in this way can be used in other T-FLEX CAD documents via the command **RP: Load Model Parameters**:

Keyboard	Textual menu	Icon
<RP>		

Working with the commands **WP: Save Model Parameters** and **RP: Load Model Parameters** is fully analogous to working with export/import commands of the variable editor, which is described in details in the chapter “Variables”.

If you need to save information about values of the drawing variables in an arbitrary form, it might be more convenient to use the report creation mechanism. A report is a text document that is saved in an external file. A report can include an arbitrary text containing values of the drawing variables. The contents of the information saved in an external file is determined by the report template defined in the T-FLEX CAD document. The template text usually includes document variables. At the time of saving the report to a file, the actual values may be written instead of variables. A report can be saved by a user request, or a the time of saving the entire document. Multiple report templates can be created in a T-FLEX CAD document.

Reports can be created while running animation within the command “AN: Animate Model”. In this case, the report file will be appended at each animation step with the contents of the template filled with the variable values corresponding to the current step of the animation.

Creating Report

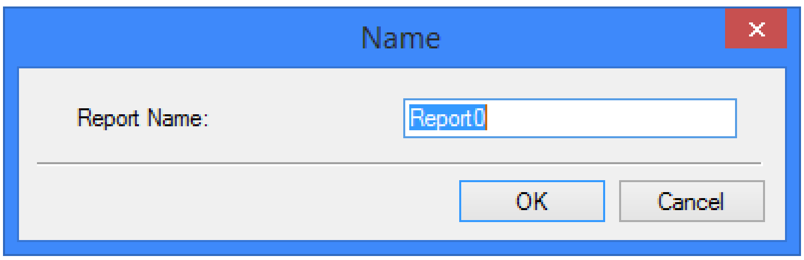
To handle reports, use the command **REP: Create Report**:

Icon	Ribbon
	Tools → Special Data → Report
Keyboard	Textual Menu
<REP>	Tools > Special Data > Report

Upon entering the command, a dialog box appears for handling report templates. If the document already contains templates, one of those will be opened in the dialog window. If no template exists in the document, the dialog window will be empty. In this case, the only accessible button will be `[New]`, allowing to create a new report template.

Upon clicking the **[New]** button, a window appears, in which you need to specify the name of the template being created. The preset default offers the name *“Report”* with the respective enumerator suffix.

Upon clicking **[OK]**, an empty report template will be opened in the command’s dialog window.



The name of the opened template appears in the **Name** parameter. Using the same parameter, you can open another template by selecting its name from the list. The selected template name can be modified using the **[Rename]** button.

Once a template is opened, you can view, define and edit its contents and properties.

The contents of the report template is displayed in the preview pane located at the bottom part of the command's dialog. It appears as plain text usually containing the document variables. The names of variables shall appear in the text within braces. The template contents can be edited directly in the preview pane, using context menu commands, or in a separate text editor window. To access the editor window, use the button **[Edit...]**.

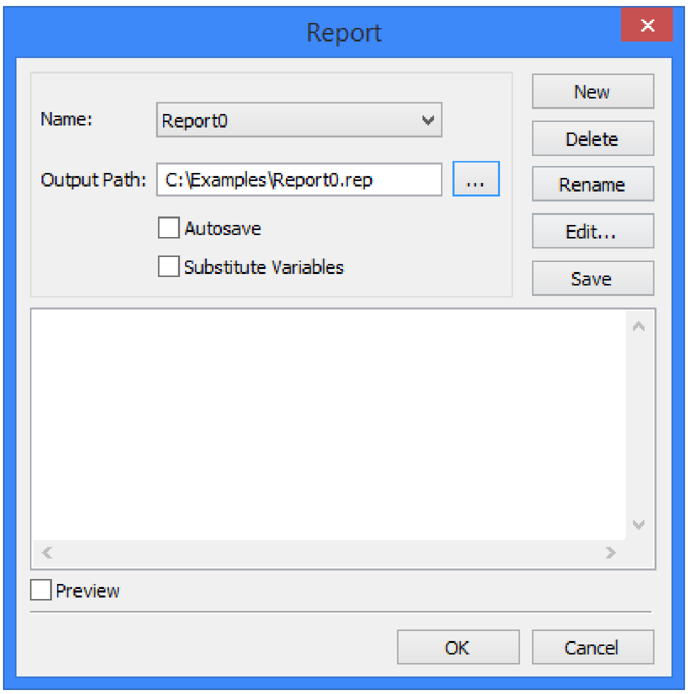
The **Preview** checkbox at the bottom of the command window allows viewing the report in the command's dialog pane exactly as it will be written to a file (with or without substituting variable values, depending on property settings defined for the report template).

The method of storing the report is described by the following group of parameters:

Output Path. This parameter defines the file name for saving the report. The button … allows browsing for another file.

Auto Save. With this flag set, the report will be automatically saved each time the drawing is saved.

Substitute variables. Defines the method of writing variables to the report. When the parameter is set, the actual current values will be substituted for the variables in the text. When the parameter is turned off, the contents of the template will be written to the report files without replacing variable names by values.



To delete the current template from the document, use the button **[Delete]**. Upon clicking the button, you would need to confirm the deletion.

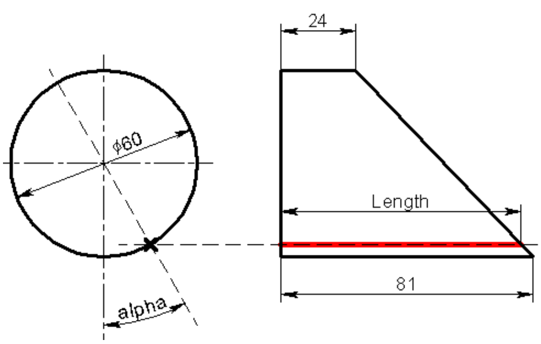
A created template can be written to the report file right from the command dialog, by clicking **[Save]**. The following text will be written for the example shown on the figure:

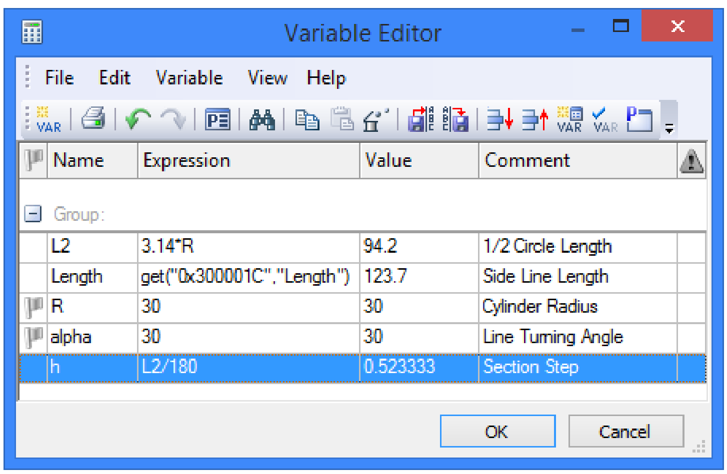
```
Parameters of the machined part for conductor:

Part length: 90
Part height: 50
Number of holes: 1
Hole diameter: 7
Surface slope angle: 10
Chamfer length: 5
```

Example of Using Report for Unfolding the Clipped Cylinder

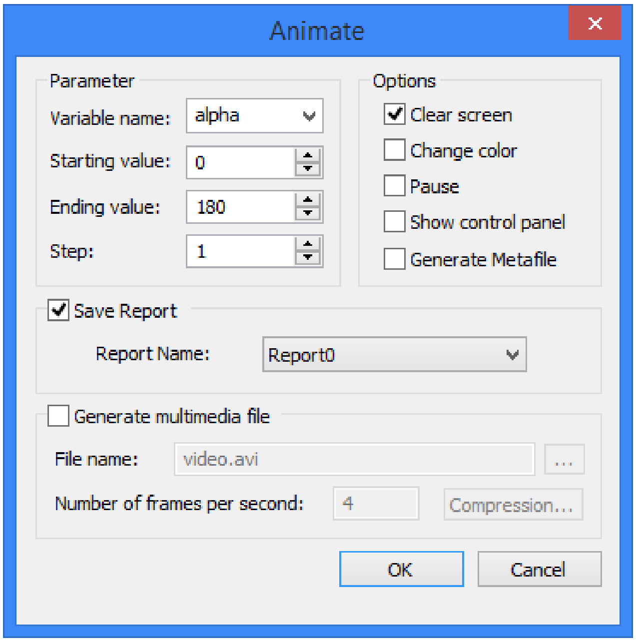
As a simple example of using the report mechanism together with the command **AN: Animate Model**, consider the solution to the task of constructing a clipped cylinder unfolding.

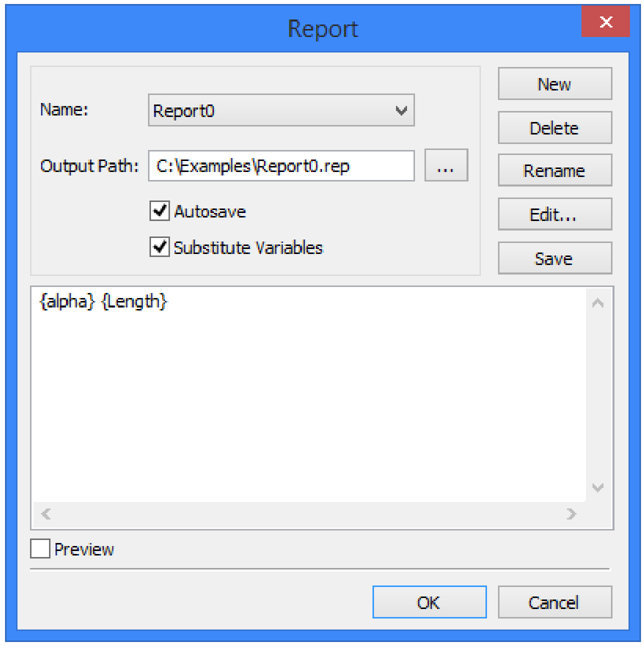
To create the unfolding, a line tilted with respect to the vertical is created at the cylinder's end face. The line parameter (angle) is defined by the variable *"alpha"*. A horizontal line is constructed through the intersection point of the former line and the circle. A segment is constructed along the intersection line of the horizontal line and the side outline edge of the cylinder. Its length is read into the variable *"Length"* by means of the function *"get"*.





Additionally, a report template was defined in the document for saving the values of the variables *"alpha"* and *"Length"*. After that, the entire drawing was animated by varying the value of the variable *"alpha"*. In the course of the animation, the variable values were output to the report.





Based on the generated report, a database was created in Excel, which was later used for creating a function in T-FLEX CAD defining the unfolding contour.

Report (the file "Report0.rep")

0 81

1 80.9957

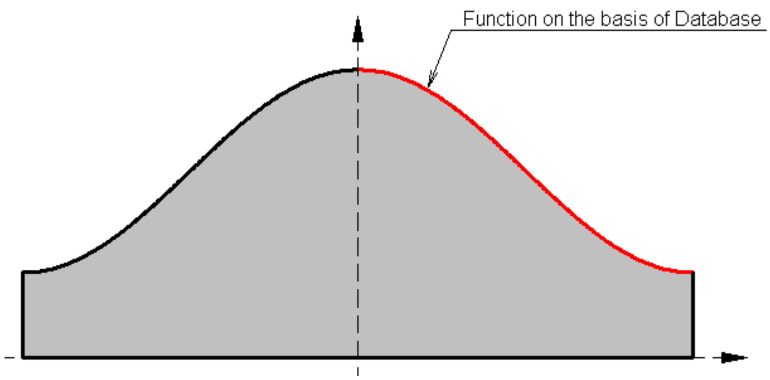
2 80.9826

3 80.9609

...

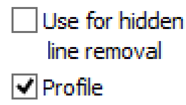
179 24.0043

180 24



PROFILES

When a part being designed is intended for further processing, which might involve use of CNC centers and creation of numerical control procedures, or for certain other purposes, then the information about the part profile geometry can be saved to a file.



A profile is defined in the commands that create and edit hatches. To designate a hatch as a profile, you would need to set the parameter **Profile** in the hatch properties dialog. If, at the same time, there is no need to fill inside of the profile contour with hatching, then you can designate it as an invisible hatch by selecting the filling method "Invisible".

Profile parameters are output to a file by the command **PR: Write Profile**:

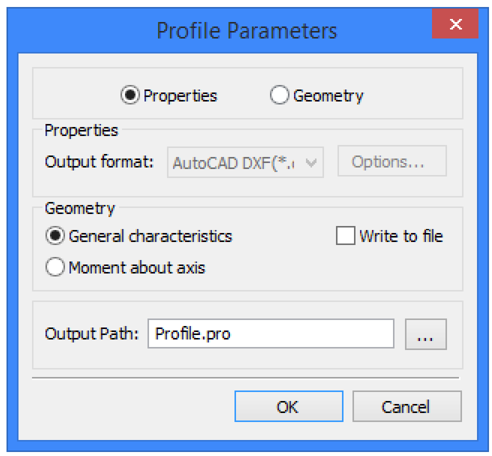
Icon	Ribbon
	Tools → Special Data → Profile
Keyboard	Textual Menu
<PR>	Tools > Special Data > Profile

After calling the command, select the hatch profile by clicking. If the drawing contains a single profile, then it will be selected automatically upon calling the command. The command parameters dialog window appears after selecting the profile.

In this dialog window, first of all you need to select which profile parameters are to write to file:

Properties. This saves information about the profile in one of the selected formats (AutoCAD DXF, AutoCAD DWG, Dragon, EIA);

Geometry. This outputs on the screen or to a file the profile geometrical characteristics (area, perimeter, moments of inertia, etc.).



The **Output Path:** parameter defines the name and the path to the file, in which the profile information will be saved. The file extension will be set automatically, based on the selected parameters and output format. To browse for the file, you can use the button .

If the general profile properties are selected, then the parameter **Output format** allows selecting the desired file format from the list:

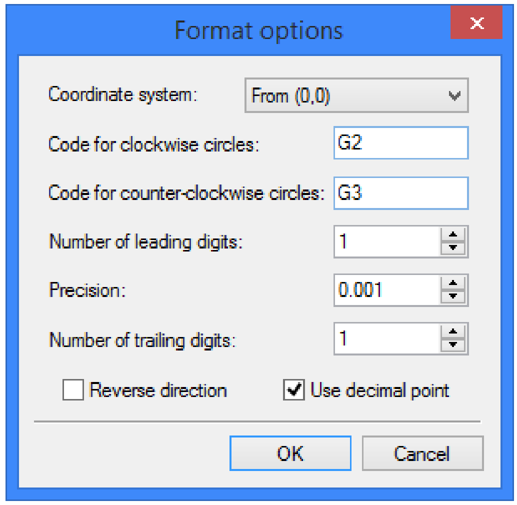
AutoCAD DXF (*.dxf). The profile information will be written in the DXF format of AutoCAD system.

AutoCAD DWG (*.dwg). The profile information will be written in the DWG format of AutoCAD system.

DRAGON (*.drg). The profile information will be written in the DRAGON system format.

EIA (*.eia). For this format, you can specify the format options by clicking the button **[Options...]**.

The EIA format parameters dialog window will appear on the screen:



Coordinate system. Defines the point, with respect to which the profile coordinates will be counted. You can select from the list: **From (0,0)**, **From contour beginning**, **Incremental**.

Code for clockwise circles. Sets the code for identifying arcs defined as clockwise.

Code for counter-clockwise circles. Sets the code for identifying arcs defined as counter-clockwise.

Number of leading digits. Sets the mandatory number of digits before the decimal point when writing numbers to a file.

Precision. Sets the rounding precision for writing numbers to a file.

Number of trailing digits. Sets the mandatory number of digits after the decimal point.

Reverse direction. This lets you get coordinates of the profile in the direction opposite to the specified one.

Use decimal point. This parameter is required for defining a fixed format without the decimal point.

Upon clicking **[OK]**, the profile information will written to the file in the selected format.

Upon selecting “Geometry” option, the **Geometry** group of parameters becomes accessible. Then, you need to select the necessary geometrical characteristics of the profile: **General characteristics** or **Moment about axis**. Upon clicking **[OK]**, the selected characteristics will be displayed in the dialog window. Additionally, if the **Write to file** flag is set, the characteristics will also be saved in an external file *“*.pro”*, defined by the parameter **Output Path**.

General characteristics include the following profile properties:

Area and **perimeter** of the profile;

X and **Y of area center** of the profile;

Central inertia about X and **Y**;

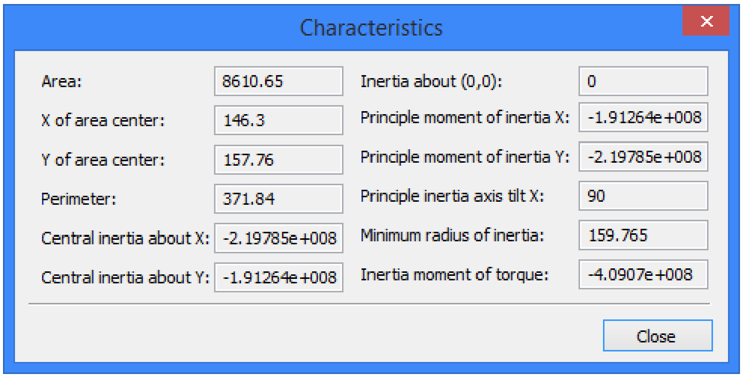
Inertia about (0,0);

Principal moment of inertia X and **Y**;

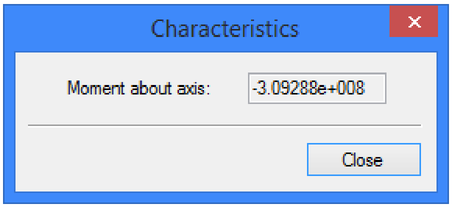
Principal inertia axis tilt X – the tilt of the principal axis of inertia with respect to the X axis;

Minimum radius of inertia;

Inertia moment of torque.



When selecting a moment of inertia about an axis for examination, after clicking **[OK]** in the command's parameters dialog, you need to additionally specify the straight line, about which the moment will be calculated. At that time, the following option will be available in the automenu:



	<L>	Select Line

After selecting the axis, the dialog window will display the calculated moment.

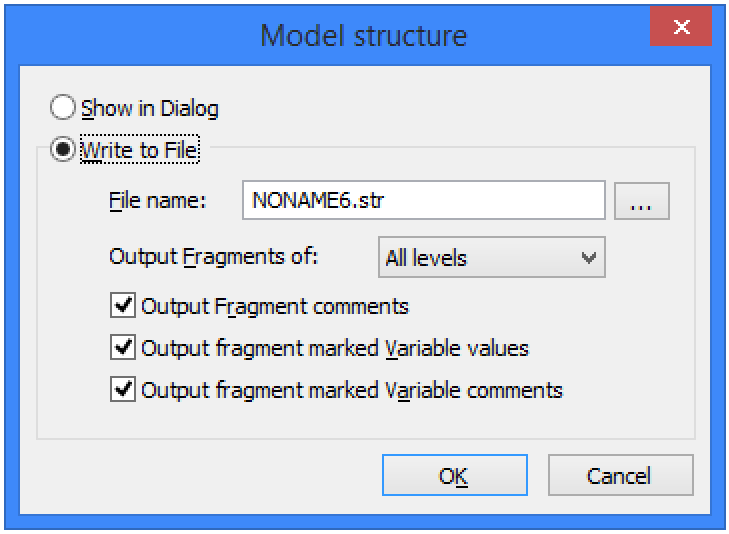
Assembly Document Structure

The command **SS: Save Model Structure** allows building the structure of a T-FLEX CAD assembly document. The result can be viewed in the command's window or written to a text file "**.str*".

The command is called as:

Icon	Ribbon
	Tools → Special Data → Structure
Keyboard	Textual Menu
<SS>	Tools > Special Data > Structure

After calling the command, the dialog window appears on the screen for defining parameters of the structure building process.



First of all, one needs to select the method of displaying the resulting structure – in the command's dialog or in a separate text file. For this, set the appropriate option – either **Show in Dialog** or **Write to File**.

If writing to file was selected, then you need to specify the following additional parameters:

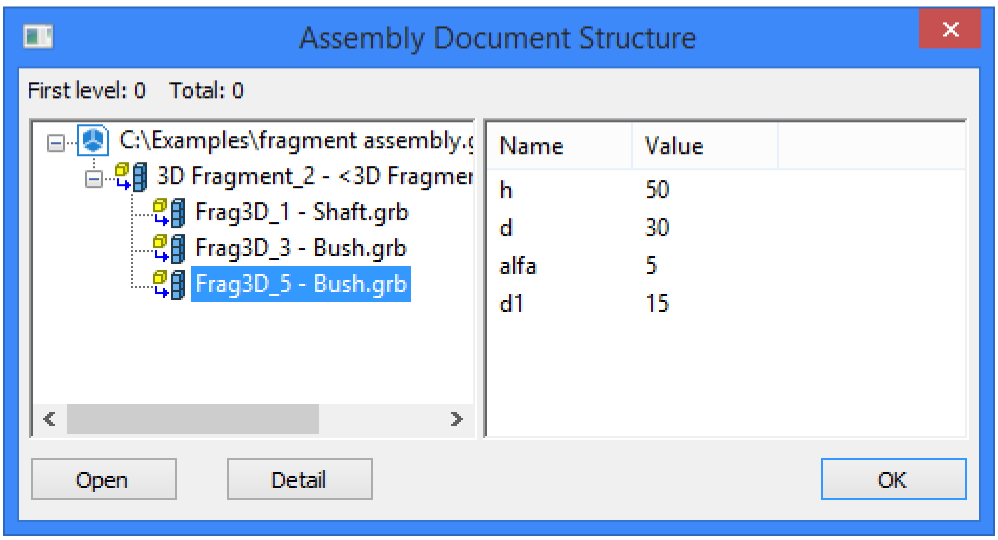
File name. This parameter specifies the file name and location, in which the document structure will be written. The button  allows browsing for an existing file to write to.

Output Fragments of. This parameter allows selecting the nesting depth of the fragments included in the assembly structure – either only the fragments at the **First level** of nesting or the entire tree of fragments (**All levels**).

Besides the assembly structure proper, additional information can be written to the file that relates to the enclosed fragments of the first level of nesting. Its availability and contents are determined by the state of the flags **Output Fragment comments**, **Output fragment marked Variable values**, **Output fragment marked Variable comments**.

Upon defining all parameters, click the **[OK]** button to get the document structure. If outputting to a file, the system will write the results of the document structure analysis to the specified external file, and then exit the command.

If in the dialog display mode, a window will appear that will contain the document structure.



This window displays the hierarchical document structure in the way of a tree of fragments, which can be browsed with the help of or the <Up>, <Down> keys. The tips box above displays the information about the contents of the selected fragment. In the case of first nesting level of fragments, the external variable values are displayed in the pane on the right hand side. The not found fragments and the fragments that were opened with errors are displayed by the following icons: for 2D fragments, and for 3D fragments.

The button **[Open]** opens the selected fragment document in a separate window. The button **[Detail]** opens the selected fragment in the detail mode (see the chapter "Creating assembly drawings. Using fragments").

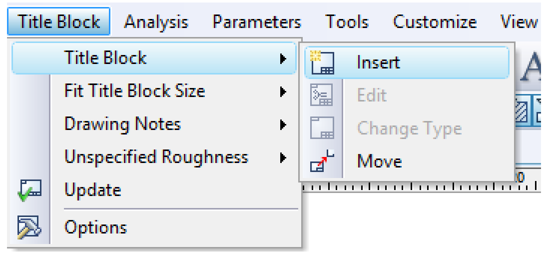
To get done with the command, click the **[OK]** button in this window.

DRAWING TITLE BLOCK

The drawing title block module is an application that can be launched from the command **Customize** > **Applications**.

This application starts by default, as indicated by the availability of the **Title Block** textual menu item.

The drawing title block commands are also available in the group “Title block” of the main toolbar.

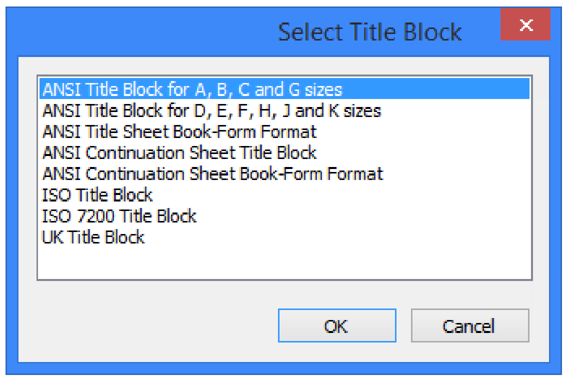


CREATING TITLE BLOCK

To create a title block, use the command **Insert Title Block**:

Icon	Ribbon
	Title Block → Title Block → New
Keyboard	Textual Menu
	Title Block > Title Block > Insert

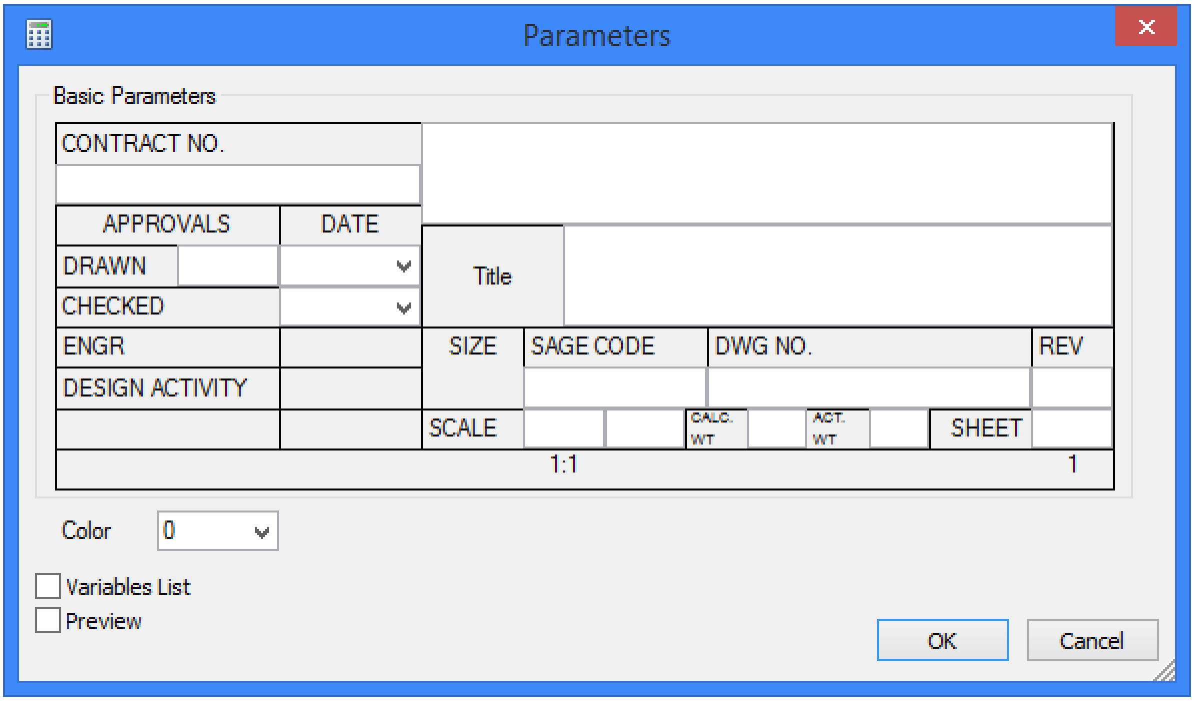
After calling this command, a dialog window appears on the screen, which lists all types of title blocks provided with the system. You can manually add to this list the title blocks of your own design (see the section “Customize”).



Select the title block type from the provided list, which you intend to apply to the drawing. After that, a window will appear for filling the title block. This window is a standard window for editing values of fragment external variables and will display by default the dialog created in the title block document using the interface control elements. Upon enabling the **Variables List**, flag this window is displayed as the external variables editor.

Any field that has the  graphic button next to it can be filled using the list of values. You will be able to create and/or modify the list of values using the context menu commands.

When applying the title block, its fields **Drawing Number** and **Title** are automatically bound to the hidden variables of the current document. The same variables define the data for the BOM. Therefore, when inputting data in these fields of the format frame, the corresponding data is automatically entered in the BOM.



Additional parameter **Color** sets the color of the title block lines and text.

To edit the contents of the title block fields, use the command **Edit Title Block**:

Icon	Ribbon
	Title Block → Title Block → Edit
Keyboard	Textual Menu
	Title Block > Title Block > Edit

You can reassign the title block type with the command **Change Title Block Type**:

Icon	Ribbon
	Title Block → Title Block → Change Type
Keyboard	Textual Menu
	Title Block > Title Block > Change Type

To modify the format frame position, use the command **Move Title Block**:

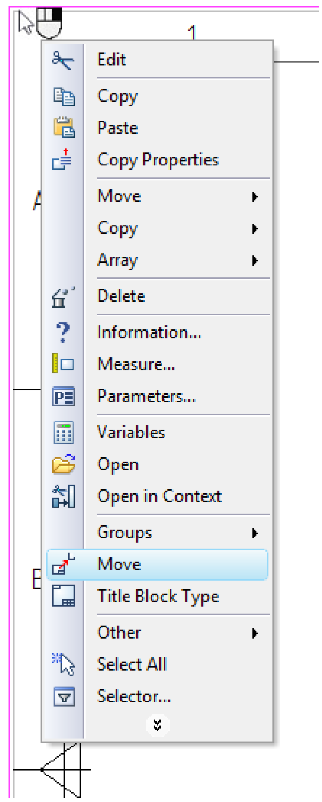
Icon	Ribbon
	Title Block → Title Block → Move
Keyboard	Textual Menu
	Title Block > Title Block > Move

After calling this command, the title block starts rubberbanding on the screen. Point the cursor at the desired point in the 2D window and click - the title block will be moved to the new position.

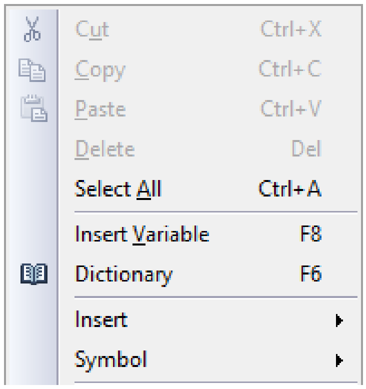
You can change the title block type and move the title block by selecting the respective item in the context menu, which appears on right-clicking the title block fragment.

The title block can also be filled directly in the drawing. To do that, place the text cursor in the title block field, which needs to be filled, and click . A blinking cursor will appear in the selected field, indicating the readiness to input text, and a button with an arrow will appear next to the selected field, serving to select a value from the list. Initially, the list is empty (except for the fields in the "Date" column). If you need to fill in the list in order to have list values available in the future, then in the selected field enter the necessary text, and then select the "Add to List" item in the context menu.

Since the title block is a drawing fragment, there is another way to apply it - by using the command **FR: Create Fragment**.



Please note that, when inserting several title blocks into the current document (for example, for drawings located on different pages), the data from all title blocks will be bound to the same variables of the current document. As a result, the contents of the fields "Drawing Number" and "Title" of all title blocks will be same. As you edit fields in one title block, the text will be changing in all the rest of them. To disable this mode, you need to cancel the established relation with the hidden drawing variables, and then define the new field value. To do this, you need, regardless of the variable editing technique, call the context menu for the respective field. In the menu, clear the check in the "Insert Variable" item. After that, you can edit the text value in the current field. To relate this field with a new variable, call the same command again and define the name of the new variable in the upcoming "Insert Variable" window.



When editing the title block using the dialog for editing fragment external variables, you have another alternative. If you enable the “Variables List” flag in this dialog, then, instead of a dialog with interface control elements you'll see the standard list of the format frame fragment external variables. Find the “$Drawing_Number” and “$TITLE” variables in it (the fields “Drawing number” and “Title”), and replace their values with constants.

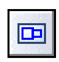
> In some cases, you may need to suppress automatic binding of title blocks variables with the variables of the current drawing. This may be needed, for example, when frequently creating documents containing several different drawings. In such a case, it is more convenient to edit the title blocks document itself, to have the automatic variable bindings suppressed on inserting it into the current document. To do this, open the format frame document, go into the variables editor and delete the contents of the “Assembly Variable Name” parameter from the above-listed variables.

Title Block Fitting

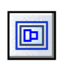
The commands below serve to define the size of the format of the drawing, and therefore the title block.

Keyboard	Textual Menu	Icon
	“Title Block > Title Block Size > Standard”	

The command picks the format of the closest standard size.

Keyboard	Textual Menu	Icon
	“Title Block > Title Block Size > By Drawing Limits”	

In this case, the format size is determined by the drawing dimensions in the 2D window.

Keyboard	Textual Menu	Icon
	“Title Block > Title Block Size > By Current Window”	

If this command is selected, then the format size will be determined by the dimensions of the current 2D window.

In the latter two cases, the automatically calculated format height and width will be entered in the drawing status (the command **ST: Set Document Parameters**). The format itself will be assigned the "Custom" value.

DRAWING NOTES

You can enter the mode of creating drawing notes with the command **Insert Drawing Notes**:

Icon	Ribbon
	Title Block → Technical Requirements → New
Keyboard	Textual Menu
	Title Block > Drawing Notes > Insert

Upon calling this command, a pane is displayed on the screen, in which you can enter the drawing notes. The drawing notes are a paragraph text by default, therefore the automenu provides the same options as those available when working with paragraph text (see the chapter "Text"). It is possible to insert fragments of frequently used text from dictionary, and also use variables and their respective values (the option <F8> – Insert Variable tab). Note that if the "Transparent Text Editing" option is set (the **ST: Set Document Parameters** command, the **View** tab), then you can edit text and variable values by pointing the cursor to the text pane and clicking .

Icon	Ribbon
	Title Block → Technical Requirements → Edit
Keyboard	Textual Menu
	Title Block > Drawing Notes > Edit

The **Edit Drawing Notes** command serves to edit the contents of the drawing notes.

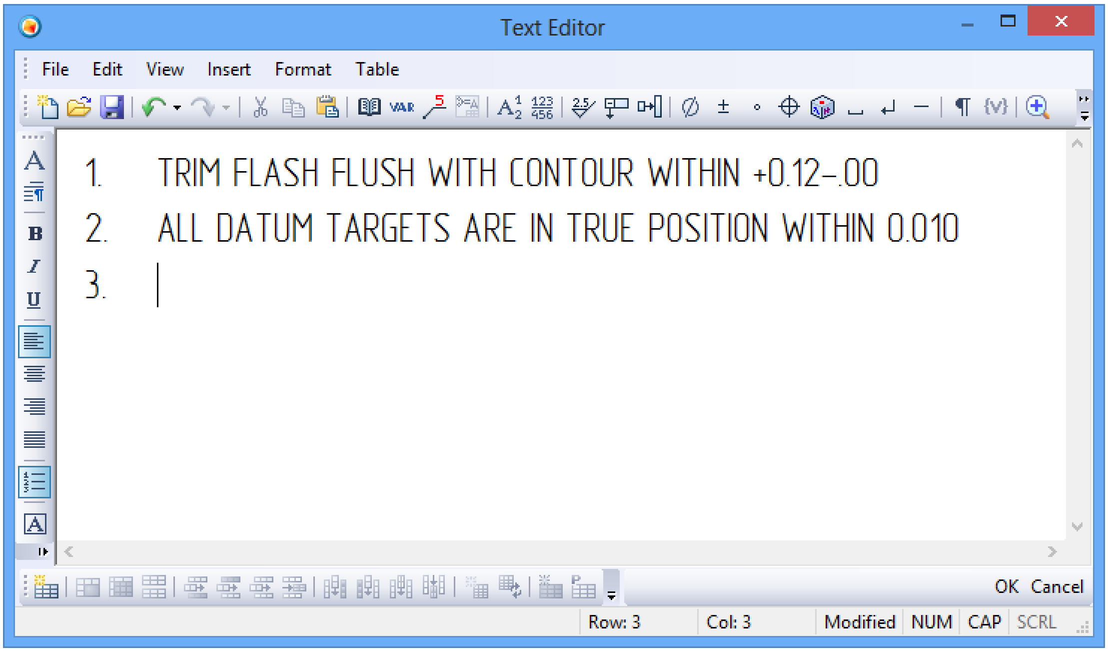
> Drawing notes location on a page, its format and initial contents are defined by the special hidden "Text" created preliminary in the title block. This Text is named "DrawingNotes" and created the way that its position and size are dependent on the position and size of the title block.

Technical requirements for the document can also be specified without initial reference to a specific drawing. For example, they can be created in the document that contains only a 3D model, and then they can be used on the drawing. Such technical requirements constitute a formatted numbered text which is related to the entire document. For example, it can be specified in the prototype.

For specifying/editing technical requirements of the document the special command is used:

Icon	Ribbon
	Title Block → Technical Requirements → Drawing Notes
Keyboard	Textual Menu
	Title Block > Drawing Notes > Technical Requirements

When invoking this command, the text editor window similar to the one used for creation of multi-line and paragraph texts is displayed on the screen.



The list of technical requirements specified in this editor is stored inside the document as a multi-line text. If we invoke the **Create technical requirements** command later on, this text will automatically be inserted into the text of technical requirements of this command. The user can delete extra lines, which are not necessary for the given page, manually.

Technical requirements created with the help of the command **Formatting** > **Technical Requirements** > **Tech requirements of document…** can also be inserted into the standard text (see the “Texts” chapter of this user’s manual).

Unset Roughness Symbol

Icon	Ribbon
	Title Block → Unspecified Roughness → New
Keyboard	Textual Menu
	Title Block > Unspecified Roughness > Insert

After calling this command, the roughness parameters window will appear (see the chapter "Roughness Symbols"). Once the parameters are defined, the roughness symbol will appear on the drawing.

Icon	Ribbon
	Title Block → Unspecified Roughness → Edit
Keyboard	Textual Menu
	Title Block > Unspecified Roughness > Properties

This command serves to edit roughness properties.

> Position of the unspecified roughness symbol is defined by the special hidden "Roughness" element created inside the title block document.

Variations Table

You need to create variations of detail/assembly to fill in a variation table. Variations are created in the **FCE: Model configurations and variations** command.

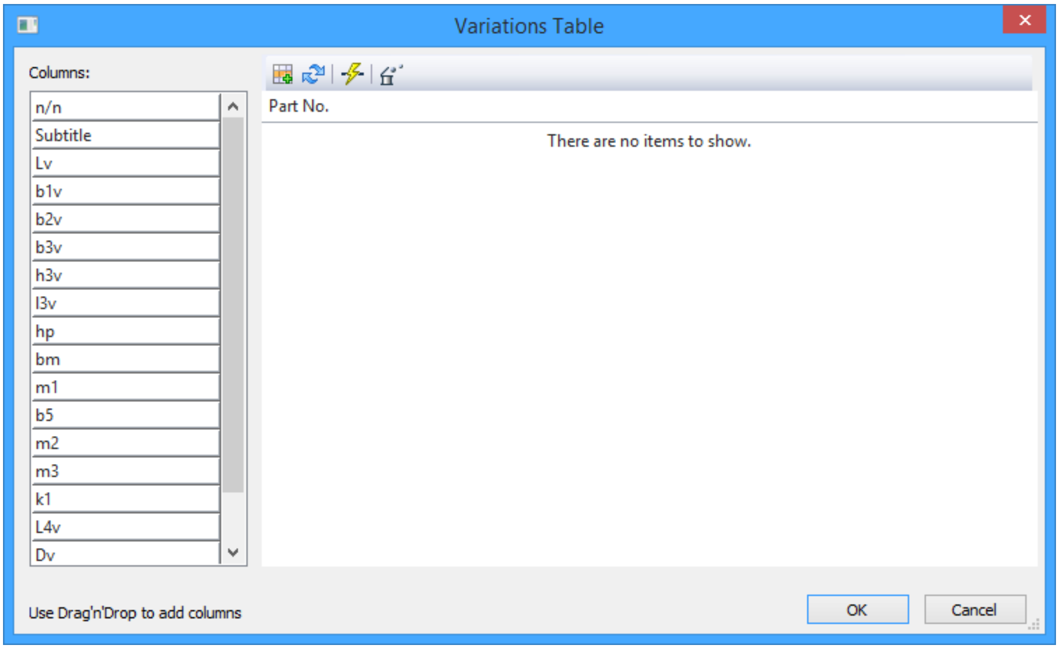
> More information about the command can be found in "Auxiliary tools for 3D assemblies modeling" chapter.

Use the following command to create a variation table:

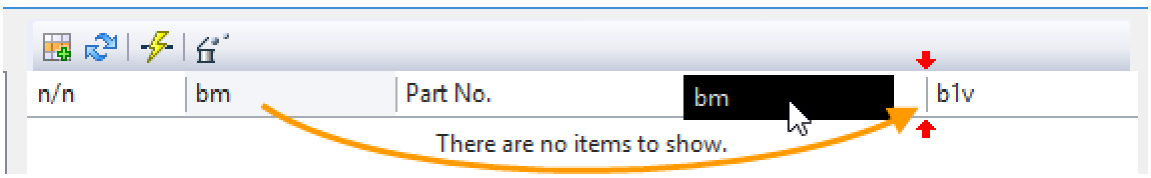
Icon	Ribbon
	Title Block → Variations Table → New
Keyboard	Textual Menu
	Title Block > Variations Table > Create

Variations table window appears after calling the command. Model external variables, subtitles and a row of string sequence number that may be used as columns are located in the left part of the window. For this purpose you need to select variable and move it to the right part of the window using Drag'n'Drop.

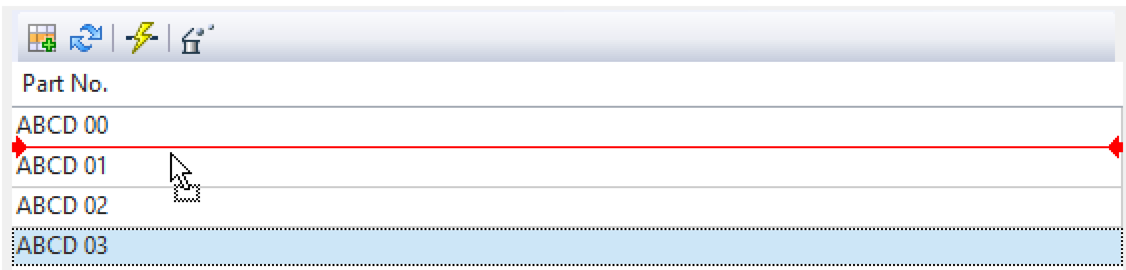
The variation table window is located to the right. It contains columns that will be used in variation table on the drawing.



You can change the order of columns. Move column header for this purpose. Two red arrows show where your column will be located. The column, moved from the left side of the window using Drag'n'Drop, can be arranged in the same way.



You can change strings sequence. You need to select string and move it to a new position.



Options for the variations table are located on the toolbar in the upper part of the window.

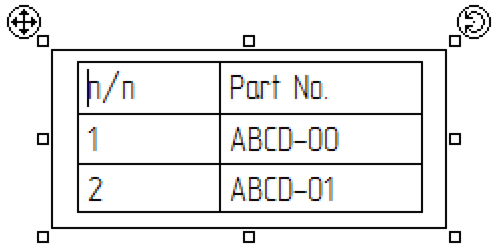
 **Insert row**. The option adds empty string to the end of the list.

 **Update sequence number**. The option updates sequence numbers in the "n/n" column. The option is used when strings are moved and sequence is broken.

 **Reset**. The option cancels all changes and returns table to the original state.

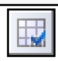
 **Delete row**. Deletes selected string from the list.

The table appears on the drawing after pressing [OK] button. You can move it using marker in the upper left corner.



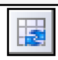
Further, you can manage the created table as a text document.

Use the following command to run a variation table editor:

Icon	Ribbon
	Title Block → Variations Table → Edit
Keyboard	Textual Menu
	Title Block > Variations Table > Edit

More information about working with text can be found in “Text” chapter.

You can add a new column or make any other changes in the table using command:

Icon	Ribbon
	Title Block → Variations Table → Change
Keyboard	Textual Menu
	Title Block > Variations Table > Variations Table…

After using the command, the **Variation table** window appears.

All changes in the variation table existing on the drawing will be discarded after pressing [OK] button, and table will be updated. It is recommended to insert all changes in the **Variation table** window.

Updating Title Block

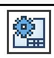
While working on a drawing, you may need to modify some layout parameters, for example, move the title blocks into a new position. In such a case, the position of the applied drawing notes and unset roughness symbol will remain unchanged. To have such elements positioned according to the new title blocks position, there is the command:

Icon	Ribbon
	Title Block → Options → Update
Keyboard	Textual Menu
	Title Block > Update

When this command is called, a window appears, in which you can mark the layout elements that you need to update (Title Block, Drawing Notes, Unset Roughness Symbol).

Parameters

To set up layout parameters, there is the command:

Icon	Ribbon
	Title Block → Options→ Options
Keyboard	Textual Menu
	Title Block > Options

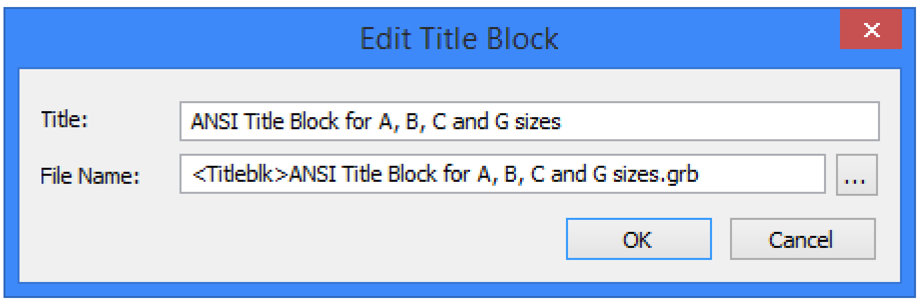
After calling this command, a dialog window appears on the screen with two tabs.

"Title Blocks" tab

This tab contains the list of title block types used for layouts. You can edit this list using the buttons **[Edit]**, **[Add]**, **[Delete]**. The first two buttons call the same dialog window, only in the first case it is used for an existing type, and in the second - for a new one.

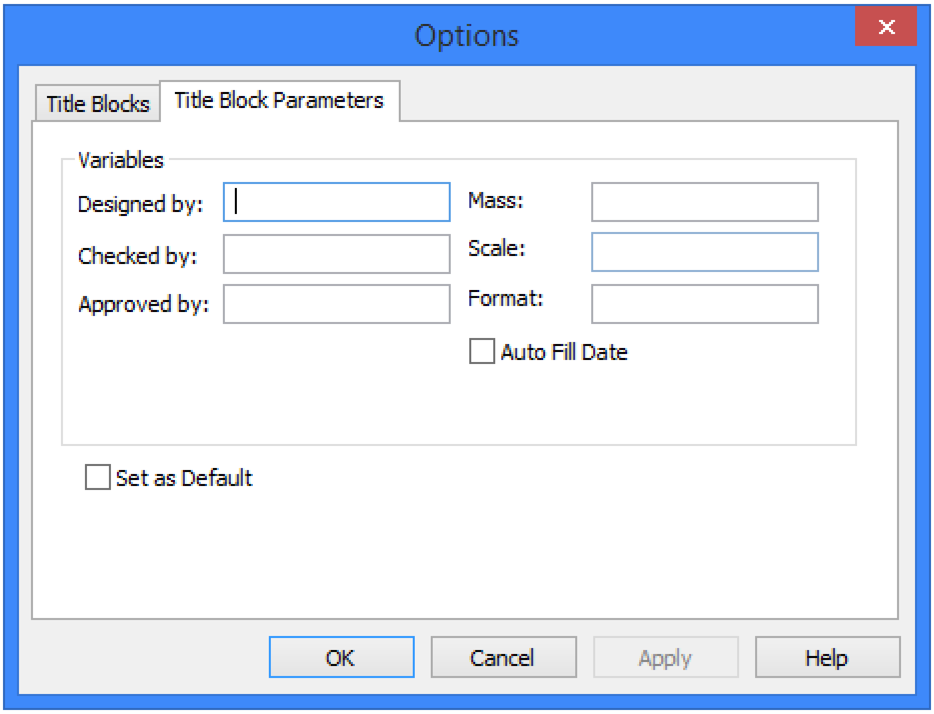
Title. Specify the title block name in this entry, which will be entered in the list of title block types.

File name. Specify the path to the format frame file in this item entry.



The "Title Block Parameters" tab

Serves to define the parameter values that will be displayed by default when filling in the title block.



MACROS

T-FLEX CAD design process often involves performing various calculations. Some tasks can be handled by using the variable editor. However, the variable editor offers only limited calculation capabilities. For example, it does not support use of loops. This limits the scope of tasks that can be handled directly within T-FLEX CAD.

Another design issue that users face is much time spent for performing repetitive tasks. For example, it might be necessary to create a set of layers in one drawing that is similar to a set of layers in another drawing, and distribute the objects of the former drawing on the newly created layers. Such a task requires certain concentration, especially considering that the number of layers in the document may be large.

Such issues can be easily addressed with the help of «macros».

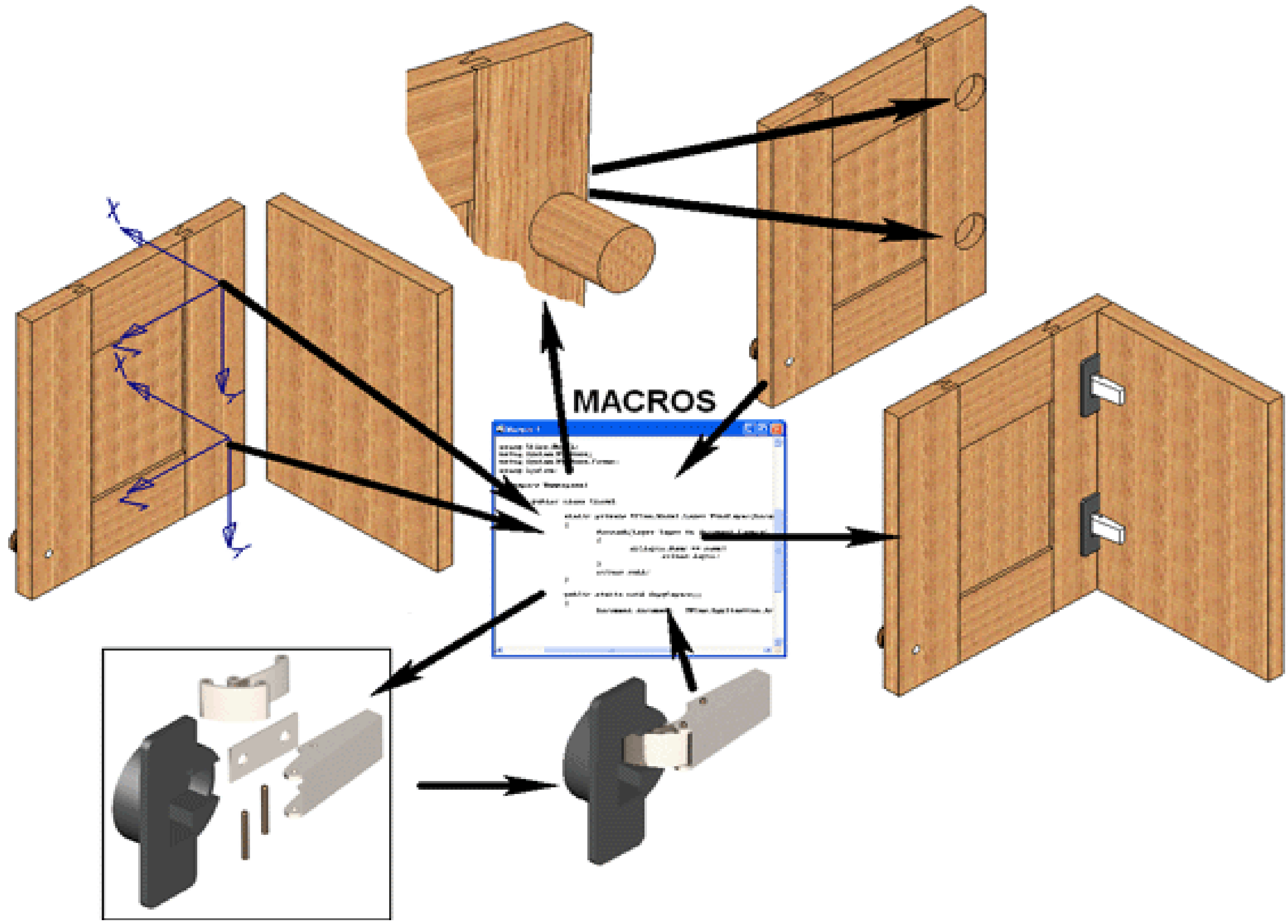
GENERAL INFORMATION

Macro in T-FLEX CAD is a program written in one of the programming languages, using T-FLEX CAD Open API functions. With the help of macros it is possible to automatize execution of various operations with documents of the T-FLEX CAD, reducing the number of the user-performed operations to minimum.

A macro is executed as a single command, that is, by launching a macro once, the whole set of actions will be performed that are contained within the macro (calculations, handling of T-FLEX CAD objects, displaying results etc.).

> A detailed description of API functions can be found in the section Help “Help on Open API…”.

Macros can be used for working with 2D drawing as well as with the elements of 3D model. Let us give one more example. When working on creation of three-dimensional assembly model, the user has to create slots and holes at the fastening location for assembling parts and insert fastener fragments with certain parameters. It is possible to reduce the number of user's operations by writing a special macro upon launching of which it will be sufficient to just select the LCS. The rest – creation of holes in a part, selection of required fastener set with certain parameters and the insertion of fastener into the assembly – will be done by the macro. In this case the large number of operations which the user had to execute manually is replaced by two operations only: launching a macro and selection of LCS.



Those are just a few examples where macros can be used.

Macros are created and stored inside the T-FLEX CAD files with extension *.grb. From the point of view of programming, the file *.grb, inside of which the macros are created, is defined as a **Project**. Each Project may contain an unlimited number of macros.

A special service window “Macros” is used for viewing available for execution macros and their launching. To make the macro available for usage and visible in the window “Macros”, the document containing the macro must be opened in the T-FLEX CAD.

Storing a macro within a particular *.grb file does not imply that this macro can only be used in the given document. The user can launch any macro contained in the Projects presently open in the T-FLEX CAD (no matter which document is presently active in the T-FLEX CAD main window)

It is possible to make the macros of some Project available for execution even without opening the Project's file in the T-FLEX CAD window. To make that possible, it suffices to place the Project's file in a special folder “...T-FLEX CAD/Program/Macros/”. The macros of the files located in this folder are accessible all the time.

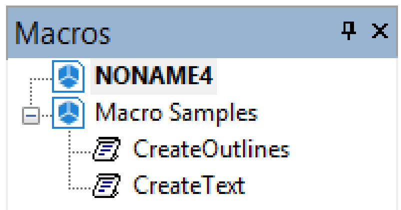
> By default the folder "...T-FLEX CAD/Program/Macros/" contains only standard macros included in the package. The user can create one's own macros library by placing the files from Projects with macros to the folder «/Program/Macros/».

«Macros» Window

By default, the service window for working with macros – the window "Macros" – is not shown in the T-FLEX CAD window. It can be made visible with the help of the following command:

Icon	Ribbon
	View → Window → Tool Windows → Macros
Keyboard	Textual Menu
<Alt><5>	Customize > Tool Windows> Macros

This window can be also invoked from the context menu by clicking in the zone of toolbars of the T-FLEX CAD window.



The main purpose of the window "Macros" – display of the macros available for execution. For launching a macro it is required to select it in the window and double-click.

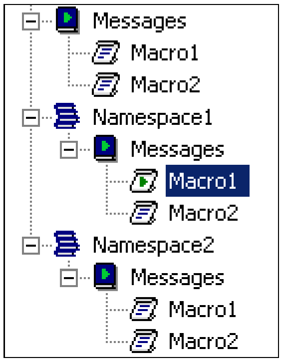
The icon preceding the macro marks its current state:

- – Nonactive macro;
- – A selected macro;
- – A running macro.

The user can launch any macro displayed in this window. For example, the project "Macros Examples.grb" is located in the folder "/Program/Macros/". The macro "Export2DPicture" of this project exports the image of the active 2D page of the current document as an internal picture to the new T-FLEX CAD document. When launching this macro, the dialog for saving the file will be invoked. After indicating the name, a new document containing an internal picture is automatically created and opened in the T-FLEX CAD window. The new document will contain an internal picture whose image duplicates the contents of the active 2D page of the source document.

> Launching a macro can be carried out automatically as well when working with the user-defined dialog (if this operation is defined for the control element "Button"). Moreover, to launch a macro, the user can define a special user-defined command and add it to the text menu or the toolbar. This option is described in more detail in the chapters "Controls. Creating User-Defined Dialogs" and "Customizing System" (Section "Adding User-Defined Commands").

The standard macros included in the T-FLEX CAD package have a simple structure which does not make use of namespaces. In the window "Macros" they are displayed as a list of macros of each Project. The Projects with more sophisticated hierarchy can be displayed as a multilevel structure of the namespace (the folders with the icon), classes (the folders with the icon) and macros contained in these classes .



Macro Editor

Window of Macro Editor

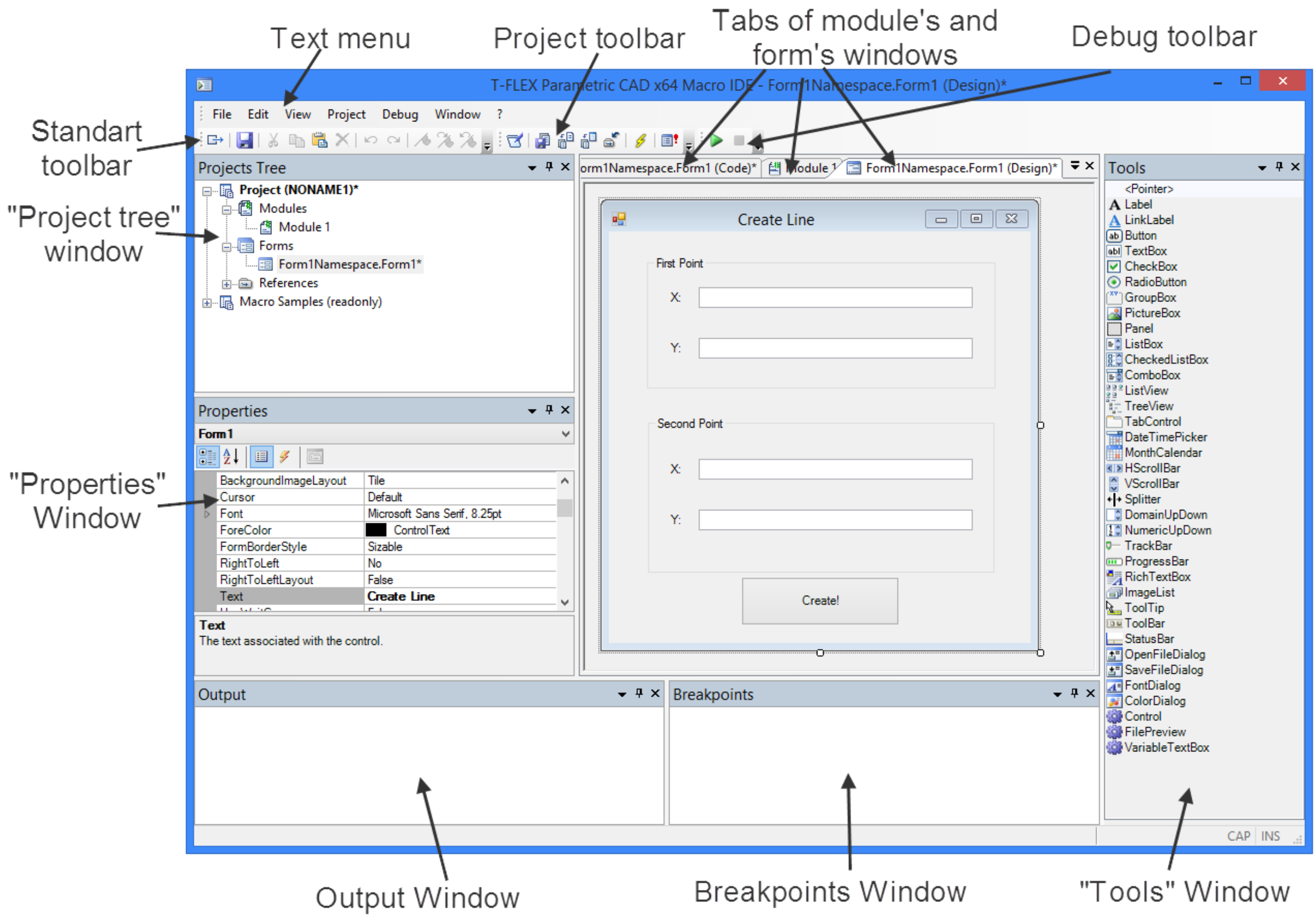
A special **Macro editor** is provided for creating macros. It is a macro development environment integrated in T-FLEX, which contains the full set of editing and debugging tools. Writing a macro does not require any application or programming environment. All tools are packed within the Macro editor.

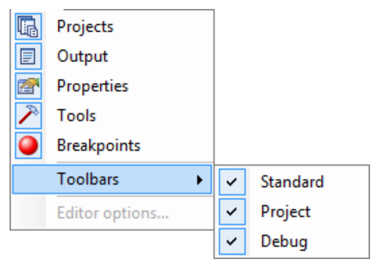
The macro editor can be opened by the command:

Keyboard	Textual Menu	Icon
<WM>	Tools > Macro Editor	

> The Macro Editor works with the main T-FLEX CAD window synchronously. That is, when the window of Macro Editor is open, the user can execute various operations in the window of the T-FLEX CAD application itself (for example open/close documents).

The figure below depicts the main tools of the macro editor.





The text menu “**View**” allows the user to control the display of the service windows and toolbars of the Macro Editor.

The placement of all service window s of the Macro Editor can be customized in the same way as it is done for service windows of the T-FLEX CAD main window. In particular, it is possible to “attach” the service windows to the edges of the main window, make them “popup” or put them into the “floating” mode. To save the working space of the screen, several windows can be combined into a single group window. The service windows which are not used can be turned off.

Projects accessible for editing are shown in the window “Projects”. These are the files opened in the T-FLEX CAD window, and also the documents contained in the folder “.../Program/Macros”. With the help of this window, the user can view the structure of the Projects and also perform various operations with their elements (create modules/forms, open them for editing, etc.).

> The window "Projects" of the Macro Editor and service window "Macros" are synchronized. Closing a Project (document) file in T-FLEX CAD also makes it close in the «Projects» window of the macro editor. Similarly, a macro that has been created and made executable in the Macro editor consequently appears in the «Macros» window of T-FLEX CAD.

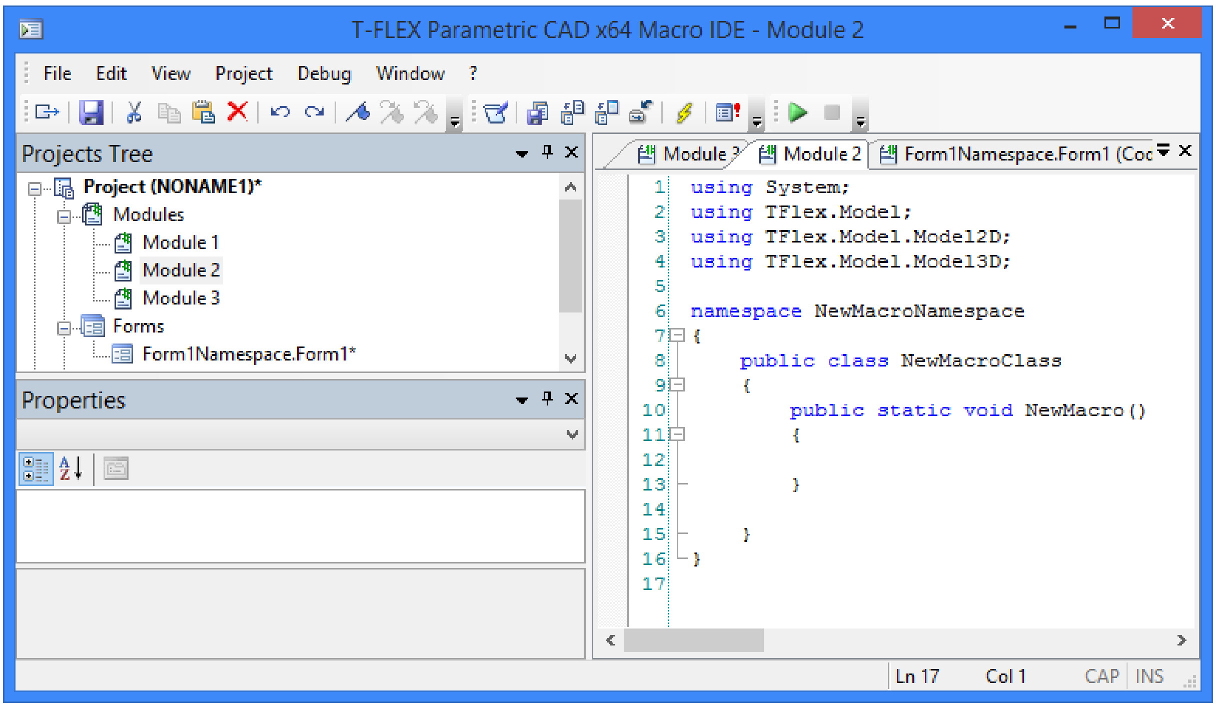
Creating/editing the codes of modules and the contents of the forms of the Project is performed in the "working" window of the Macro Editor. This is the main window of the Macro Editor. The code of each module or form of the Project is opened in a separate window. For switching between the windows of modules and forms it is possible to use the tabs of these windows located in the upper part of the main window of the Macro Editor. The windows of codes can be combined into vertical or horizontal groups.

The windows "Properties" and "Tools" are required for creating/editing graphical forms of Projects. The windows "Break Points" and "Output data" are used during compilation and debugging of the created module.

Managing Projects. Project Structure

The window "Projects" in the Macro Editor is used for managing Projects. When opening the Macro Editor, this window shows all currently accessible Projects. These include all documents opened in the current T-FLEX CAD application, and also Projects stored in the special folder "...T-FLEX CAD/Program/Macros/". It is possible to work on several Projects synchronously.

Each Project is displayed as a hierarchical structure which includes the sets of modules, forms and references.



A so-called modular programming approach is used when writing macros in T-FLEX CAD environment. Modular programming is the way of structuring a program as a union of small independent blocks, called modules, whose structure and behavior are governed by certain rules.

Module is a program unit which includes various components (types, constants, variables, namespaces, classes, procedures and functions). From the point of view of program code, the macro is a procedure – a part of the program used for executing a separate, specific task.

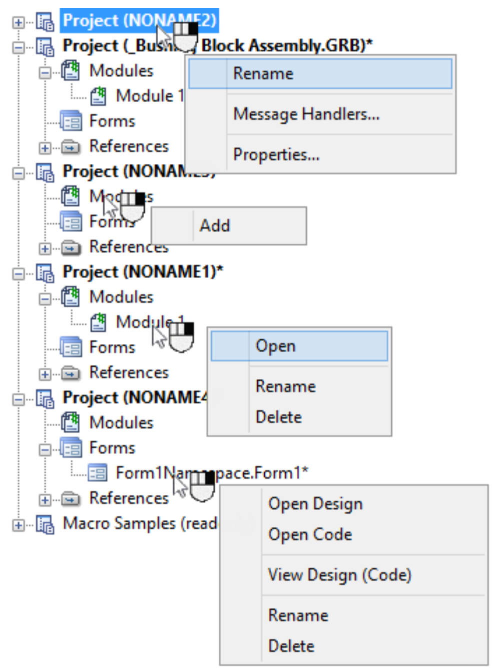
Form is the dialog box of a macro, within which control elements can be placed (such as text, buttons, text input boxes, radio groups etc.), used for operating the macro. More details on working with this window will be revealed in the topic «Creating macros with screen forms» of this chapter.

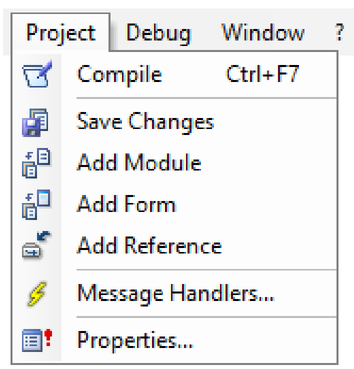
References are required for accessing properties, methods and events of a particular object and for using this object while programming a macro. For example, to use T-FLEX CAD commands in a Project, you would need to reference TFlexAPI.dll. At the time of creating a new project, main references are already added to it for using system's objects and also data and objects of the T-FLEX CAD.

The Project can contain any number of modules, forms and references. In turn, each module of the Project can include any number of macros.

To avoid problems associated with repetitions of names of the macros in the complex Projects, the namespaces can be used.

For any object of the Project's tree (Project, folder of modules or a separate module, form, reference), the context menu with a collection of special commands is available. These commands allow the user to rename a selected Project, add a new module or a form into it, etc. Some of these commands can be also invoked from the text menu "Project" and from the toolbar "Project".





Creating Project. Project Properties

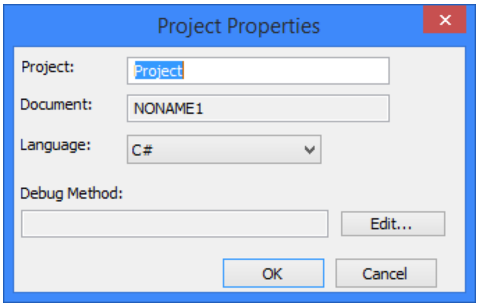
To create a new Project, open a new document in the T-FLEX CAD. In the window "Projects" in the Macro Editor a new Project will appear automatically. The structure of this Project will contain only references.

The programming language C# is used by default for any new Project. If it is desired to use another programming language, it should be specified right after creating a new Project. The change of programming language after creation of modules and forms leads to throwing into the discard of already existing modules and forms. In this situation it will be required to remove all modules and forms of the Project, and then create them again.

The programming language of the Project (macro) is indicated in the dialog of Project properties:

Keyboard	Textual Menu	Icon
<WM>	Project > Properties...	

> The dialog of Project properties can be also invoked from the context menu of the Project in the window "Projects".



After calling this command, a dialog appears with the properties of current Project. The Project name is displayed in the "Project" box of the dialog. The field "Document" displays the name of the T-FLEX CAD file that contains the current Project.

The "Language" combo box provides selection of the programming language, which will be used in the current Project. Currently, the polling programming languages are available for selection: Visual Basic, C#.

An additional parameter **Debug Method** is used only in the mode of debugging a macro. If the debug mode is not used, this method does not have to be selected.

To specify a method to be debugged, it is required to press the button **[Edit...]**. As a result, the dialog with the list of methods of the current project will be invoked. Manipulation with this dialog is described in the section "Debugging Macro".

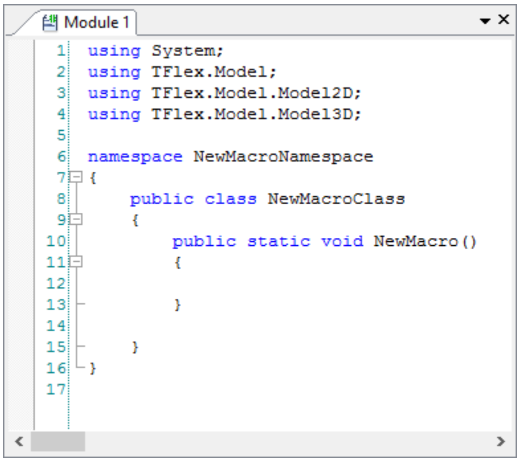
Creating Modules. Window of Module's Code

After selecting a Project's programming language, the user can start creating required modules and forms.

Creating a new module in the Project is simple. For that, it is sufficient to place a cursor on the title "Modules" in the Project's tree (window "Projects"), click and select the command **"Add"** from the appeared context menu. You can also call the command from the text menu or from the toolbar "Project" of the Macro Editor:

Keyboard	Textual Menu	Icon
-	Project > Add Module	

As a result, in the main window of the Macro Editor a new window with the code of the module will appear. In this window, a part of the code will be created automatically; references, namespace, class and procedure will be declared.



To create a simple macro, it is sufficient to write one procedure and compile a Project.

Note that, by default, the new Project contains only standard references to four libraries. If, in the macro, you use the library the reference to which does not exist in the standard list, you must add it to the list of references independently. To do that, use the command:

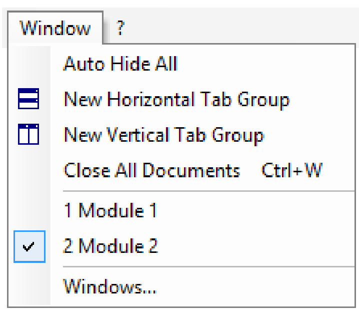
Keyboard	Textual Menu	Icon
-	Project > Add Reference	

This command can be also invoked from the context menu of the window “Projects”. To do that, select the section “References” in the tree of the current Project and click . The command **Add** can be found in the appeared context menu.

Managing Windows of Modules' Code

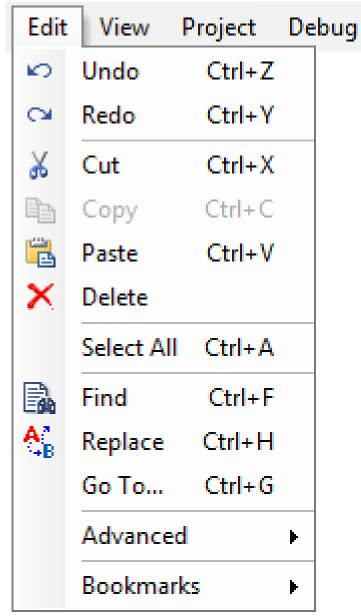
The code of each module of the Project is displayed in a separate window. The tabs located in the upper part of the main window are used for switching between the modules' windows. The windows of the code can be combined into vertical or horizontal groups.

The commands of the text menu “Window” are used for managing the modules' windows.



Settings of Code Editor

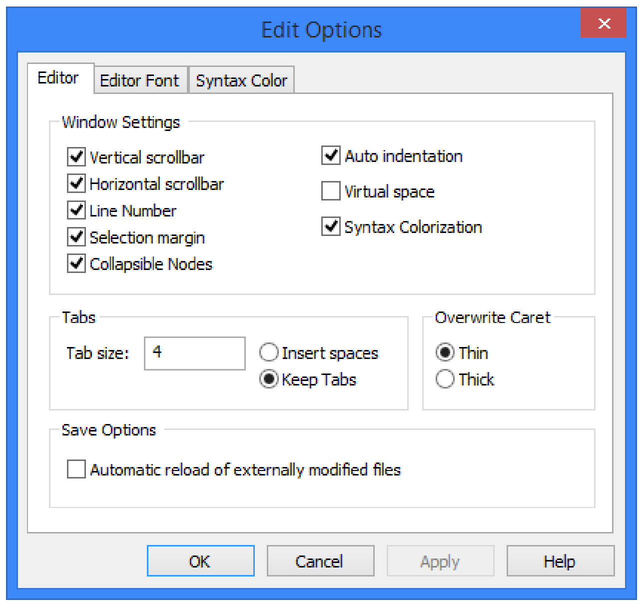
The window of modules' code is a text editor with the standard text editing capabilities (“Copy”, “Paste”, etc.). Commands for working with the text can be invoked from the context menu, the toolbar “Standard” and from the text menu “Edit”. In addition to that, in the dialog of main window settings, you can define specific settings of the code editor: automatic line numbering, automatic creation of indents, highlighting various syntax entities of the code with different color, etc.



Settings of module's code editor are specified with the help of the command:

Keyboard	Textual Menu	Icon
-	View > Editor Options...	-

When this command is invoked, the window of code editor settings is opened.



The following parameters are specified on the tab “Editor”:

Group of parameters **Window Settings**:

Vertical scrollbar and **Horizontal scrollbar**. These parameters control the view of the scrollbars in the windows of code;

Line Number. Parameter controlling the display of line numbers in the code;

Selection Margin. This parameter controls the view of the margins of the assigned bookmarks and breakpoints at the left side of the windows of code;

Collapsible Nodes. This parameter shows\hides nodes that allow collapsing code sections.

Auto indentation. If this parameter is checked, each newly created line of the code will have the same indent as the previous line;

Virtual space. When this parameter is turned on, the cursor can be moved to any place in the code's window. When this parameter is turned off, the cursor is located within the boundaries of the existing text;

Syntax Colorization. Parameter allows highlighting various syntax structures of the code with different colors. Color palette is specified on the tab “Syntax Colors” of the given dialog;

Parameter **Tabs/Tabs size** sets the length of tab symbols used for creating automatic indentation of lines of code. The width of the space symbol serves as a unit of length;

Group of parameters **Overwrite Caret** defines the view of the cursor in the text replacement mode.

The last parameter of the given tab – **Automatic reload of externally modified files** – is used in cases when the document of the Project is open simultaneously in several applications of the T-FLEX CAD. When this flag is checked, the Project will be automatically synchronized in all applications.

Two other tabs of the dialog of settings – tab “Editor Font” and tab “Syntax Color” – define font parameters used for displaying the code and parameters of color highlighting of various syntax structures of the code.

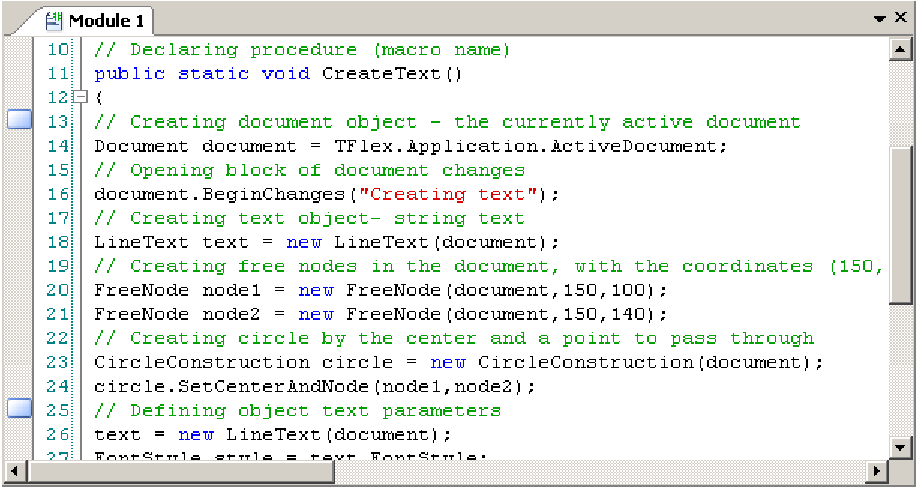
Using Bookmarks

To make the work with the large code more convenient, in the window of the code editor it is possible to define *bookmarks* on specific lines of the code. The bookmarks allow the user to quickly move inside the code's window from one marked line of the code to another.

To add a bookmark it is required to place a cursor at the required line of the code and call the command **Toggle Bookmark**:

Keyboard	Textual Menu	Icon
<Ctrl><F2>	Edit > Bookmarks > Toggle Bookmark	

After invoking this command, in the code's window a bookmark will appear to the left of the chosen line (if, in the code's window, the margin of bookmarks is activated).



After “placement” of bookmarks it will become possible to quickly move across the module's text with the help of the following commands:

Keyboard	Textual Menu	Icon
<F2>	«Edit > Bookmarks > Next Bookmark»	
<Shift><F2>	«Edit > Bookmarks > Prev Bookmark»	

The active bookmark (i.e., the bookmark located on the line of cursor's present position) is highlighted with the color – .

To delete the bookmark, call twice the command **Toggle Bookmark** for the marked line of the code.

Example of Macro 1

As an example, let us create a macro which will automatically build a construction line – a circle and a text string attached to a node of this circle.

To accomplish that, create a new 2D document in the T-FLEX CAD. In the Macro Editor select its Project in the Project's tree (window «Project Tree") and create a new module inside the Project. In the window of module's code insert the following text:

```
// Declaring references
using System;
using TFlex;
using TFlex.Model;
using TFlex.Model.Model2D;

// Declaring class
   public class NewMacroClass
   {
```

```
// Declaring procedure (macro name)
    public static void CreateText()
    {
// Creating document object – the currently active document
        Document document = TFlex.Application.ActiveDocument;
// Opening block of document changes
        document.BeginChanges("Creating text");
// Creating text object– string text
        LineText text = new LineText(document);
// Creating free nodes in the document, with the coordinates (150,100) and (150,140)
        FreeNode node1 = new FreeNode(document,150,100);
        FreeNode node2 = new FreeNode(document,150,140);
// Creating circle by the center and a point to pass through
        CircleConstruction circle = new CircleConstruction(document);
        circle.SetCenterAndNode(node1,node2);
// Defining object text parameters
        text = new LineText(document);
        FontStyle style = text.FontStyle;
        style.FontName = "Arial";
        style.Bold = true;
        style.Italic = true;
        text.Color = 1;
        text.Node = node2;
        text.Circle = circle;
        text.TextValue = "Text on circle";
        text.HorizontalAlignment = TextHorizontalAlignment.Center;
// Closing block of document changes
        document.EndChanges();
    }
  }
```

Once the code of the macro is written, you need to compile the Project.

Debugging, Compiling and Running Macros

Compiling Project

To start compilation, use the following option:

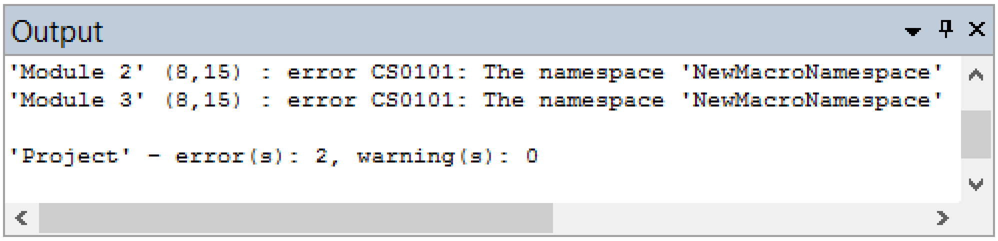
	<Ctrl><F7>	Compile

Project compilation is a required action before executing the macro. When compiling, the system analyzes the macro code and finds errors. Messages about found errors are shown in the window “Output data”.

Compilation errors occur when the system fails to interpret the input text. Those errors could be due to incorrect syntax of an instruction or due to specifying an incorrect method or property.

«Output» Window

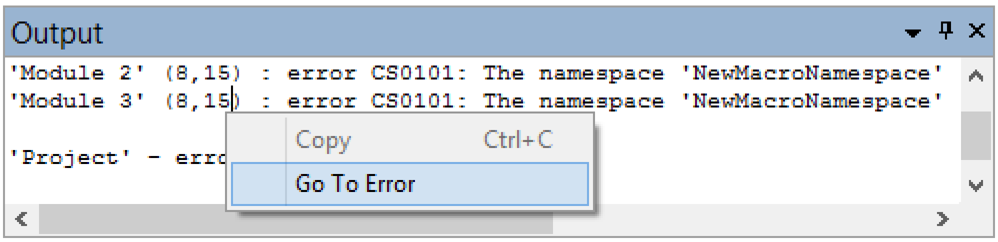
The «Output» window is located at the bottom of the macro editor. It serves for displaying messages about the current state of the Project. Errors and warnings resulting from Project compilation are displayed in this window.



An error message displayed in this window contains information about the error location in the code window (line and column number) and the error number.

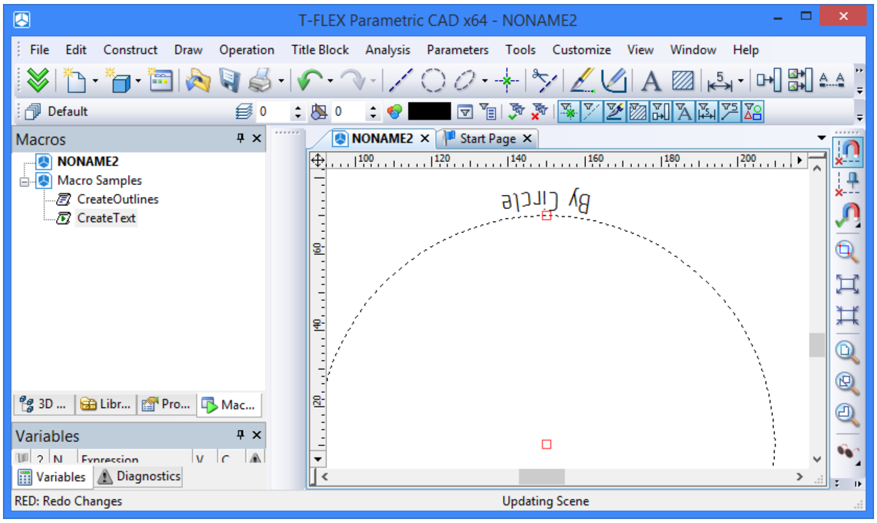
To jump to an error location in the code window, you can either double-click or call the command **Go To Error** from the context menu accessible by right clicking .

The error type and information about the error can be found in the Microsoft Developer Network (MSDN) application development guide by the code displayed within the error message in the «Output» window.



Upon a successful project compilation, the macro will be added into T-FLEX CAD “Macros” window and can be executed thereafter.

The figure below depicts the result of running the macro that creates a construction circle and a text string wrapped on the circle. The code of this macro was shown above as part of the module's code window description.



Debugging Macro. Breakpoints

If, after executing a macro, it is evident that it does not work correctly, it is possible to use the *macro debug mode* to find the errors in the code.

In the debug mode the macro is compiled (if necessary) and then launched for execution with the possibility of stopping at the intermediate points - *breakpoints*. At the moment of the stop the user can check the current state of the objects of macro with the help of the command "Evaluate expression".

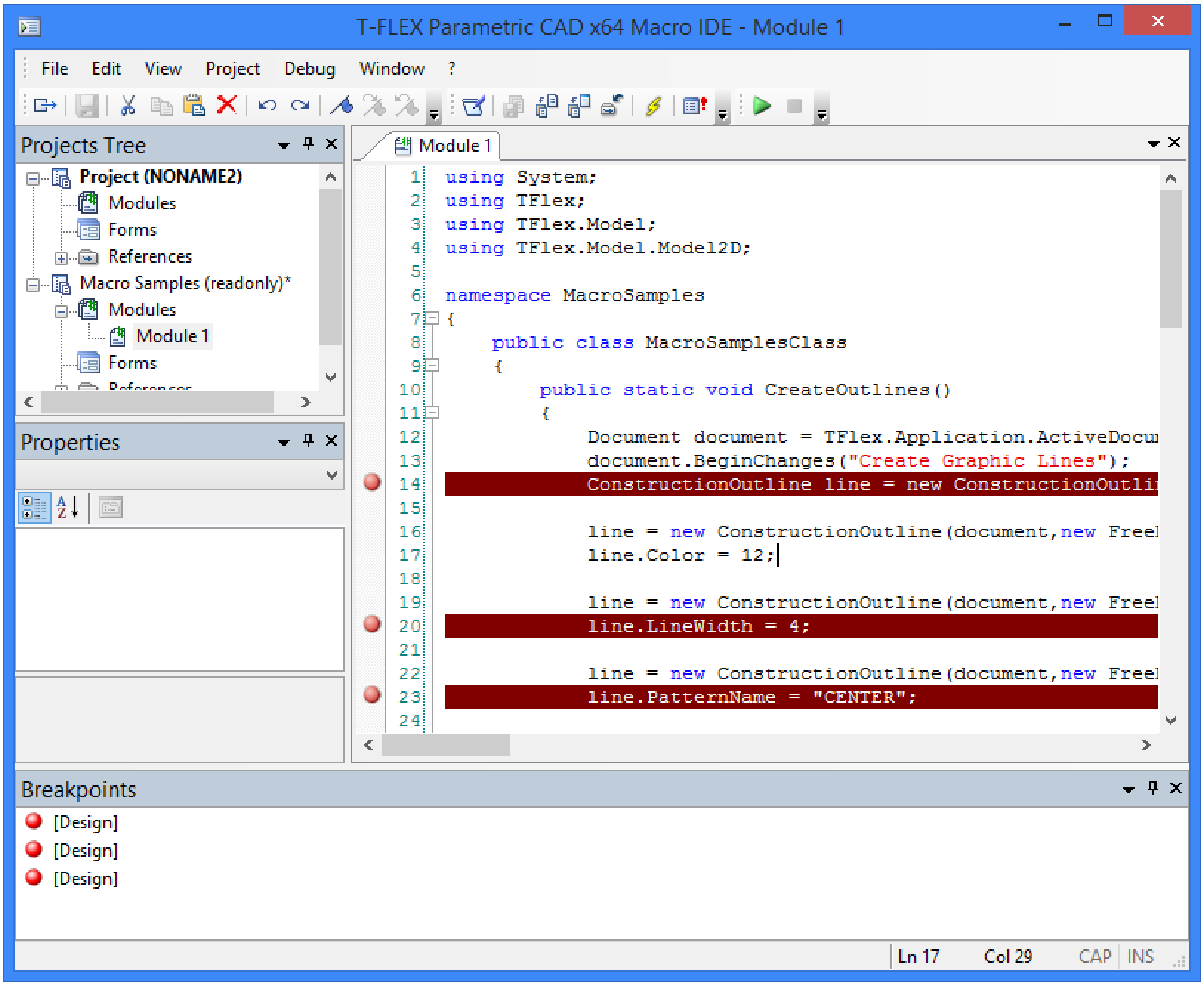
Specifying Breakpoints

To create break points use the command **Toggle Breakpoint**:

Keyboard	Textual Menu	Icon
<F9>	Debug > Breakpoint	

A breakpoint is placed on the line of the code containing the cursor at the moment of starting the command. That is, to create a breakpoint the user has to place the cursor on the line of the code *before* which the execution of macro must be discontinued and press <F9> (or call the command in a different way). To the left of the selected line of the code, in the margin region, a symbol of the breakpoint – will appear; the line itself will be highlighted with a color. To create one more breakpoint, place the cursor on the next line and call again the command **Toggle Breakpoint**, etc.

The list of all selected breakpoints is shown in the window "Breakpoints".



To delete a breakpoint, it is sufficient to place the cursor on the line with the breakpoint and call twice the command – **Toggle Breakpoint**.

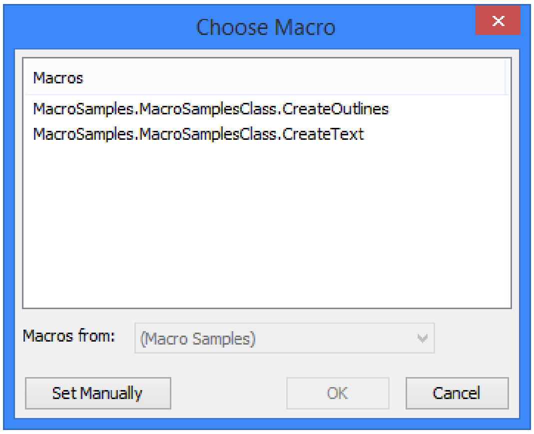
Starting Debug Mode

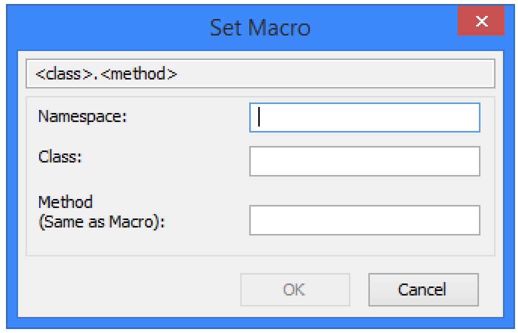
The user can start the mode of debugging a macro with the help of the following command:

Keyboard	Textual Menu	Icon
<F5>	Debug > Start Debug	

Since the module can contain several macros (procedures), the user will need to specify in advance which procedure must be launched. To accomplish that, in the command "Project properties" (before the start of the debug mode) select the default method (macro).

If the default method is not specified in advance, after starting the debug mode, the dialog "Macros" prompting the user to select the method (macro) to be debugged will appear on the screen. The method can be selected either from the displayed list of methods of the current project or entered manually after pressing the button **[Set Manually]**. When the button is pressed, the dialog "Set Macro" is opened in which the user will need to specify the namespace, class and name of the procedure that will be launched.





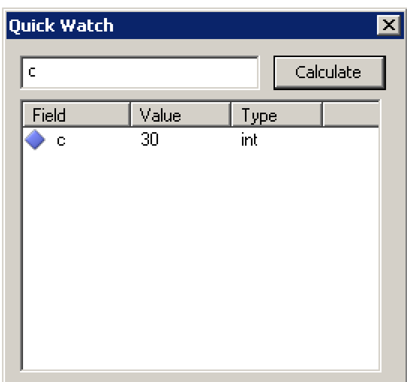
After selection of the method, it will be launched. If, before the launch of the method, the code of the module was modified, the system will prompt the user to recompile the project when the command "Start Debug" is activated.

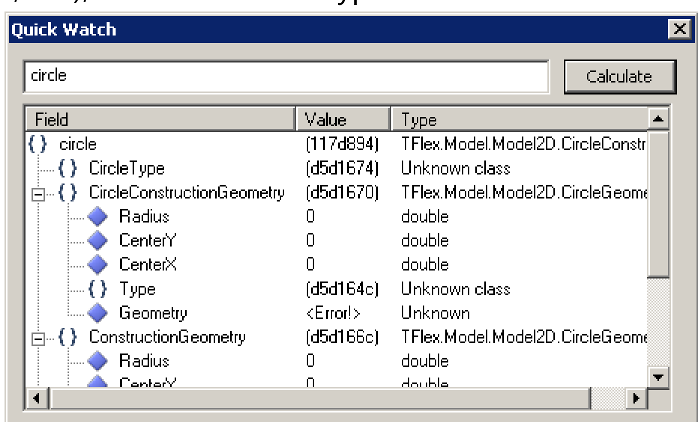
Working in Debug Mode

After the start, the execution of macro reaches the first breakpoint in the code and stops. Execution of macro is passed to the macro editor at the line with the code located *before* the breakpoint. The main window of the T-FLEX CAD becomes inactive. At this moment, you can call the command "**Quick Watch**":

Keyboard	Textual Menu	Icon
<Ctrl><Alt> <Q>	Debug > Quick Watch	

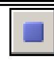
This command allows you to check the value of any object during the stop of macro. The object can be both of the system type (string, integer, real, etc.), and user-defined type.





The user can resume the execution of macro (until the next breakpoint) by calling twice the command "**Start Debug**". For complete execution of macro, this command must be called as many times as the number of breakpoints created in the code.

For quick execution of the macro (without stops at breakpoints) use the command **Stop Debug**:

Keyboard	Textual Menu	Icon
-	Debug > Stop Debug	

Creating Macros with Screen Forms

In certain situations, the user needs to be able to input certain element parameters in the dialog while running a macro. To realize this capability, the macro editor supports creation of screen forms that can contain controls. Follows below is the overview of the macro editor tools supporting creation of macros with screen forms.

Creating Form

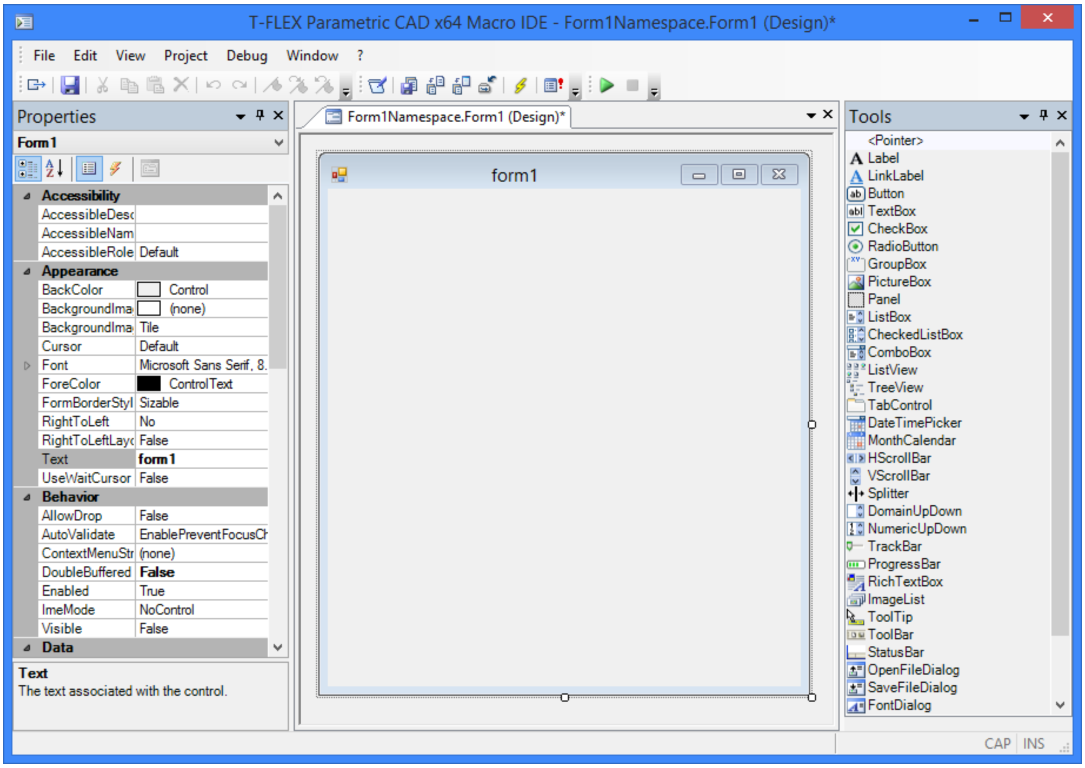
To create a form in the current Project, use the command:

Keyboard	Textual Menu	Icon
-	Project > Add Form	

This command can be also invoked from the window «Project Tree". For that, place the cursor on the title of the section "Forms" in the tree of the current Project and click . In the opened context menu select the command **Add**.

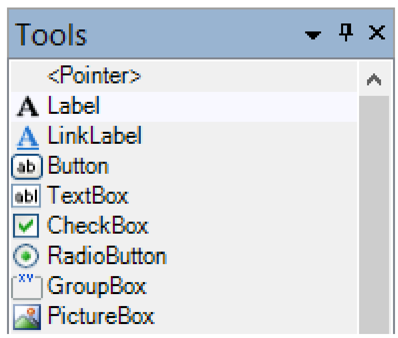
After the new form is added to the Project, in the main window of the macro editor the window of the new form will be opened automatically. It contains a blank template of the form ready to be filled with controls. Title of the window includes a name of the form with the clarification in brackets – "(Design)".

In addition, along with the window of the form, the service windows "Properties" and "Tools" are automatically opened.



When adding a form to the Project, two new references: "Systems.Windows.Form" and "Systems.Drawing" are automatically added to the list of references.

«Tools» Window



The window "Tools" is opened automatically when the new form is created. You can invoke the window "Tools" independently from the text menu of the macro editor: **View** > **Tools**.

The "Tools" window contains a set of control elements that can be placed on a form. The elements appear in this window only when the form is active. Otherwise, the window will be empty.

«Properties» Window

The window can be called via the textual menu of the macro editor: **View** > **Properties**. It serves for viewing and specifying properties and events (methods) of a form and elements placed on the form. This window remains empty until a form or an element located on the form is selected.

Control element properties are the parameters defining an object's characteristics (name, color, position etc.).

Events of control element are the actions performed over the object of a control element, such as clicking a «Button» control element. While a program (macro) is running, a control element event gets bound to executing certain commands. For example, the event of clicking a «Button» control element can be bound to creating a T-FLEX CAD object.

The upper part of the «Properties» window contains the list with form elements. To view and modify properties of a form element, select it in the list or select the element on the screen form itself by .

The «Properties» window works in one of the two modes:

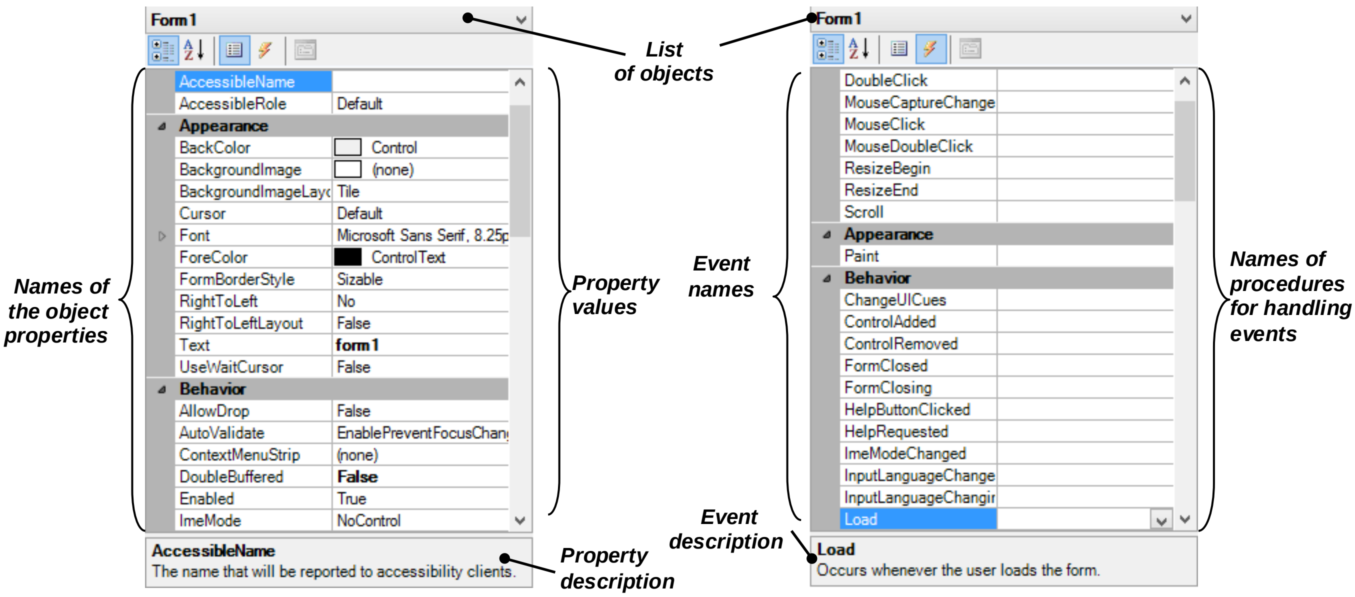
If the «Properties» option is active on the window toolbar, then the window will be displaying element properties.

If the «Events» option is active, then the window displays events.

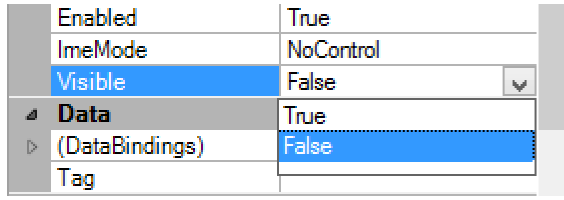
The «Properties» window is divided into two parts. The left column displays the names of the object properties. The right column displays property values.

If the window is in the mode of showing events, then the right hand side column of the window displays the events attainable for the given element type. The left-hand side column shows the names of the functions that are currently used for the given element and event.

The buttons and allow sorting the list of object properties alphabetically and by categories, respectively.

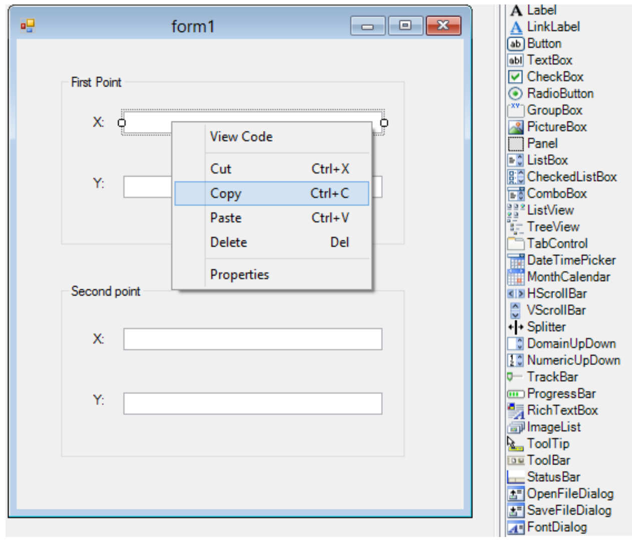


For some properties, you can specify only strictly defined values. In such a case, the value input field will be represented by a combo box; when accessed, this combo box displays the property values available for selection.



Placing Control Elements on Form. Specifying Parameters of Control Elements

To place control elements on a form, you need to complete the following steps:



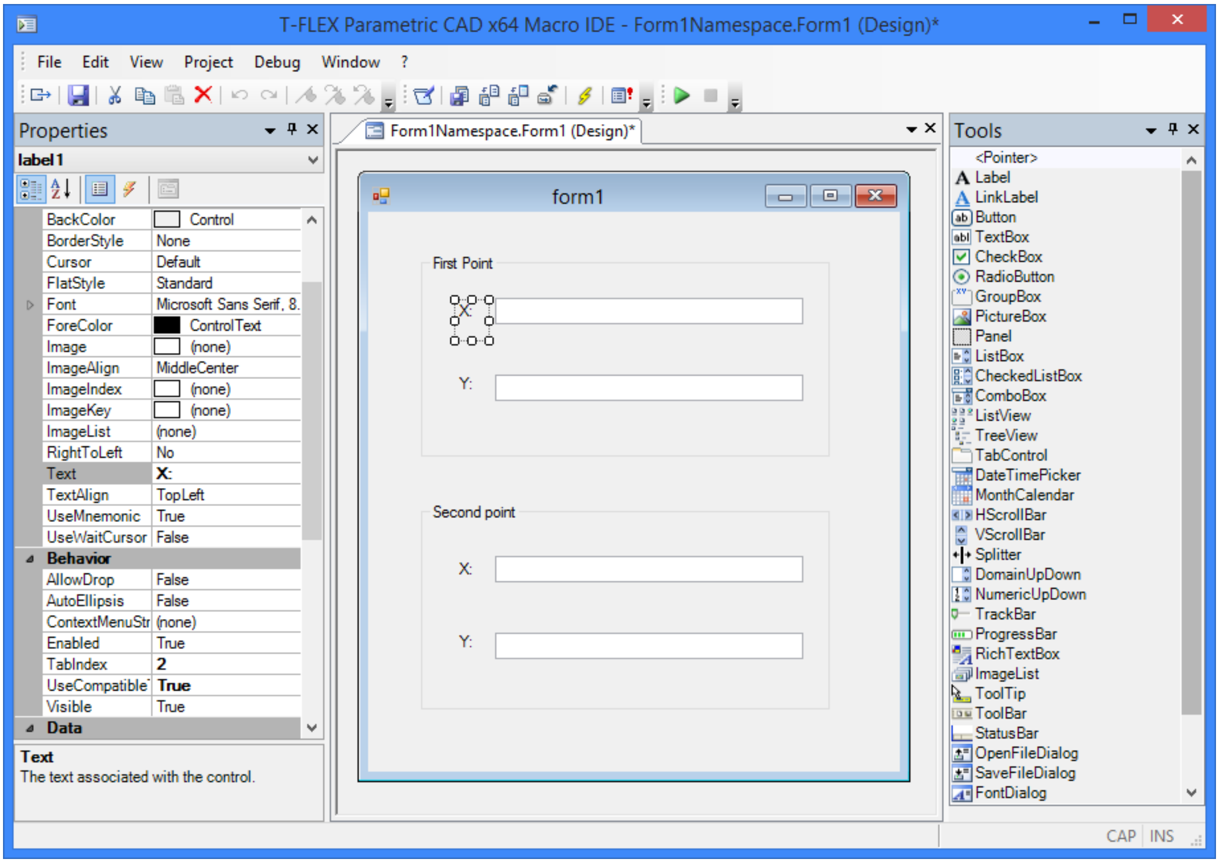
Select the form to which you need to add a control.

Select the desired element in the “Tools” window.

Next, you can choose any of two options. The first option – click on any point of the form. After that, a selected control element of standard size will be added to the form. Then, you can move the control to the desired place on the form and modify its dimensions.

The second option – by clicking twice indicate location of opposite angles on the diagonal of the rectangle of control element. In this way, the location and dimensions of the control are defined right away.

The user can also specify parameters of the controls placed on the form with the help of the window “Properties”. The user can select the element to be edited directly in this window (in the drop down list at the top) or in the form with the help of . After selection of the element in the window “Properties”, parameters of the element will be displayed.



If necessary, you can view automatically generated code of creation of form's graphic image. This can be done with the help of the command **View Design (Code)** found in the context menu of the given form in the window «Project Tree”. The window with the form's design code is also opened in the main window of the macro editor. The header of the window includes the name of the form with the clarification in brackets – “(Design)”.

> The name of the window with the form's design code coincides with the name of the window of form. They have different icons:  – for the window of form,  – for the window of the form's design code.

After placing all necessary elements of the form, you can start creating the code of the form.

Writing Procedures for Control Elements on Forms

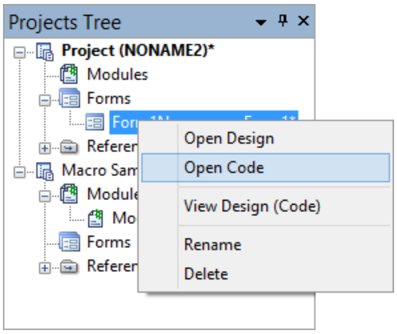
After placing control elements on forms, you need to bind those elements to the executable code.

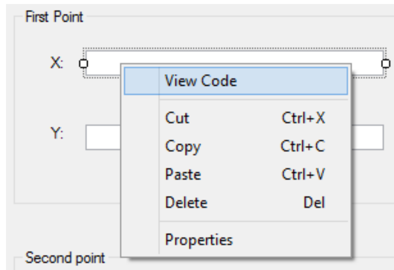
The code of the form is created in a separate window similar to the window of module's code. The header of the window includes the name of the form with the clarification in brackets – “(Code)”.

The user can invoke this window in two ways:

In the “Projects Tree” window, call the command “Open Code” from the context menu of the selected form. Doing so opens the window containing executable code for the form, so that you can add procedures for processing control element events in this form.

Call the command “Open Code” from the context menu available for any control element of the form.





Double-click the control element on the form. This brings up a window and automatically creates a procedure there for processing the driving method of the respective element. Default events are defined for each control element.

You can switch between the form window and the code window either by the context menu commands of this form (“Open Design”, “Open Code”), or with the help of the respective tabs located above the code window.

The window of the form's code, as the windows of the modules' code, is a text editor. By default, the window of the form's code contains only one procedure of form initialization.

The form's code usually consists of form initialization procedure (it is created automatically when you open the window of the form's code for the first time) and procedures for handling events of control elements of the form.

To create a procedure of handling the event for an object of the form, the user needs to select a required element, activate the mode “Events” for the window “Properties”, select a desired event in the list and double click . A blank procedure of handling the event for the given element of the form will be automatically added to the window of the code. The body of the procedure is blank, it is filled by the user.

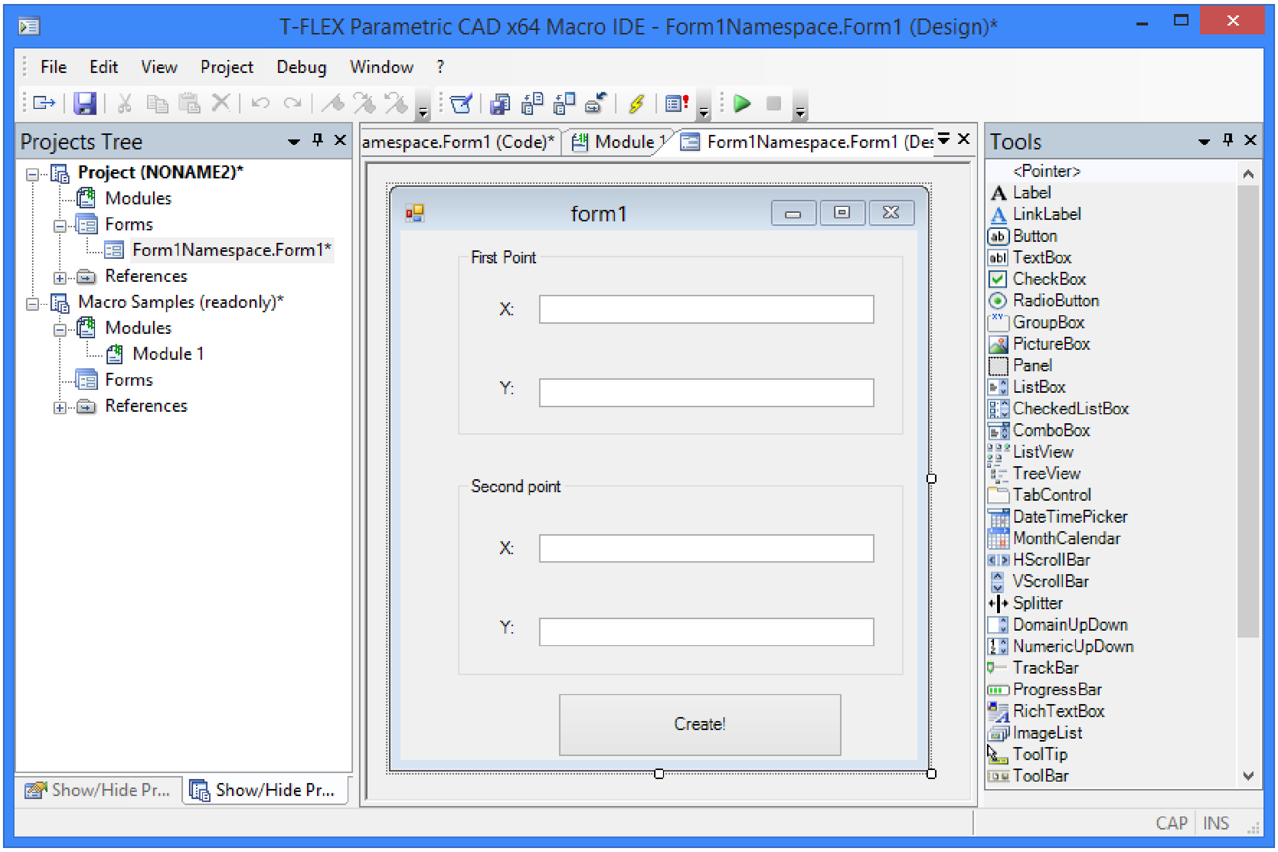
For some control elements you can also use double clicking on the element of the form. After that, the procedure of handling the main event of the given control element is added to the form's code.

Example of Macro with Screen Form

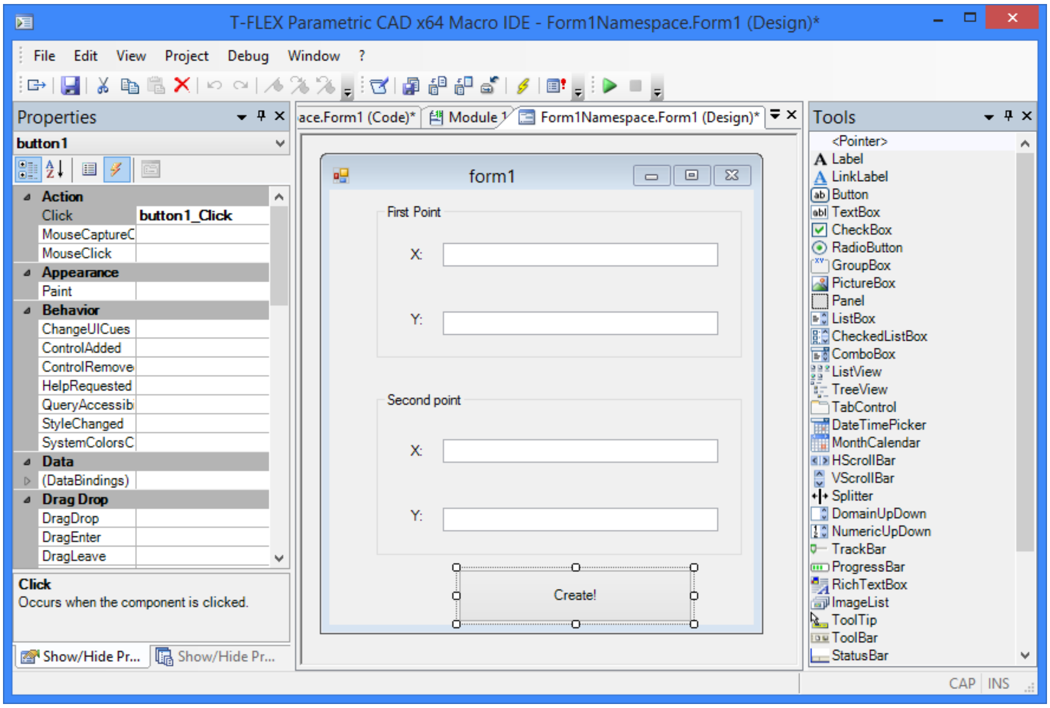
Let’s review an example of a macro written in the Visual Basic programming language. Executing the macro creates a segment between two 2D nodes. Coordinates of those nodes are entered via a dialog.

Initially, a form “LineForm” was created in the Project “CreateLine.grb”, and then controls were placed on it, which will be used for defining 2D nodes coordinates. Another control element was added to the form

– a button, clicking which would launch the function creating the 2D nodes and the segment between them.

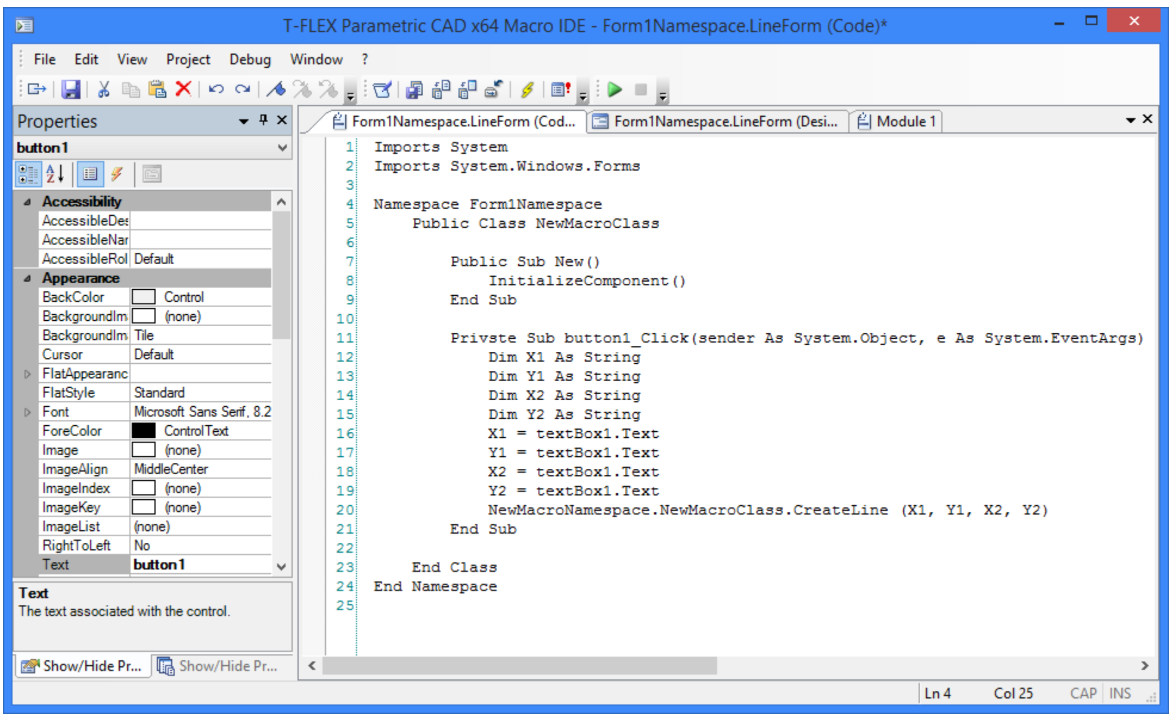


Upon placing control elements on the form, one needs to define processing of the event of clicking the control element – the button. To do this, select that element on the form, switch to the “Properties” window and set the latter window into the event mode . In the right hand side column of the “Click” event, press , which will create the procedure “button1_Click”. The same result could be achieved by double-clicking the control element – the button.



In the coming up form code window, one needs to write the procedure “button1_Click”.

The figure below depicts the form code window containing the function “button1_Click”, in which the variables “X1”, “Y1”, “X2” and “Y2” are assigned values from control elements – the textBox1, textBox2, textBox3 and textBox4 input boxes, and “CreateLine” function (macro) is called.

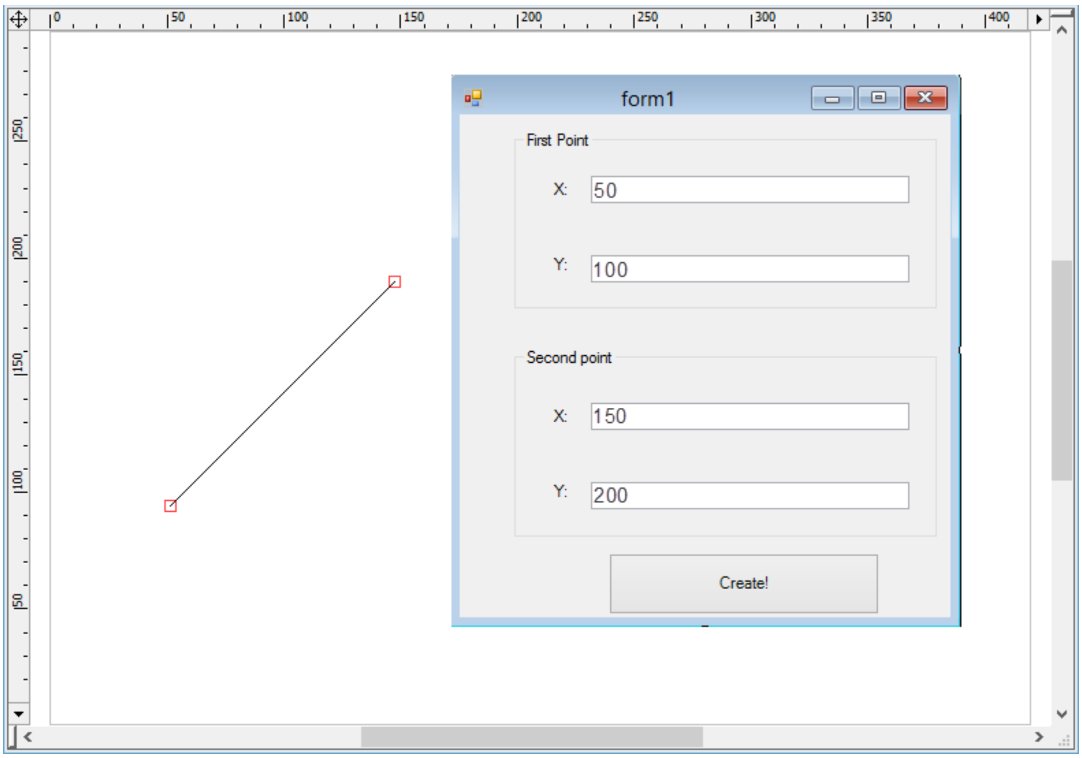


Next, create a module in the Project. The module's code will consists of two functions. The function "*ShowDialog*" (a standard function of the development environment) will be displaying the screen form upon launching the macro. The previously defined function "CreateLine" creates 2D nodes, whose coordinates are the values specified by the user via the "FormLine" form dialog, and the segment between those nodes. The module's source code is shown below.

```
Declaring references
Imports System
Imports TFlex
Imports TFlex.Model
Imports TFlex.Model.Model2D

'Declaring namespace
  Namespace NewMacroNamespace

'Declaring class
  Public Class NewMacroClass

'The function, which will display the screen form «form» when executed
    Public Shared Sub ShowDialog()
       Dim form As Form1Namespace.LineForm
       form = new Form1Namespace.LineForm()
       form.ShowDialog()
    End Sub

'The function with parameters (the macro), which creates a graphic line between two 2D
'nodes. Coordinates of those nodes are input in the function as the dialog parameters
    Public Shared Sub CreateLine(ByVal NodeX1 As String, ByVal NodeY1 As String, ByVal
                                        NodeX2 As String, ByVal NodeY2 As String)
       Dim document As Document
       document = TFlex.Application.ActiveDocument
'Opening block of documents changes
       document.BeginChanges("Creating graphic lines ")

'Creating graphic line and 2D free node objects
       Dim line As ConstructionOutline
       Dim node1 As FreeNode
       Dim node2 As FreeNode
       Dim X_1, Y_1, X_2, Y_2 As Double
```

```
    X_1 = System.Convert.ToDouble(NodeX1)
    Y_1 = System.Convert.ToDouble(NodeY1)
    X_2 = System.Convert.ToDouble(NodeX2)
    Y_2 = System.Convert.ToDouble(NodeY2)

'Creating 2D free nodes
    node1 = new FreeNode(document,new Parameter(X_1),new Parameter(Y_1 ))
    node2 = new FreeNode(document,new Parameter(X_2),new Parameter(Y_2 ))
'Creating graphic line between two nodes
    line = new ConstructionOutline(document,node1,node2)
'Closing block of document changes
    document.EndChanges()
  End Sub
 End Class
 End Namespace
```

When the macro is executed in T-FLEX CAD, the "LineForm" dialog appears. Upon clicking the control element – the "OK" button, a segment will be created between the two nodes in the drawing area.



HANDLING EVENTS OF DOCUMENTS WITH THE HELP OF MACROS

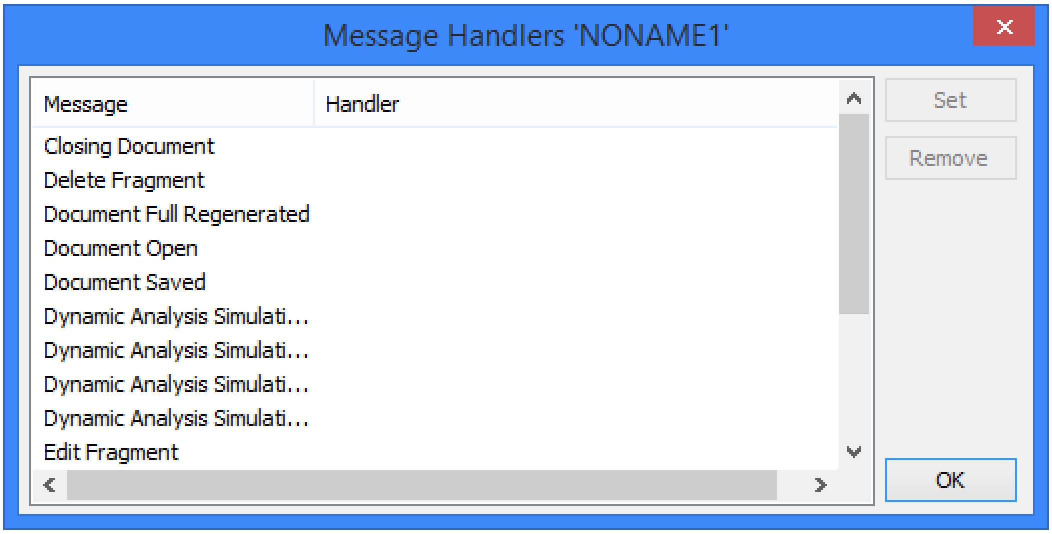
Macros can be executed not only from commands of the user but also upon the start of some event of the document. To define relation of the events of the document with the macros you can use the dialog of the command "**Message Handlers…**":

Keyboard	Textual Menu	Icon
-	Project > Message Handlers…	

When calling this command, the dialog "Message Handlers" is opened. In this dialog there are events the start of which can be related to the execution of macro:

Closing Document – event arising before closing document;

Document Full Regenerated – event arising after document regeneration;



Document Open – event arising after opening document;

Document Saved – event arising after saving document;

Dynamic Analysis Simulation After Step – event arising after executing a step in solution of the dynamic analysis problem;

Dynamic Analysis Simulation Before Step – event arising before executing a step in solution of the dynamic analysis problem;

Dynamic Analysis Simulation Finished – event arising after completing solution of the dynamic analysis problem;

Dynamic Analysis Simulation Started – event arising after starting solution of the dynamic analysis problem;

New Document Created – event arising after creation of new document;

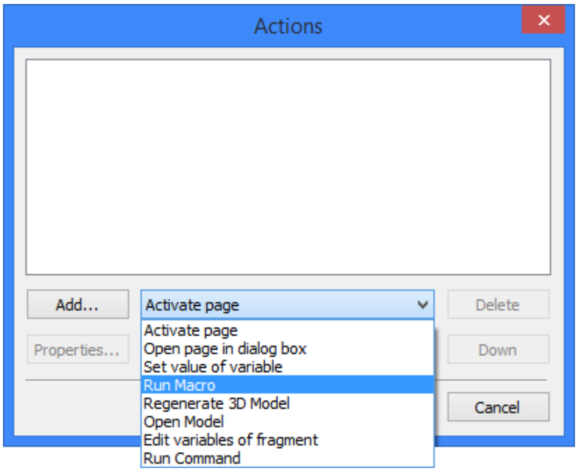
Saving Document – event arising before saving document;

View Activated – event arising after activating 2D or 3D window of the document;

View Deactivated – event arising after deactivating the image of document.

To specify the handler of some event, the user needs to select the desired event in the list and press the button **[Set]** or double click . The dialog with the list of macros of the given document will appear on the screen.

STARTING MACRO FROM USER-DEFINED DIALOG



As it was mentioned earlier, the macro can be launched from the window “Macros”, from the macro editor in the debug mode or automatically upon the start of some event of the document. However, there is one more way to launch a macro. It can be done from the user-defined dialog of the document if the function of launching the macro is appointed to the control element “Button”.

More detailed information on actions of control element “Button” can be found in chapter “Controls. Creating user-defined dialogs”.

Converting Documents Created in Earlier Versions of T-FLEX CAD

The newer versions of T-FLEX CAD use different data format than the older ones. If this is the case when opening an old file in a newer version, full model regeneration is recommended for saving information in a new format.

When working with assemblies, saving just the assembly document is not enough. In this case, you need to subsequently open and resave in the current T-FLEX CAD version all model documents, starting with "bottom most" fragments and finishing with the top level assembly document. This task might be possible to do manually for a small model. However, it becomes quite difficult for very large assemblies with deep fragment nesting.

The "Old version documents converter" helps solve this problem. This command saves the whole active assembly in the format of the current T-FLEX CAD version. When converting, the whole model structure is analyzed. Conversion is performed according to the fragments hierarchy: the bottom level fragments are opened and regenerated first, then the next level is handled, and so on up to the top assembly document. The user can manually add any T-FLEX CAD documents to the list of files subject to the conversion. Those files will also be converted according to their hierarchy.

A report file is created as the conversion progresses, reflecting on the performed tasks (warnings, information messages, errors). The original files can be saved as backup copies.

Using the Application "Old Version Documents Converter"

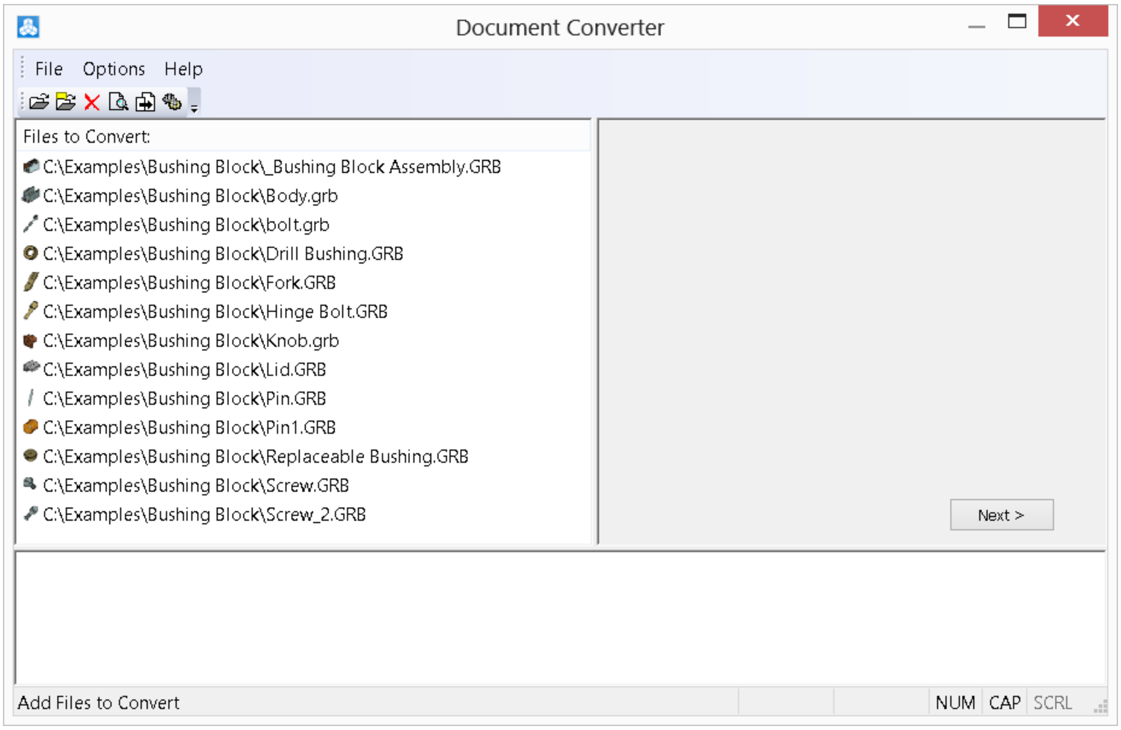
Startup of Documents Converter

The converter is called just like any other system command:

Icon	Ribbon
	File → Document Converter
Keyboard	**Textual Menu**
<AC>	File > Document Converter

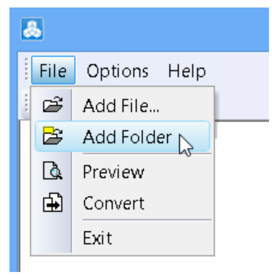
After calling this command, the converter window will appear on the screen. It is divided into 3 areas:

- Area with the list of files (upper left);
- Area with the service information and buttons (upper right);
- Area of diagnostics and display of messages about the progress on the conversion process (the lower part of window).



Specifying List of Converted Documents

The list of files selected for conversion is shown in the upper left field of the converter dialog. By default, the list of files is empty.



If some documents were open in T-FLEX CAD at the time of calling the command, the system will prompt you for closing them (this is required for successful converter operation) and will automatically add them to the list of files to be converted. If necessary, the user can remove some files from the list and/or add other T-FLEX CAD documents to the list.

To add files to the list, the buttons  ("Add files") and  ("Add folder") found on the toolbar of the converter can be used. These commands can be also called from the textual menu.

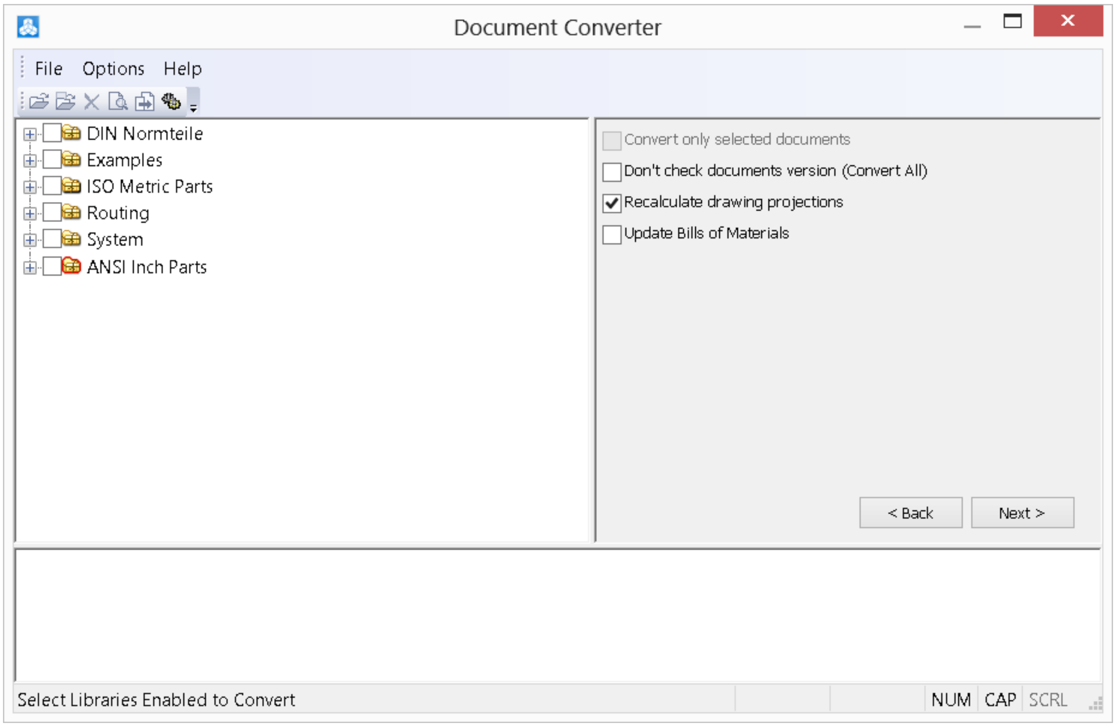
The first button allows adding separate documents to the list, while the second one – all T-FLEX CAD documents in the selected folder. If the selected folder has subfolders, the system will ask you, whether to add those to the list.

To delete a file from the list, select it in the list and press the button  ("Delete Files from List") or the button <Del>. To complete creation of the file list for conversion, press the graphic button **[Next>]**.

In the progress of conversion, the system may automatically decide to add to the conversion list the necessary fragment files of the assemblies being converted, upon analyzing the structure of the model to be converted. This behavior can be controlled by a conversion parameter (see section “Customization of Converter”).

Performing Conversion

After forming the list of converted files, press the button **[Next>]** in the right part of the converter window. After pressing this button, the contents of the upper left area of the converter window will be modified. The list of the converted files will be replaced by the list of the T-FLEX CAD libraries. At this stage, a user has to indicate the elements of which libraries should be subject to transformation, if they are found in the structure of the converted documents. The libraries the elements of which can be converted are marked with a “tick” in the list of the libraries (to mark it, point with the cursor at the symbol ☐ next to the name of the library and press 🖱).

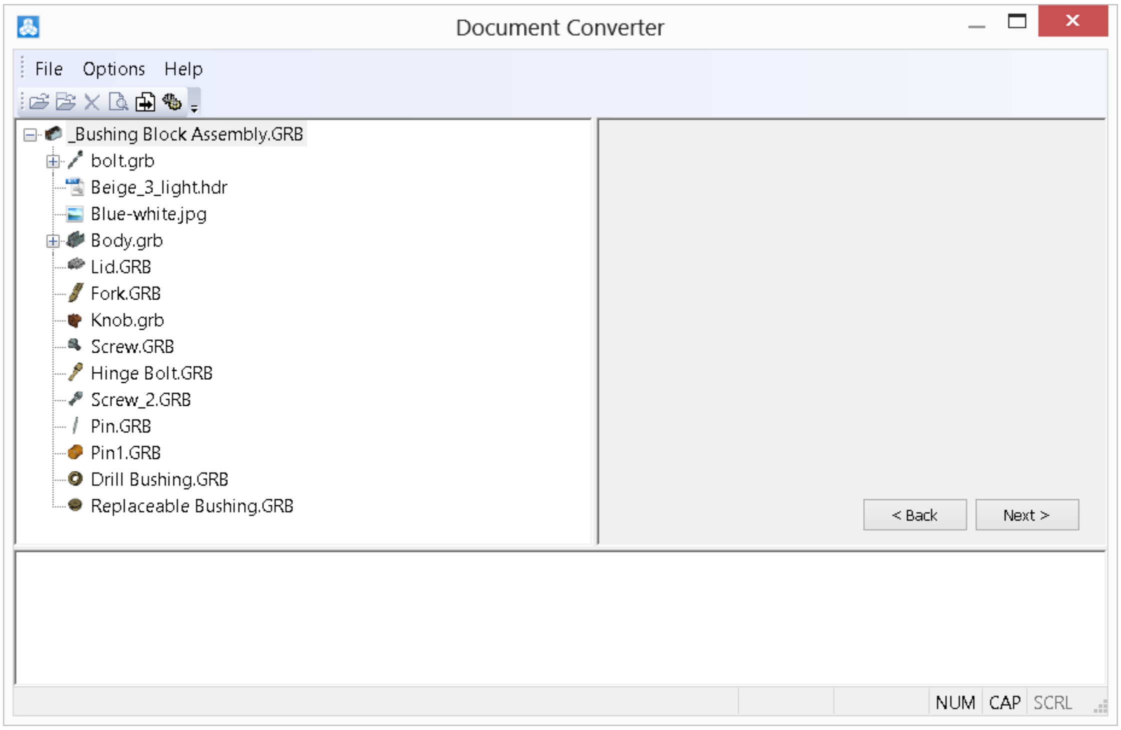


Parameters shown at this step in the right part of the converter window duplicate parameters in the conversion options dialog (see section “Customization of Converter”).

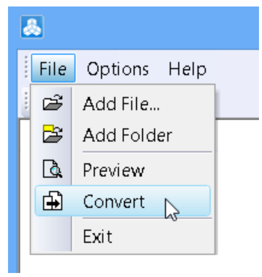
After specifying libraries the elements of which can be modified upon conversion, press the button **[Next>]** again to proceed to the next step.

To return to the step of selecting documents (for example, if you need to modify the list of selected files), use the graphic button **[<Back]**.

At the next step, the structure of relations of converted files will be shown in the upper left area of the converter window. If necessary, additional files are added to the list, that correspond to the fragments of the assemblies being converted.

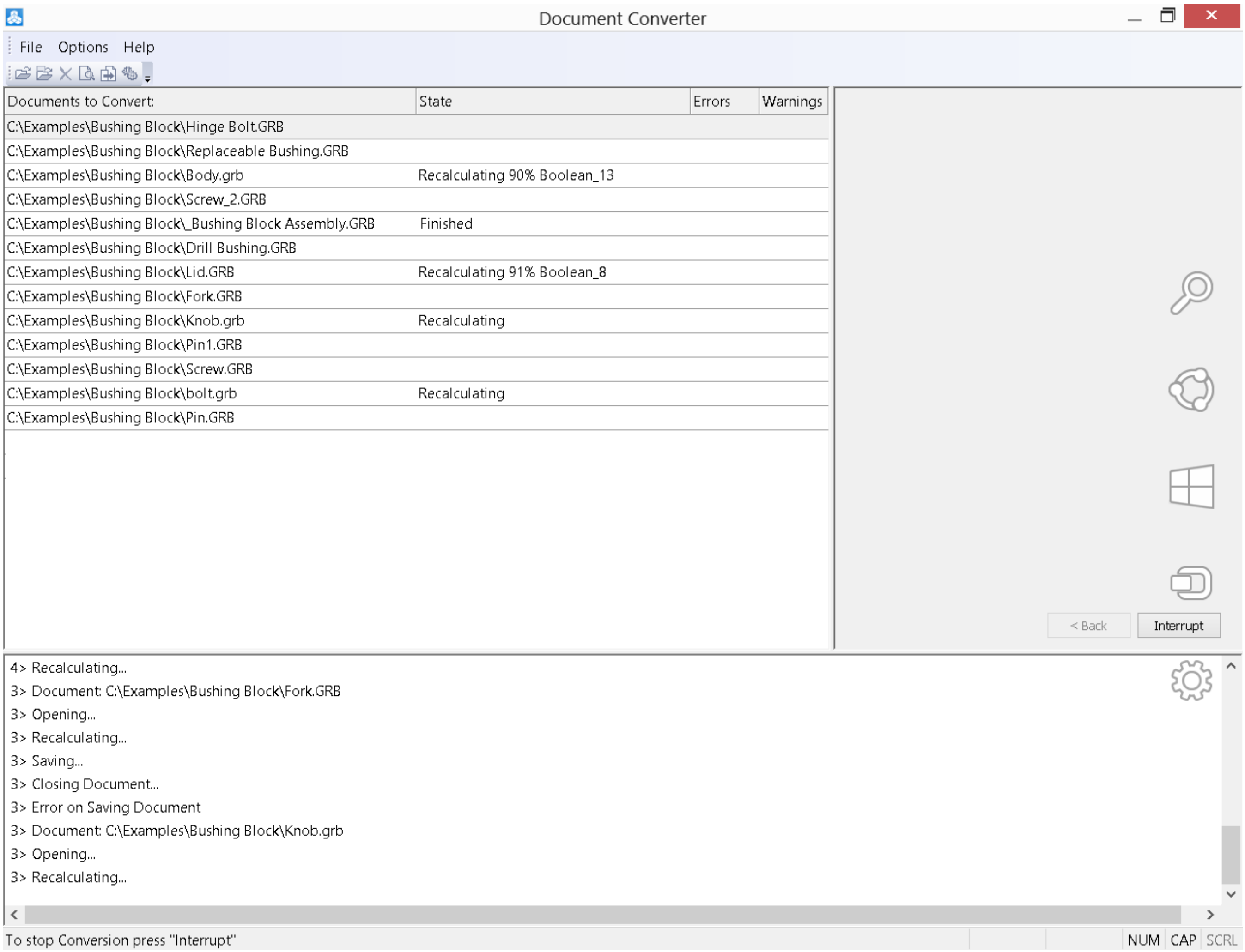


To start regeneration of selected files, press the button **[Next>]** again.



The step of library selection and viewing the document connections structure can be skipped, if for proceeding to the next step of the conversion process, a user does not press the button **[Next>]**, but rather calls the command  (**Preview**) and  (**Convert**) found on the toolbar or in the textual menu of the converter. In this case, for the library elements the default settings will be used (by default, conversion is allowed only for user-defined libraries; standard libraries supplied with the system are not converted).

In the process of regeneration of selected files, the information on the progress and results of the converter actions will be displayed in the lower part of the converter window, whereas in the upper left area a user can view again the list of the files being converted in which the current state of working with each file is shown.



After completing the file transformation, in the lower part of the converter window, the message about the process completion and indicating the number of occurred errors will be displayed.

To close the application press the graphic button **[Exit]**.

Customization of Converter

Dialog of options of the conversion process can be invoked with the help of the button  on the toolbar of the converter or from the textual menu **Options** > **Options...**.

The group of parameters **When Error Occurred** defines the system response on the occurrence of errors during regeneration or saving of the documents:

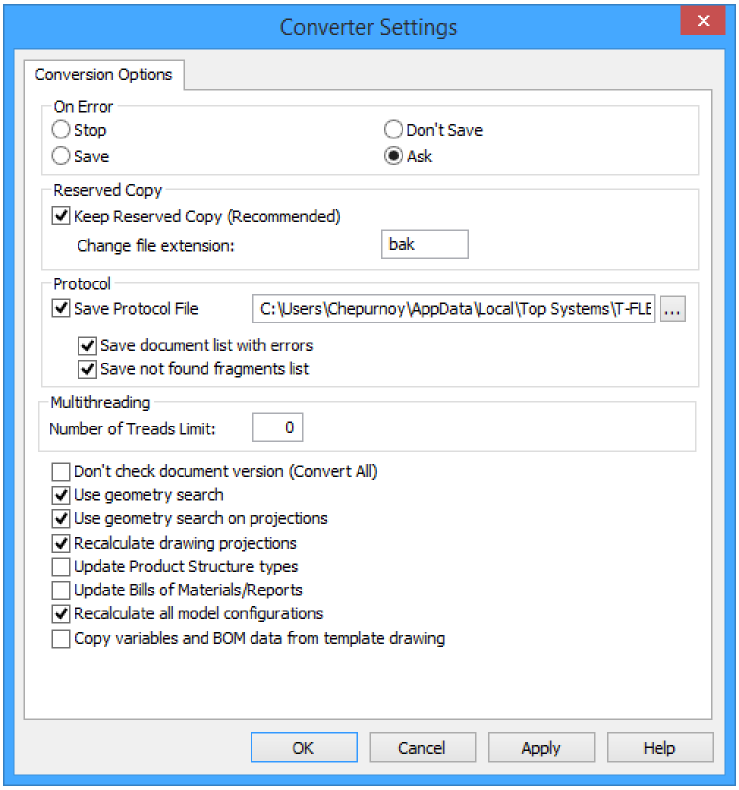
Stop – stop the conversion;

Save – saved the document despite the errors;

Do Not Save – do not save a document if errors occurred during its regeneration;

Ask – display a dialog box with the error message and ask the user about further actions.

The group of parameters **Backup Copy** allows turning on the mode of creating backup copies of all the files with the specified extension.



The group of parameters **Log** is provided for controlling the information displayed in the lower part of the converter window and in the protocol file.

The flag **Save Log** allows saving the information about the conversion process in the protocol file. The name and the path of the protocol file can be specified to the right of this flag. If this flag is disabled, the protocol file is not saved.

The parameters **Output List of Documents with Errors** and **Output List of not found Fragments** control the output of certain information into the log.

The group **Multithreading** has only one parameter **Number of Treads Limit:**. It allows a user to limit the maximum number of simultaneously started processes upon the conversion.

Upon conversion, the T-FLEX starts several processes in which simultaneous regeneration of not depending on each other files takes place (if they do not have relationship parent/child in the assembly). While doing it, by default, to achieve the maximum effectiveness, the number of started processes is equal to the number of processors on the computer (this allows engaging all resources of the computer). This parameter allows put a limit on the number of processes and the computer resources used by the processes.

Don't Check Documents Versions (Resave All). If not set, the documents in the up to date version of T-FLEX CAD format will not be converted (except the assemblies, whose fragments were converted). If set, all files will be converted, regardless of the format.

The following group of parameters is used only when converting documents containing 3D models:

Use geometry search. With this parameter turned on, if errors occur during the model element regeneration (for example, an error regenerating a 3D node created at a body vertex, or an error regenerating an edge or face blend), the geometry search procedure is launched, that allows restoring the lost geometry references.

> Use of the mode "Use geometry search" is strongly recommended with the files of the versions earlier then 7.2. However, in some rare cases, use of this procedure may result in establishing incorrect references within the model.

Use geometry search on projections. This parameter turns on handling of the regeneration errors that occur in the elements based on 2D projection lines. The error handling procedure restores references of these elements.

Regenerate drawing projections. With this parameter set, all 2D projections are regenerated regardless of the projection parameter setting "Update". Regeneration of projections and sections of complex models may take considerable time, therefore it should be unchecked if desired.

Update BOMs. This parameter controls recalculation of all BOMs of the document and also updating tables of databases. By default it is disabled.

Recalculate All Model Configurations. If this parameter is set, then the configurations present in the document being converted are resaved when converted. This parameter has higher priority than the respective parameter in the document being converted (command **ST: Set Document Parameters** tab **Save** parameter **Model Configuration Regeneration on Save** item **don't regenerate model onfigurations.**

To return to the step of selecting documents for conversion (for example, if you need to modify the list of selected files), use the graphic button **[<Back]**. Pressing the button **[Finish]** launches the process of the document conversion.

The application analyzes the structure of the specified files and creates a tree of dependencies for the documents. If necessary, additional files are added to the list, that correspond to the fragments of the assemblies being converted.

Thereafter, the files are converted according to the determined hierarchy. As the file conversion progresses, the log is displayed in the application window, reflecting on the accomplished tasks. Upon the completion of the file conversion, the message is displayed in the application window about finalizing the process, showing the number of the encountered errors. To close the application, once again press the graphic button **[Finish]**.

Recommended Order of Steps when Converting Models from Older Versions of T-FLEX CAD

To insure successful conversion of the models from earlier T-FLEX CAD versions into the current version format, the following order of steps is recommended:

1. Make sure that the full model regeneration completes without errors in the previous T-FLEX CAD version, while opening the assembly – without messages about not found fragment files.
2. Make a backup copy of the assembly.
3. Start the converter with default settings (the modes “Use geometry search”, “Regenerate drawing projections” and “Use geometry search on projections” turned on).
4. If problems were encountered during the step 3, such as not found model elements – not found edges, faces, etc., or the conversion took too long time, then you may want to restart the converter with the unset flags “Use geometry search on projections” and “Use geometry search”.
5. When converting crowded drawings of extra-large assemblies on the computers with limited performance resources, an out of memory error may occur. In this case, you can try to convert the model with the drawing projection regeneration turned off (postponing this to a later manual handling).
6. When converting crowded drawings of extra-large assemblies on the computers with limited performance resources, an out of memory error may occur. In this case, you can try to convert the model with the drawing projection regeneration turned off (postponing this to a later manual handling).