t-flex

CAD/CAM/CAE/CAPP/PDM

T-FLEX CAD

Brief Introductory Course

Топ Системы

T-FLEX PARAMETRIC CAD

BRIEF INTRODUCTORY COURSE

«Top Systems» Moscow, 2008

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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:

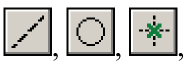
<Enter>, ***<L>***, ***<Esc>***, etc. – notations for the keys on the computer keyboard.

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

 - Left mouse button click.

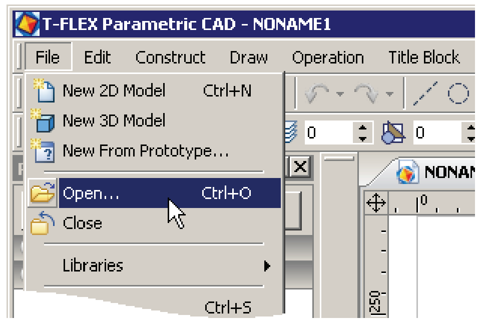
 - Right mouse button click.

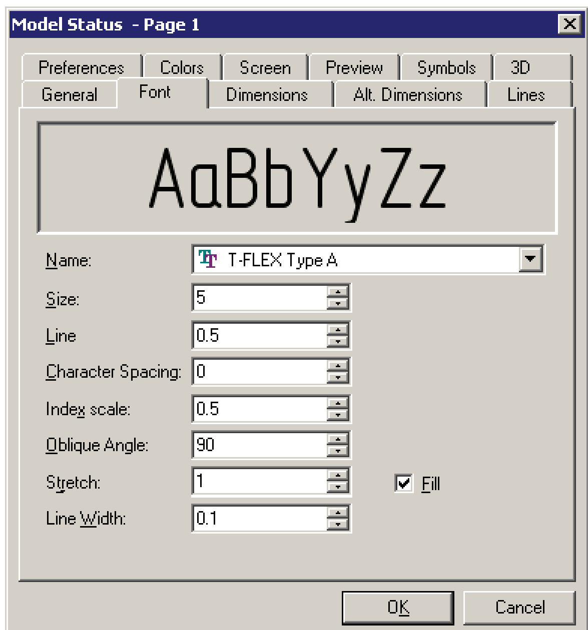
 - Left mouse button double-click.

, etc. – icons on a toolbar or on an icon automenu.

"File|Open…" etc. – selection of a textual menu bar item **"File"**, followed by a pull-down menu item **"Open…"**.

"Font|Name" etc. – selection of a tab **"Font"** in a dialog box followed by an item **"Name"**, or selection of a group of parameters in a dialog box followed by a particular parameter.





"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:

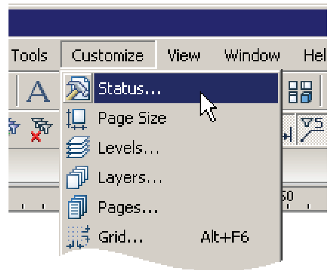
By typing,

By selecting the toolbar item, and

By selecting the textual menu item.

The system manuals describe the commands in a table. For instance, the command **"ST: Set Model Status"** would appear in a table as follows,

Keyboard input	Textual menu	Icon
<ST>	**"Customize|Status…"**	

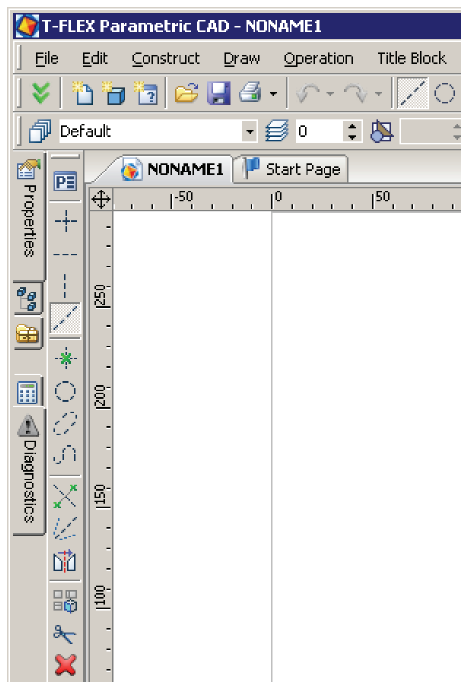


This means, the command can be invoked in the following ways:

Press the key ***<S>*** then ***<T>*** on the keyboard, or – select the entry **"Customize"** in the menu bar, then select **"Status…"** from the pull-down menu, or – select the icon in the appropriate toolbar.

> 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>***.



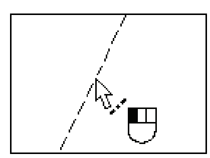
Select an icon, press an icon, select an input box, press a button instruction means placing the cursor over the item (icon, input box, dialog box button) and pressing left mouse button.

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:

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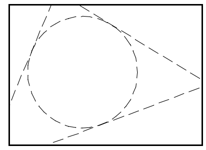


<L>, ***<L>***, ***<L>*** - a circle tangent to three lines.

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 a left mouse click 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).

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

Computer:	PC with USB port
Processor:	Pentium IV or compatible
Minimum hard disk size:	300 MB
Minimum RAM: Recommended RAM:	1G 2G и больше* (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.

Software Requirements

Operating system:	Windows 2000/XP/Vista

T-FLEX CAD System Setup

Running Installation

T-FLEX CAD is distributed on a CD ROM disk. To begin installation, insert the T-FLEX CAD disk into the CD ROM drive of your PC and run the ***SETUP*** from the root folder of the CD.

Then follow the wizard prompts and input desired settings.

Installing Hardware Protection Key

T-FLEX CAD is distributed with a hardware protection key (HASP). To run the system, connect the HASP key to USB port of your PC.

The driver for the HASP key is installed automatically as part of the T-FLEX CAD installation process.

Shall any problems occur with this driver, it can be re-installed separately, with custom settings specified as necessary. To do this, run ***haspdinst.exe*** from the ***name\PROGRAM\Hinstall*** folder, where ***name*** is the installation folder of the T-FLEX CAD. To get detailed instructions of the program usage, run it with the option ***/help***.

Running ***haspdinst.exe*** requires administrator privileges.

The driver is not needed when using a network protection key.

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:

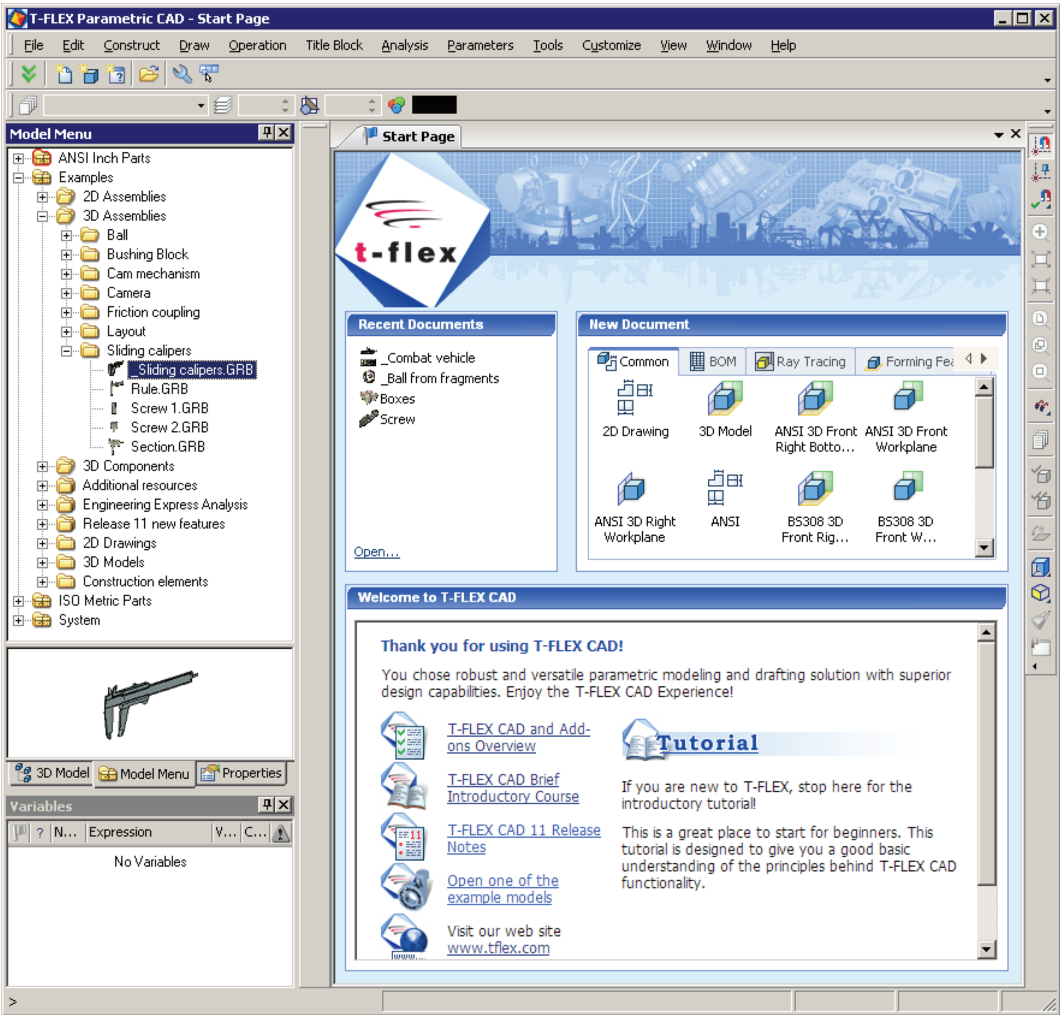
`\TFLEX\`

`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 installation the dialog box **"Start Page"** opens up in the T-FLEX CAD window. 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 **"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", "Forming Feature", "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").

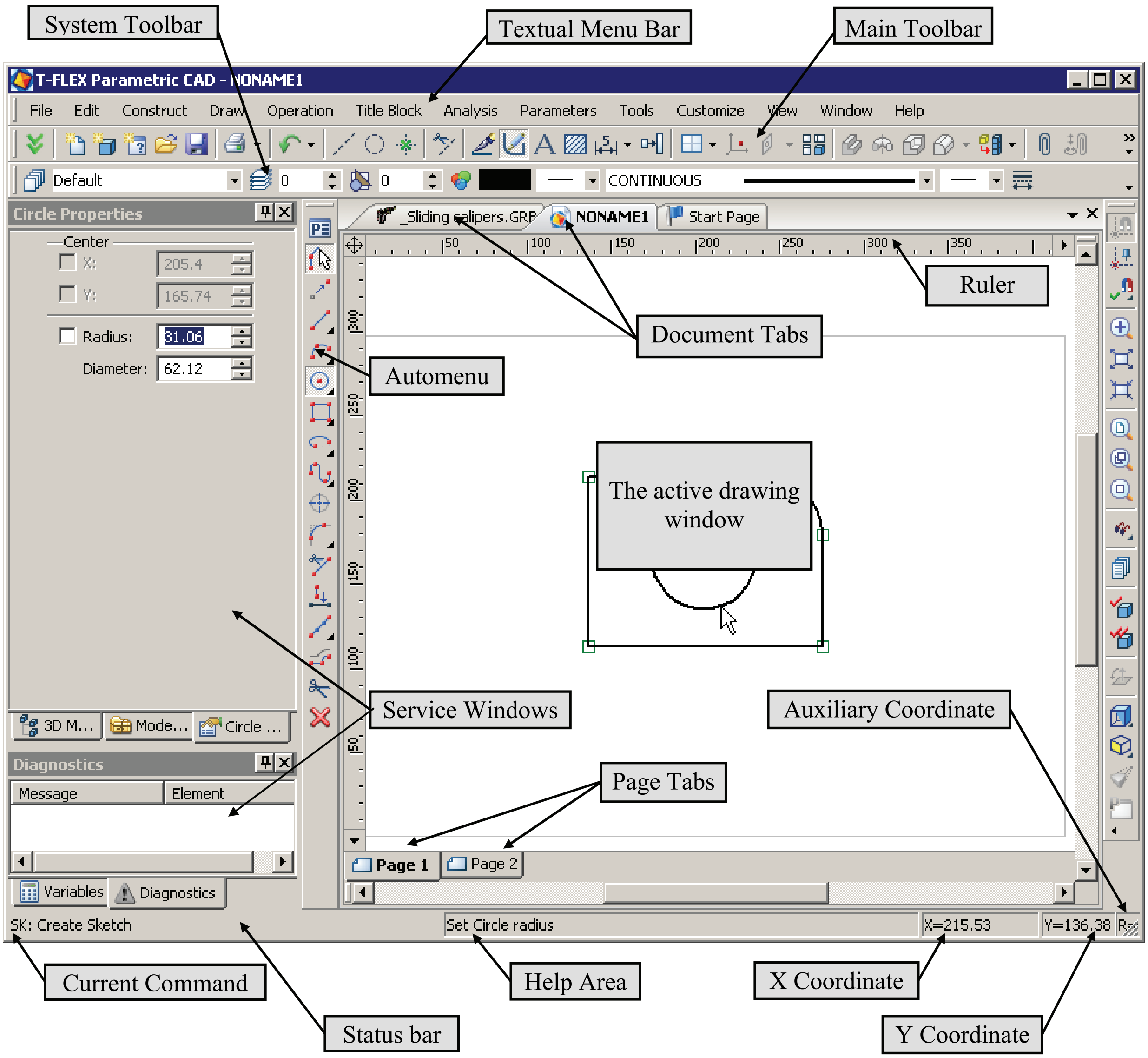
The last chapter – **"Welcome to T-FLEX CAD"** – contains various useful links related to working with T-FLEX CAD.



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 view of this dialog box can be controlled by using the flag **"Customize|Tool Windows|Start Page"**. This flag is active during one T-FLEX CAD session, i.e. if the flag is disactivated, the dialog box "Start Page" will not be shown in the current session, but upon the next start of the T-FLEX CAD the dialog box will be shown on the screen again. Control of the view 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 Start Page on Start” on the tab “Preferences”).

In addition to the dialog box “Start Page”, the main window of the T-FLEX CAD contains different service windows and elements of control used in working with the system.

Service Windows and Elements of Control of T-FLEX CAD



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…**”. Alternatively, click at the automenu or one of the toolbars with the right mouse button.

Elements of Control

The active drawing window	The graphics window for displaying the drawing. Drawings can only be created and edited in this window.
Ruler	Indicates current X and Y coordinates in the active drawing.
Automenu	A menu of icon buttons for the options available within the current command. If no command is current, the automenu is empty.
Main Toolbar	Contain 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.
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.
Textual Menu Bar	Contains the textual menu of the T-FLEX CAD commands by groups.
The 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.
Page 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.
Document Tabs	Help quick navigation through the open documents. To activate a document, select the respective tab.

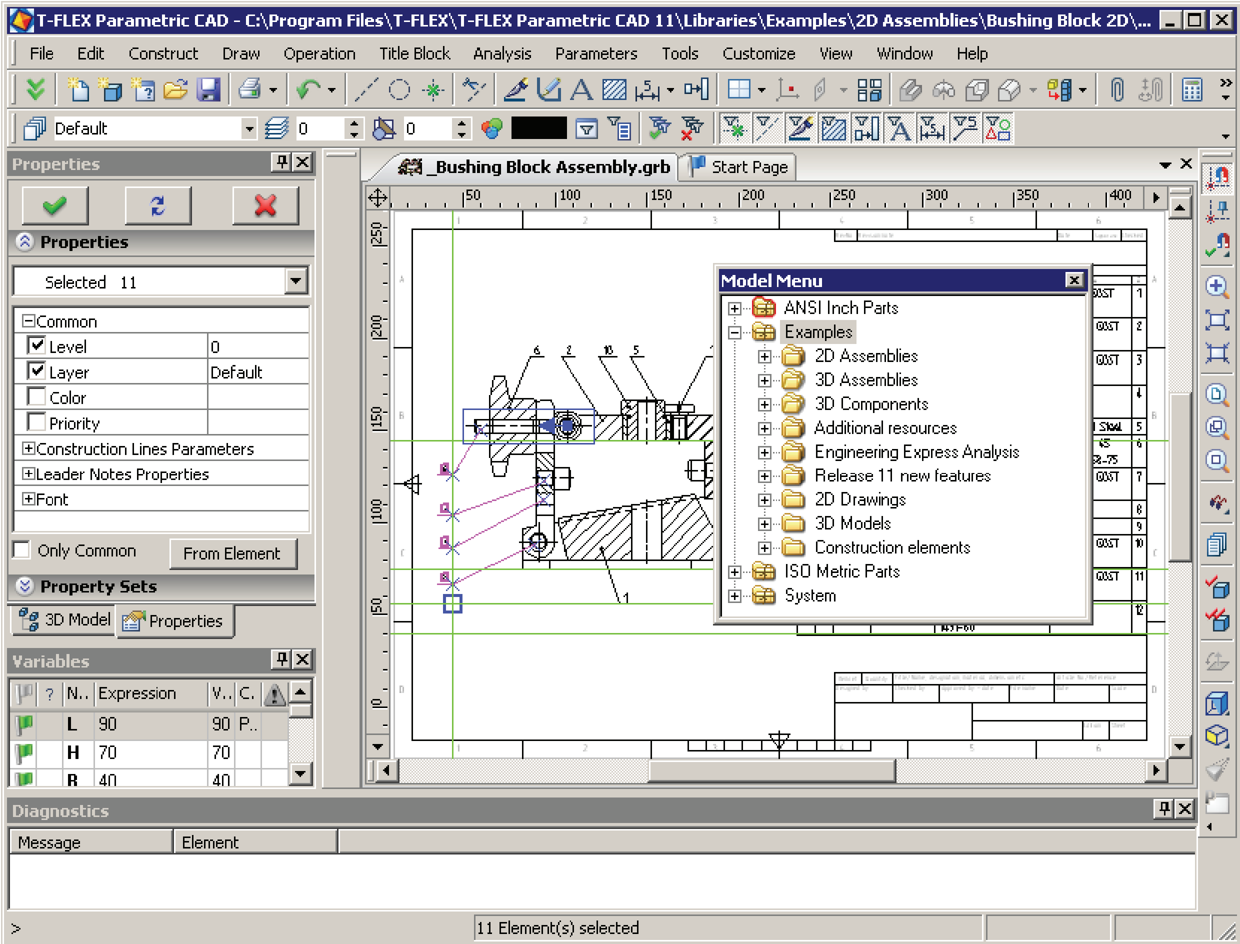
Service Windows

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.
Model Menu 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.
Window "Variables"	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.
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.

Working with Tool Windows

The system tool windows (the properties window, "3D model", "Model Menu", 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.

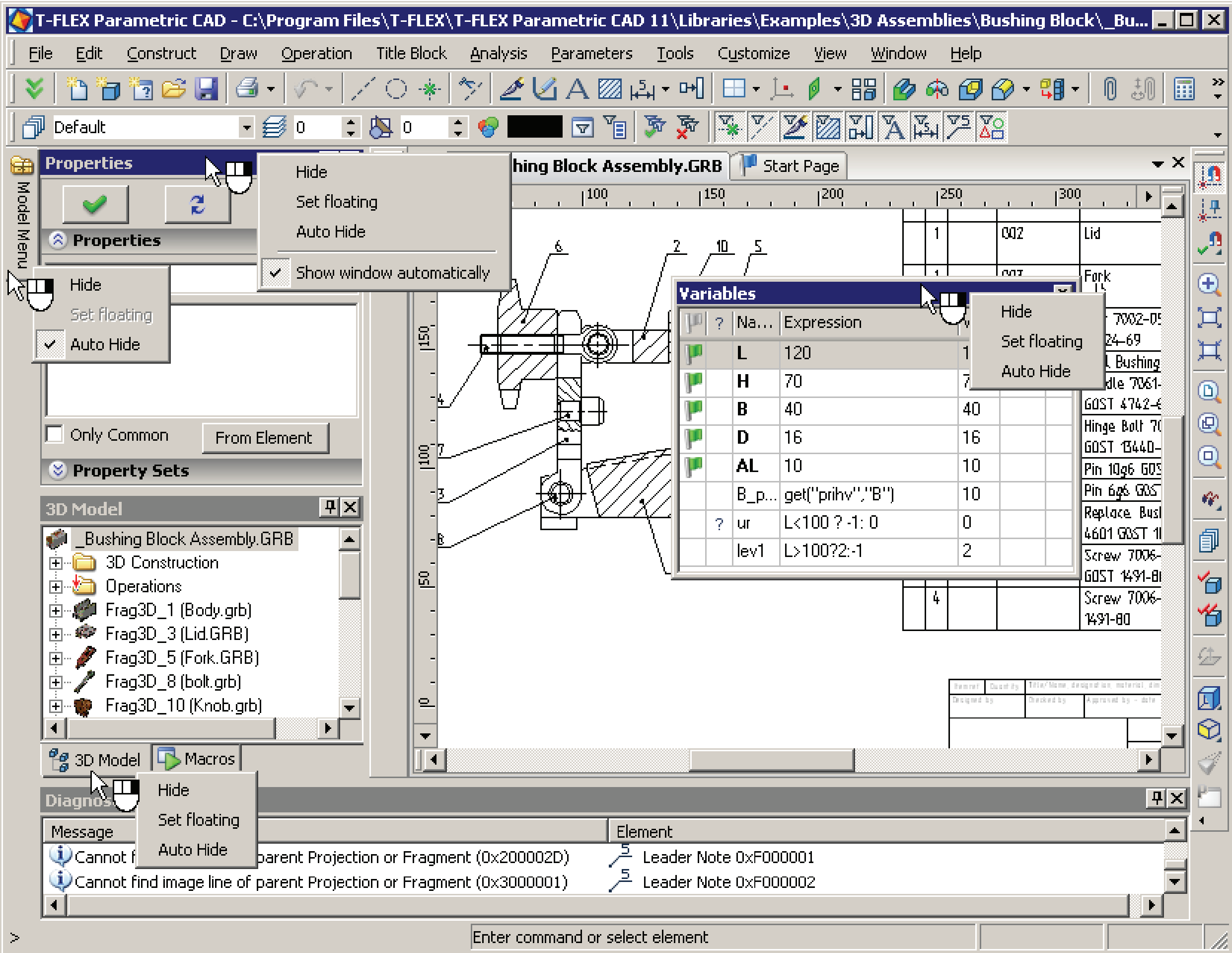


To engage a tool window, use the menu **"Customize|Tool Windows"**. The same dialog can be accessed by right clicking over an automenu of any other toolbar. Windows are closed by the button located on the title bar of the tool window.

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.

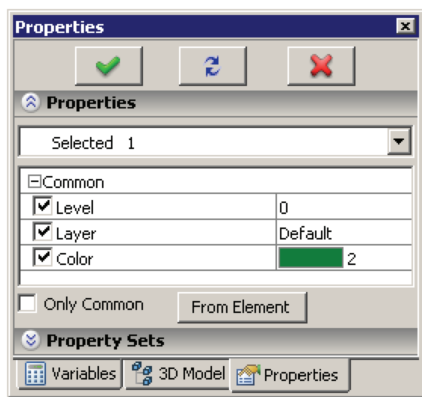


Upon the first launch of the system, the “3D Model”, “Model Menu” and “Properties” windows are already present in the application workspace. Those are placed in the “**docked”** mode along the left border of the workspace and are joined in one group window. If necessary, the two windows can be moved 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.

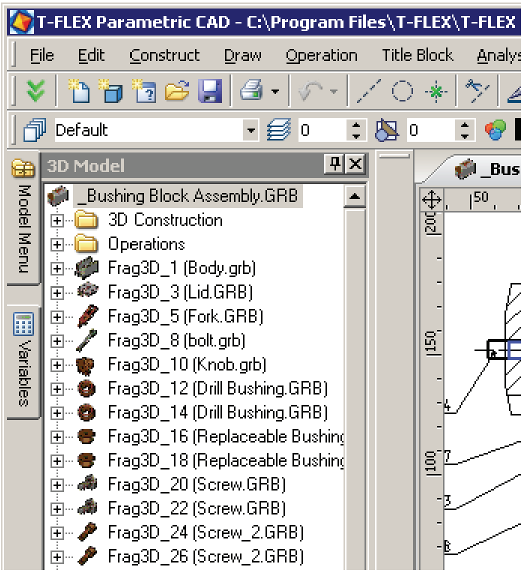
Upon dragging the windows several prompt signs will emerge showing where the window will be placed when the mouse is released.

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,

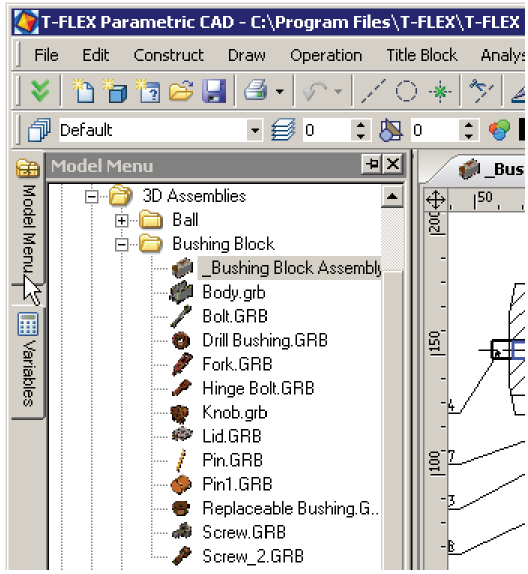


it will automatically collapse.

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

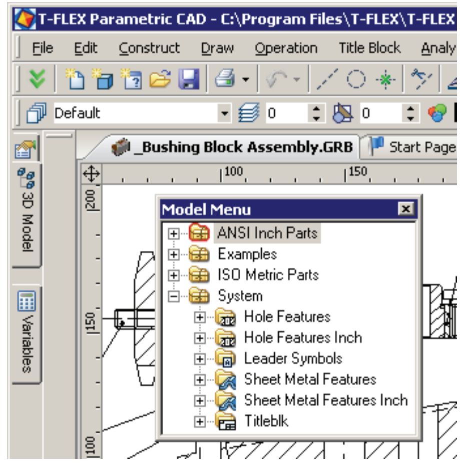
The auto hide mode can be canceled by right clicking on the window tab and clearing the flag of the **"Auto hide"** parameter. This mode helps save significant space on the screen while maintaining benefits of the tool window functionality. Also, to turn off the auto hide mode, the button on the header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



Drawing Basic Terms

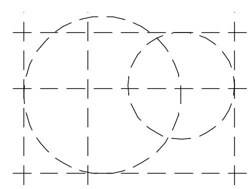
Drawing in T-FLEX CAD involves using several types of entities.

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.

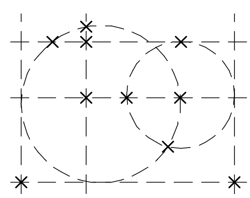
The Auxiliary Entities of T-FLEX CAD are variables, databases, reports and other certain system data.

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.

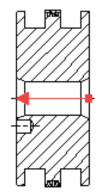
The typical technique of creating a parametric model implies creating nodes at construction line intersections. This technique is called **“constrained drawing mode”**. While in “constrained drawing” mode, creating a node at some location will undergo automatic snapping to the nearest to cursor pair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

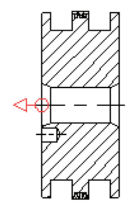
The “constrained drawing” mode is indicated by the icon  of the T-FLEX CAD automenu.

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.

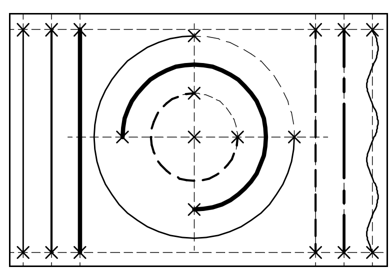


Fixing Vector is a construction entity that helps defining the location and orientation of the drawing that is used as a fragment in an assembly drawing.



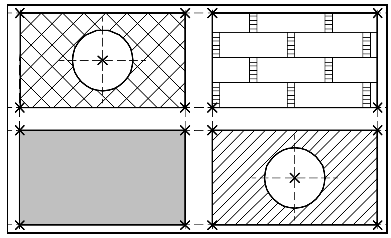
Connector is a construction entity that provides a placement reference for 2D fragments. Besides the geometrical location (the origin of the coordinate system and the axes orientation), a connector can keep additional data (both the dimensional and non-dimensional) that is necessary for “plugging in” 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



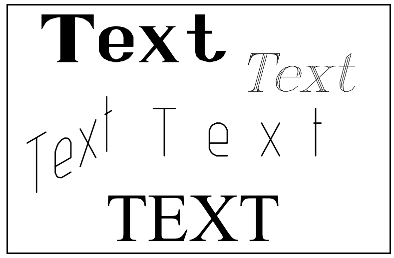
Graphic Lines are the lines constituting the actual drawing of the drawing. Graphic lines include straight segments bound by a pair of nodes, full entities, such as circles, closed splines and so on, except for the infinite straight lines, and the portions thereof bound by pairs of nodes, also splines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

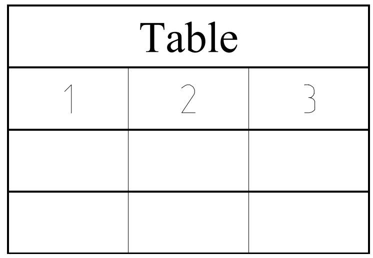
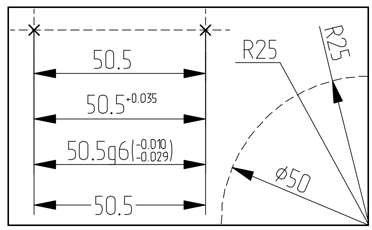
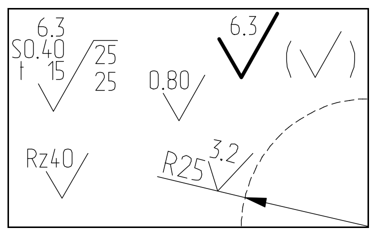


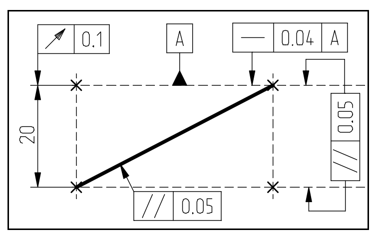
Table is an element of drawing layout. It is composed of lines and textual data. Tables are created by the same command as text. A table can either be located in absolute coordinates and thus independently from the construction entities, or be snapped to nodes.



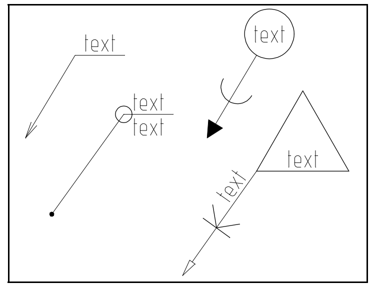
Dimension is a standard element of drawing layout. It is composed of lines and textual data. A dimension is created with respect to construction lines and nodes. T-FLEX CAD supports several dimensioning standards, including ANSI and Architectural ANSI. Dimensions automatically adjust to parametric modifications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



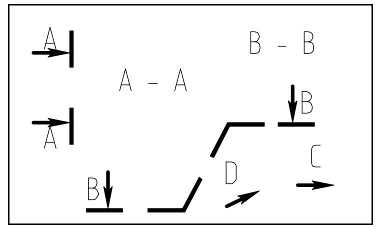
Roughness Symbol is a standard element of drawing layout. It is composed of lines and textual data. A roughness symbol can either be located in absolute coordinates, or be snapped to a node, construction or graphic line, and to a dimension.



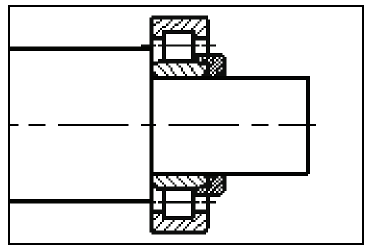
Geometric Datum and Tolerance Symbol (GD&T Symbol) is a standard element of drawing layout. It is composed of lines and textual data. A GD&T symbol can be snapped to a node, construction or graphic line, and a dimension, or located in absolute coordinates.



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.

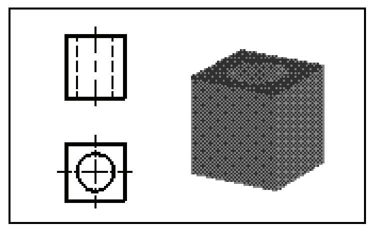


Section symbol is a standard element of drawing layout. It is composed of lines and textual data. This symbol marks various views, sections and cuts. The element can either be located in absolute coordinates, or snapped to a node.

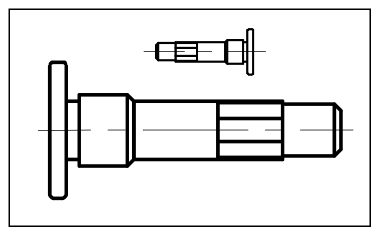


Fragments are T-FLEX CAD drawings that are used in other drawings in subassemblies and assemblies. Any T-FLEX CAD drawing can be used as a fragment.

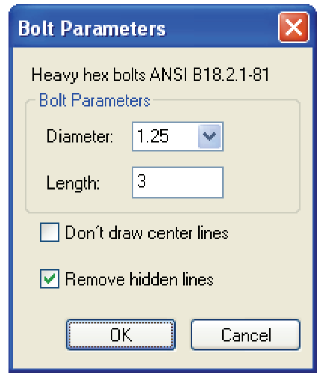
A parametric fragment in T-FLEX CAD is a drawing that can be inserted (assembled) into another drawing to a specified location and with modified parameters. The fragment appearance shall change to satisfy the parameter values. In order to create parametric fragments, the user needs to follow certain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



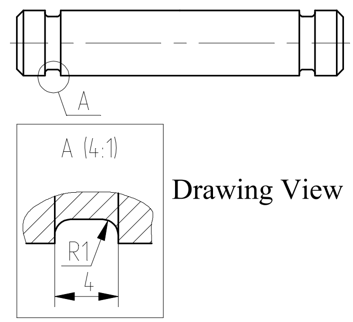
Pictures are graphic images saved in various file formats.



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

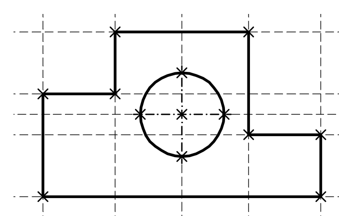
Variable is a system element for specifying non-geometrical dependencies between the various parameters. One main use of the variables is assigning their values to the construction line parameters. Consider, for example, a line parallel to a given line, at a certain distance. This distance can be defined 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.

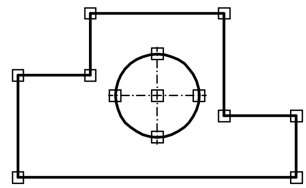
Reports are textual documents that are created with the T-FLEX CAD text editor. Reports can include the system variables and are used for creating various text documents.

Drawing Techniques

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

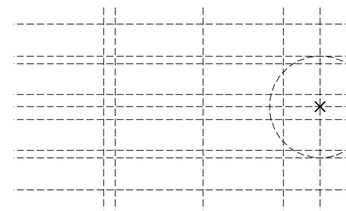


Parametric Drawing. This is the recommended drawing technique in T-FLEX CAD. Take the advantage of parametric design capabilities of T-FLEX CAD to create a drawing that can be easily modified according to your design intent. Such a drawing can also be added to a parametric model library to be later used 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.



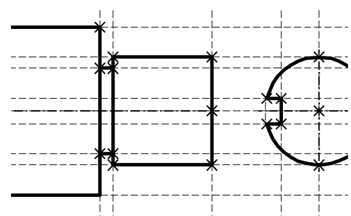
Non-parametric Drawing (Sketch). This is a conventional drawing similar to those created by most CAD systems. This drawing is created by using the standard set of functions for plotting different basic entities (straight lines, arcs, circles, ellipses, splines etc.) and by using the mechanism of objects 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



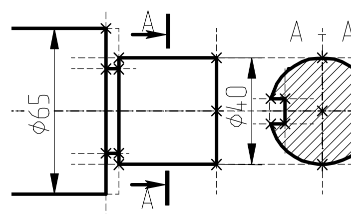
Creating a drawing in T-FLEX CAD begins with creating construction entities. Construction entities can be created by various means. First, create the base construction lines that will be used as a reference for additional construction lines. The base lines can be vertical or horizontal. Next, create 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.



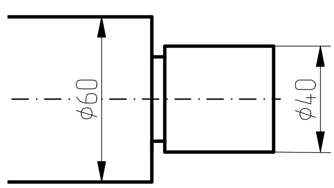
Thus, the early stage of creating a drawing involves building parametric dependencies among construction entities that become the parametric framework of the drawing.

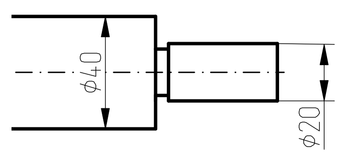
Once the construction framework is built, proceed with drawing the graphic entities. Create line segments, arcs and circles by drawing over the construction lines, snapping to nodes.



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.

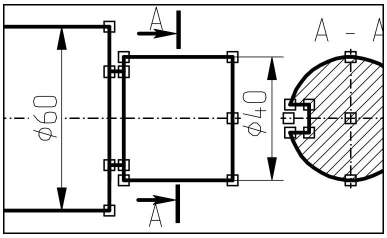
> Note that the above scenario for creating a parametric drawing in T-FLEX CAD is just one recommended technique. One can create construction entities and graphic entities in an arbitrary sequence. What is important is that the graphic entities are constrained to the construction ones.

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.

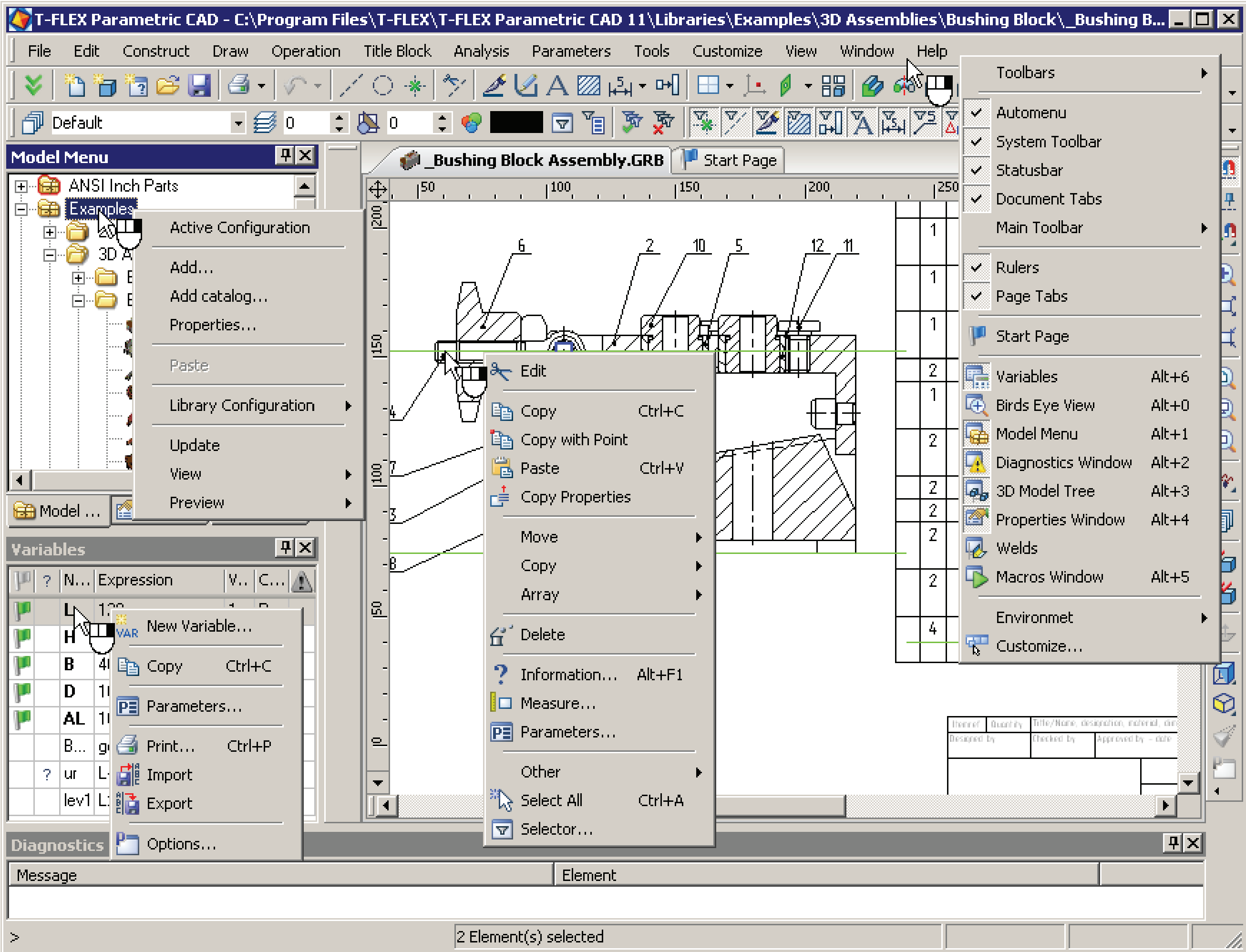
  If the mouse is moved from left to right the entities will be selected that are entirely within the selection box. The box is drawn with continuous line.

  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:

Keyboard input	Textual Menu	Icon
<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.

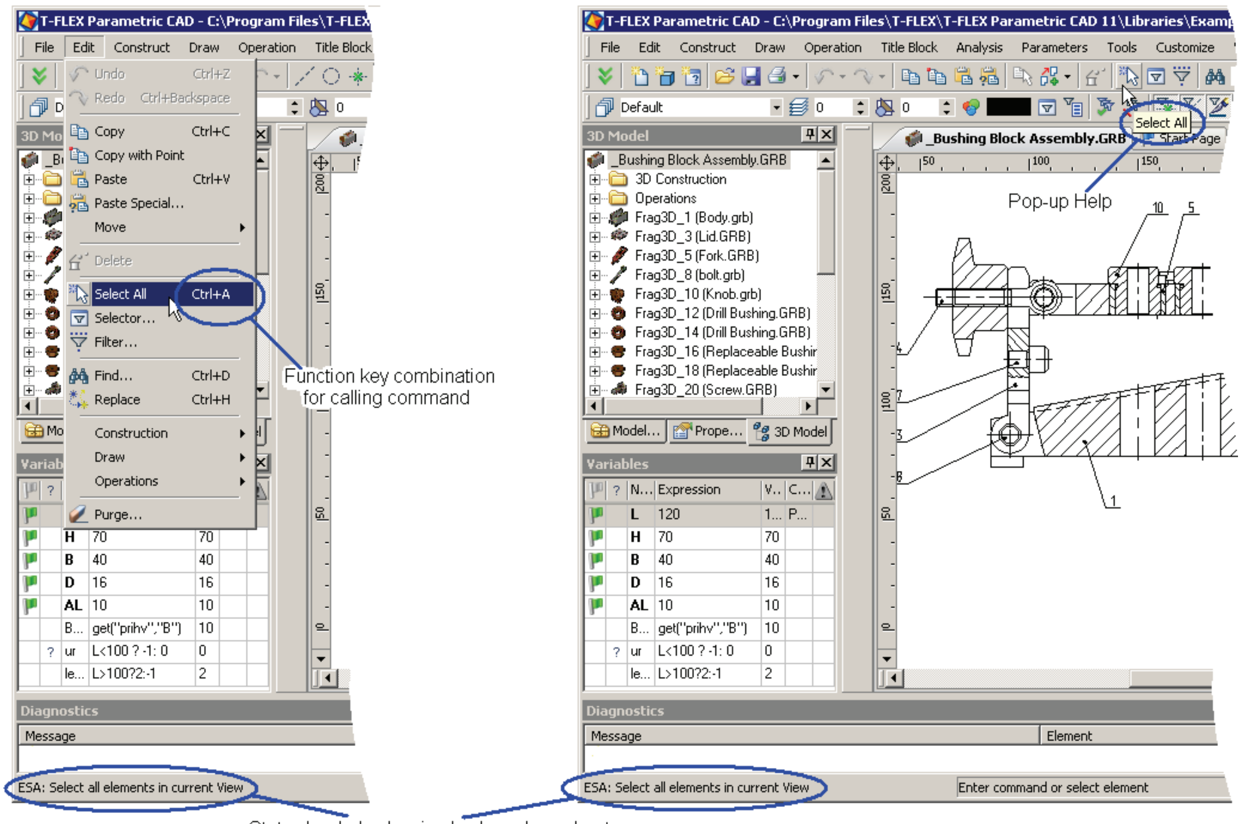
The second column contains the access sequence for the command via the textual menu. The name before the dividing line is the name of the appropriate group item in the menu bar. It is followed by the item name in the pull-down menu that stands for the command. The menu item name may be different (abridged)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

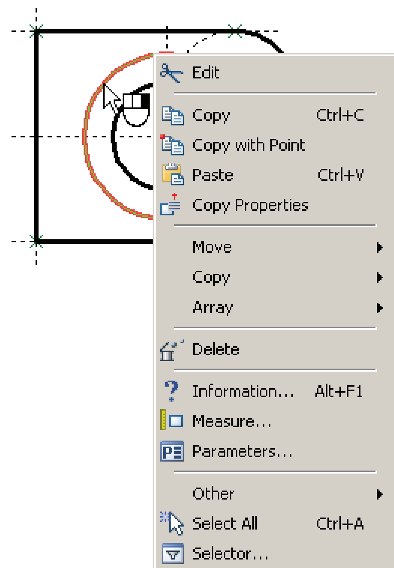
The third column of the table contains the icon image for the command. Normally, the particular toolbar containing the icon has the same name as the menu bar group item. For user convenience, a popup with the command name appears when the cursor is briefly held over an icon. Once a command is activated 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 [mouse icon] 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.

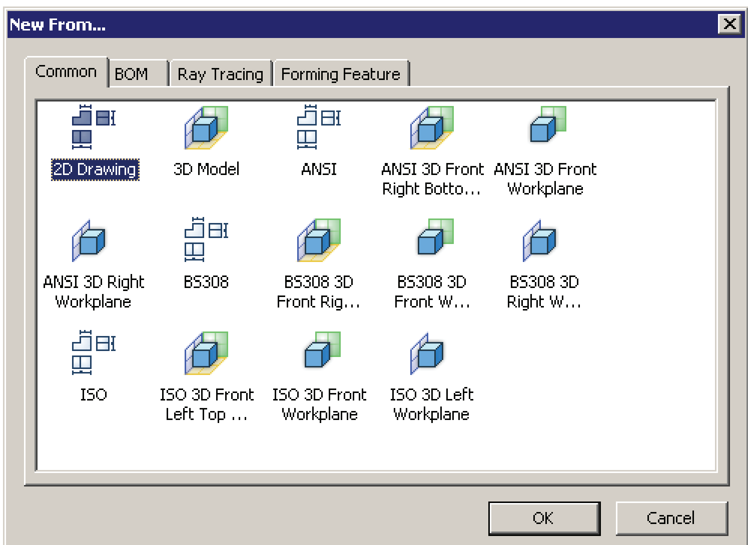
In addition to the dialog box "Start Page", to create new documents and open already existing ones, the system commands gathered in the textual menu "**File**" can be used.

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

Keyboard input	Textual Menu	Icon
<FN>, ***<Ctrl><N>***	**"File|New"**	

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

Keyboard input	Textual Menu	Icon
<FP>	**"File|New From Prototype…"**	

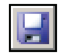


"O: Open Model" command brings up the standard "Open" dialog box to open a document for editing:

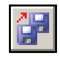
Keyboard input	Textual Menu	Icon
<O>,	**"File|Open…"**	

<Ctrl><O>		

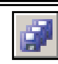
"SA: Save Model" command saves the current document:

Keyboard input	Textual Menu	Icon
<SA>, <Ctrl><S>	**"File|Save"**	

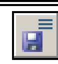
"SV: Save Model As" command allows the user to save the current document into a new file with a different name without changing the original document:

Keyboard input	Textual Menu	Icon
<SV>	**"File|Save As…"**	

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

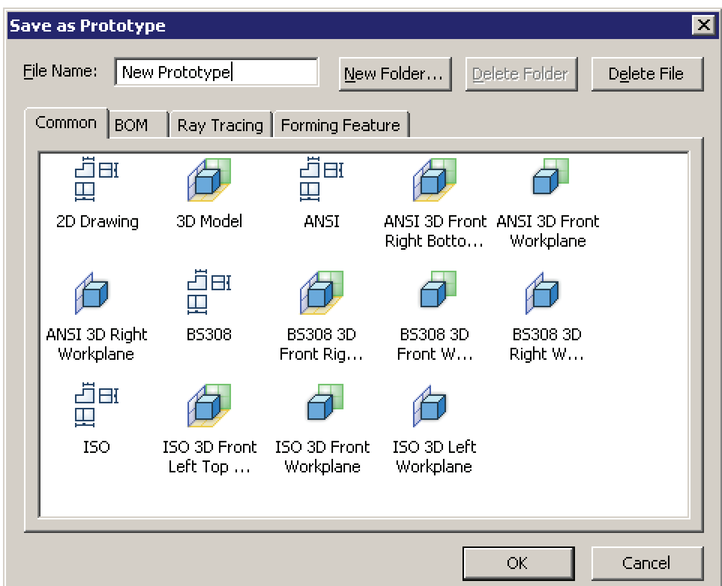
Keyboard input	Textual Menu	Icon
<SL>	**"File|Save All"**	

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

Keyboard input	Textual Menu	Icon
<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.

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

Keyboard input	Textual Menu	Icon
<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:

Keyboard input	Textual Menu	Icon
<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.

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”). Recall that this dialog box is always present on the screen when the standard settings of the system are used, and 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 document, use the command **“FN: Create New Model”**:

Keyboard	Textual Menu	Icon
<FN>*, *<Ctrl><N>	**“File|Create New Model”**	

Upon calling the command, a new unnamed document is created (`NONAME1, NONAME2`…). You will have to specify a name when saving your drawing.

An additional command **“F3: Create New 3D Model”** is available in the 3D version of the system:

Keyboard	Textual Menu	Icon
<F3>	**«File|Create New 3D Model»**	

Upon calling this command, a new 3D model is created. As when creating a new drawing, you need to assign a name to the created document.

> Recall that the 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”. Those may contain elements and settings that will be automatically created or enabled with the new document creation. In the case when new document settings need to be modified, edit 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”).

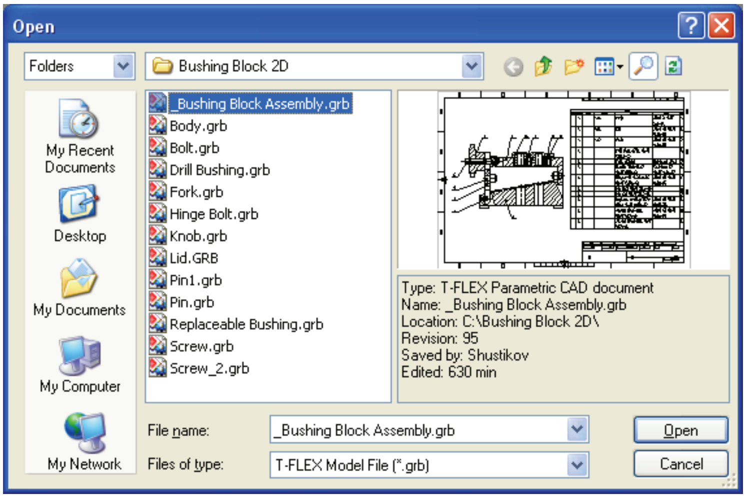
A user on his own can create an arbitrary number of prototype files. A new file can be created from a prototype using the the option “New document” of the dialog box “Start Page”. Otherwise, use a similar dialog “New From...” by calling the command **“FP: Create New Document Based on Prototype”**.

Opening Document

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

Keyboard	Textual Menu	Icon
<O>*, *<Ctrl><O>	**"File|Open…"**	

The new window "Open" will appear on the screen.

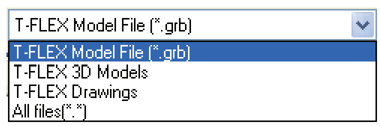


This is the standard Windows dialog box for opening application files, except for some additional elements.

A box in the top-left corner of the dialog window is provided for selecting the storage source of the document.

This can be either the standard Windows folder tree (**"Folders"**), or the T-FLEX CAD libraries pool (**"Libraries"**).

The **"Files of type"** box at the bottom of the dialog helps filtering the files of the desired type. This can be a 2D drawing file (**"T-FLEX Drawings"**), a 3D model (**"T-FLEX 3D Models"**) or any T-FLEX CAD document (**"T-FLEX Model File"**). Once a particular type is selected, files of this type only will be listed in the dialog.

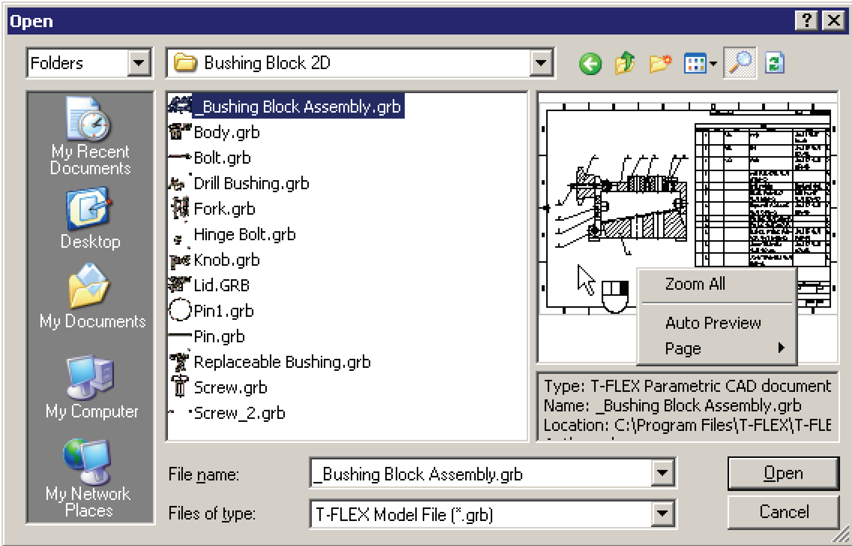


There are two panes at the right-hand side of the dialog for previewing the document content and the document properties. A button is provided in the top-right corner for showing/hiding the preview panes.

The preview image, displayed in the "document content" preview pane, uses by default a vector graphics representation or a bitmap thumbnail stored with the document. If no preview data is available in the document, a message is displayed instead, "Click to Preview". In this case, the actual document content will be displayed upon pointing the cursor at the preview pane and clicking .

To force automatic preview generation for documents without preview data, press button. The same can be done by pointing cursor at the preview pane and calling the context menu by right-clicking . In the menu, select the "Auto Preview" button. (Next time the button will be displayed with a checkmark.) Once displayed, the preview image can be zoomed by clicking .

To refit to the full image in the preview pane or to set other options, click , and press the "Zoom All" button in the coming up context menu.



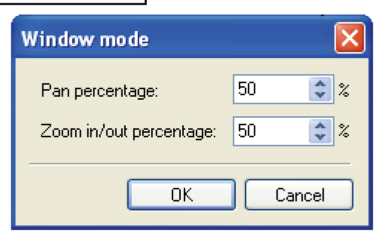
Panning and Zooming in Active Drawing Window

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, such as IntelliMouse. 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:

Keyboard	Textual Menu	Icon
<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 Model to paper size

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

	<M>	Zoom Limits

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

	<M>	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

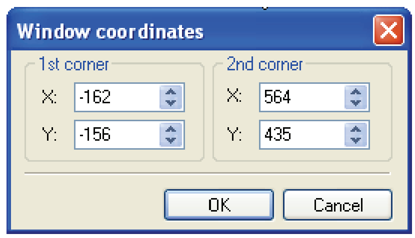
These options respectively magnify and shrink the drawing image each time by a fixed percentage ratio specified in the command parameters.

	<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.

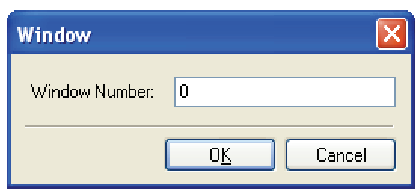


	<BackSpace>	Set previous window coordinates

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.

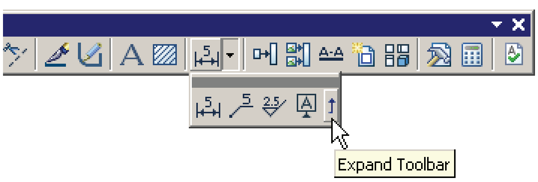
Any toolbar can be docked along any of the application window borders, or floating within. When floating, a toolbar window is titled, and can be resized.

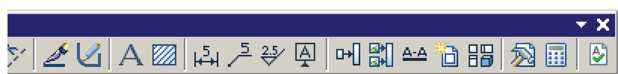
Embedded toolbars

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 Default . Displays the layer name of the model elements being created and edited.

Button for setting visibility levels of the model elements .

Level input box 0 . 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 0 . 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.

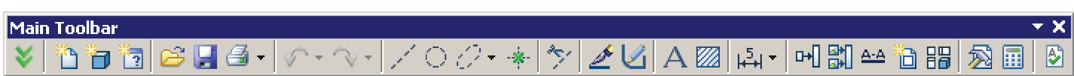
Quick selector setup buttons:

The buttons and help to quickly allow/disallow selection of all types of elements;

The buttons , , , , , , , and 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

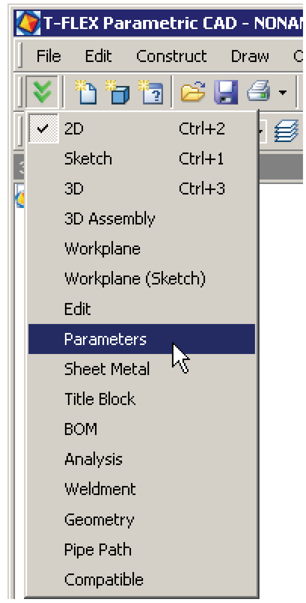


The main toolbar has a set of buttons which, depending on the currently solved problem and the settings of the system, can be selected by the user or automatically activated.

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.

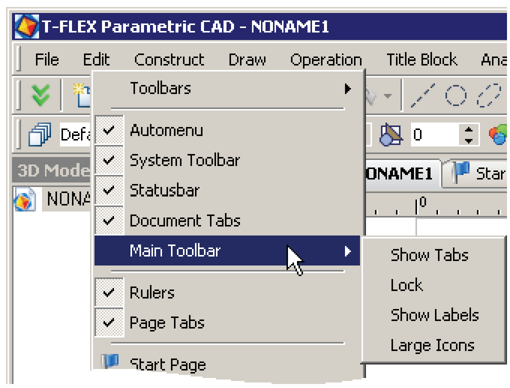
Switching between the button sets can be done manually by using the button on the left side of the toolbar . Upon pressing this button the list of available sets pops up.

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.

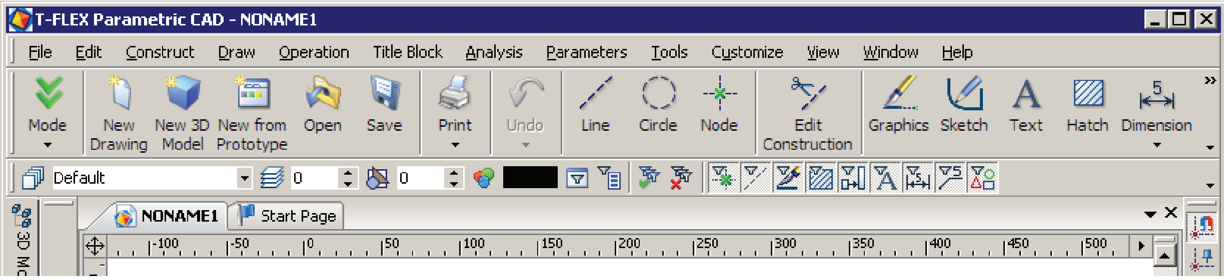
The user selected set is stored in the window of the current document and automatically recovered when the window becomes active. The given setting is stored in the document and gets activated when the file is open.

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.

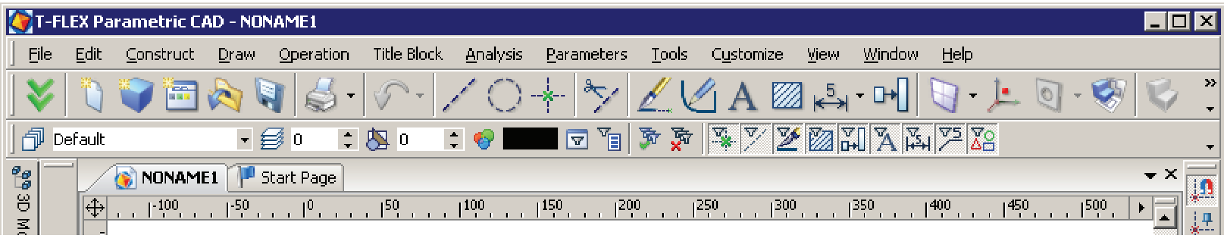
One more flag is available in the same context menu which allows controlling the view of main toolbar – **“Show Tabs”**. It controls visibility of the tabs on the main toolbar. The tabs allow quick switching between the button sets of the toolbar. The tab of the active set is marked with the color.



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.



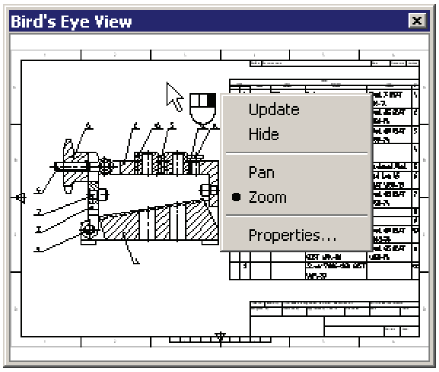
The flag **"Large icons"** allows turning on the mode of the large icons for the main toolbar (no matter what the size of the icons in other system toolbars is).



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.

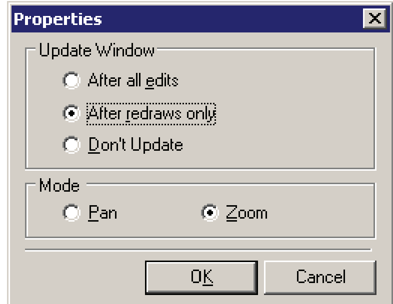
The "Bird's eye view" window can be docked along any of the application window borders or stay floating.

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 pans 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 .

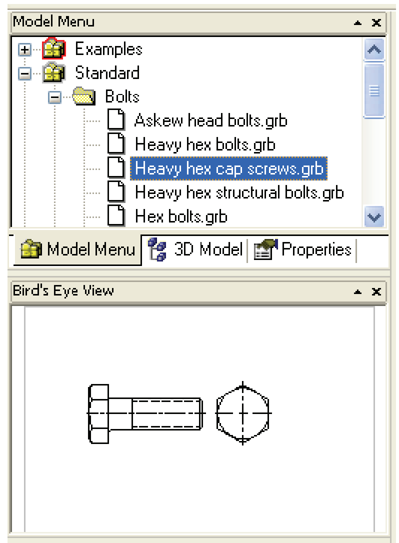
Once defined, the viewing area will be highlighted in the "Bird's eye view" window, and the respective portion of the drawing will be zoomed on in the active drawing window.



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

Using Model Menu

The model menu 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 model menu window via the textual menu item **"Customize|Tool Windows|Model Menu"** or in the context menu on right mouse button click over any toolbar.



The model menu 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.

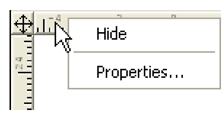
The model menu window may have a preview pane at its bottom or right-hand side. This pane will display the preview image or the properties of the selected document.

The model menu window can have various different settings accessible by .

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

Rulers

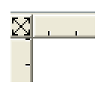
The rulers display X and Y coordinates of the current drawing window. Ruler properties can be set with the help of the context menu (see the figure on the right).



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.



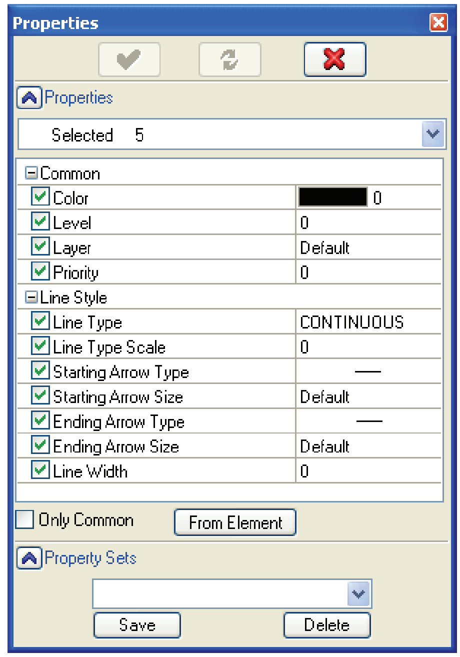
Pressing the button at the horizontal and vertical rulers crossing with switches to another button mark. In this mode, the rulers can be used for zooming the drawing in and out.

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.

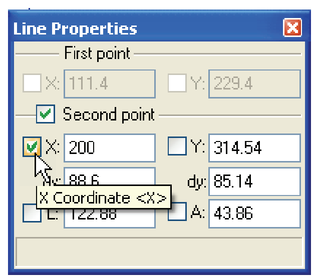
Property Window

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.

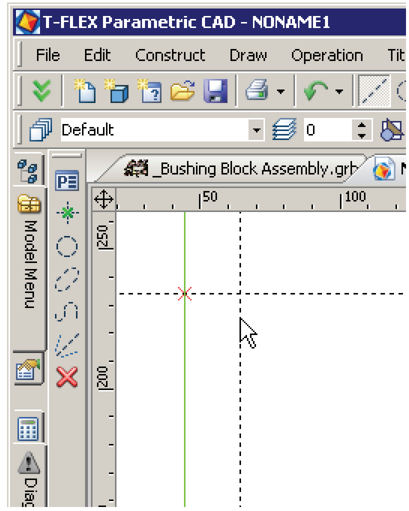


The property window may be expandable. In many commands, portions of the property window dialog box may be collapsed by default. The special buttons -  and  are provided for expanding and collapsing such portions. Once a portion of the dialog was "expanded" while in some command, this setting will be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

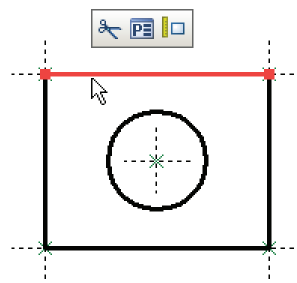
Automenu is a special toolbar that contains icon buttons of the available command options. Automenu is context-sensitive. This means, its content changes depending on the current command and a state of the command.



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.

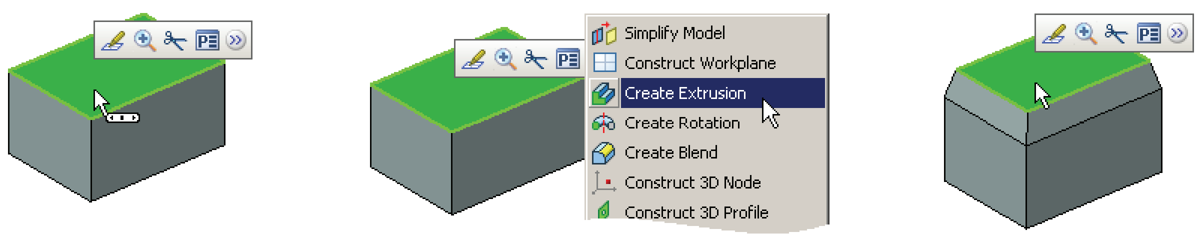
Dynamic Toolbar

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.

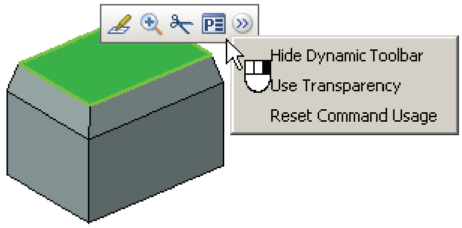


The presence of the dynamic toolbar, while choosing 2D and 3D elements, depends on the settings of the command **"SB: Show Toolbars"**. For 3D elements the dynamic toolbar will be shown if the flag "Use Dynamic Toolbar" is turned on in the given command dialog box on the tab "Preferences". In addition 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.

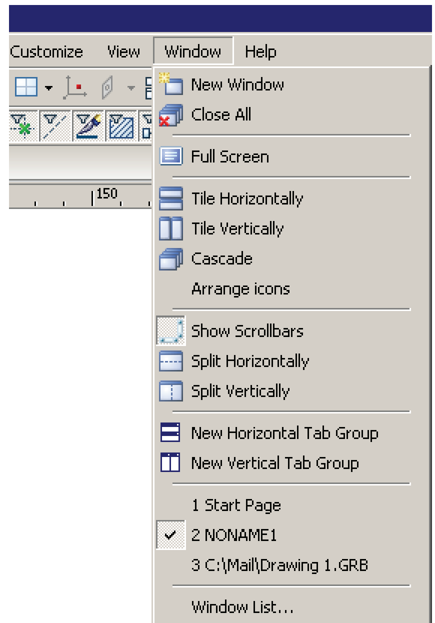


In the context menu of the dynamic toolbar, the flag **"Use Transparency"** is also available. When this flag is set on (for default settings), the toolbar looks semitransparent when it appears on the screen. The semitransparency diminishes as the cursor moves closer to the toolbar. When the flag "Use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

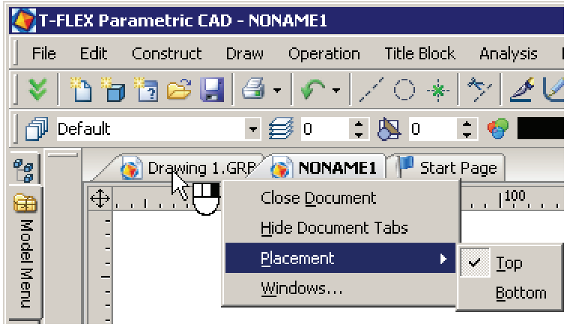
The commands designed for working with document windows are grouped in the submenu "Window" found in the textual menu.



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.

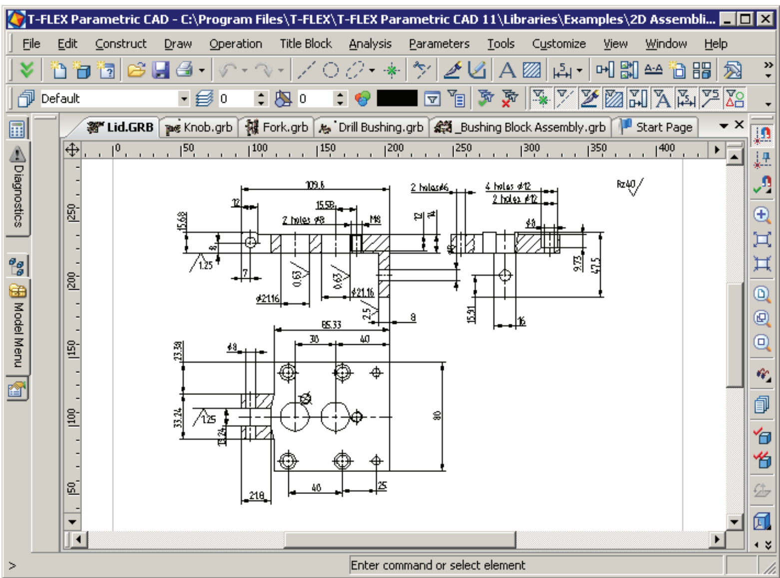
By default the document tabs (if they are visible in the T-FLEX CAD window) are placed above the upper border of the document window. Their location can be changed if desired. For that, it is necessary to point at the tab of any document with the cursor and with the help of call the context menu.



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. At the upper right corner of the active window (when the standard arrangement of the document tabs is enforced) there is a button ✕. By pressing this button the current window will be closed.



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

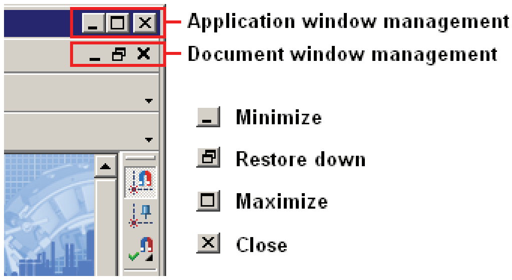
When the tabs are turned off, the windows of open documents can be in one of the three states:

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.



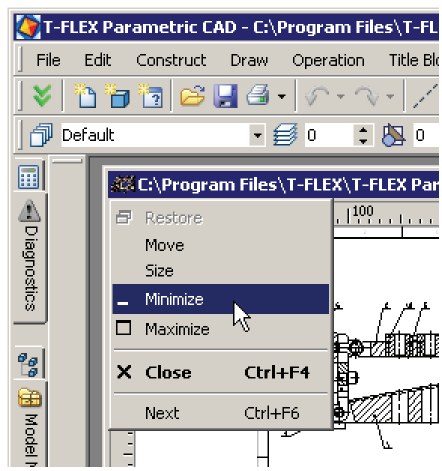
When the window is maximized, these buttons are located under the respective ones of the application window. Pay attention as to what button to use when closing a window.

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.

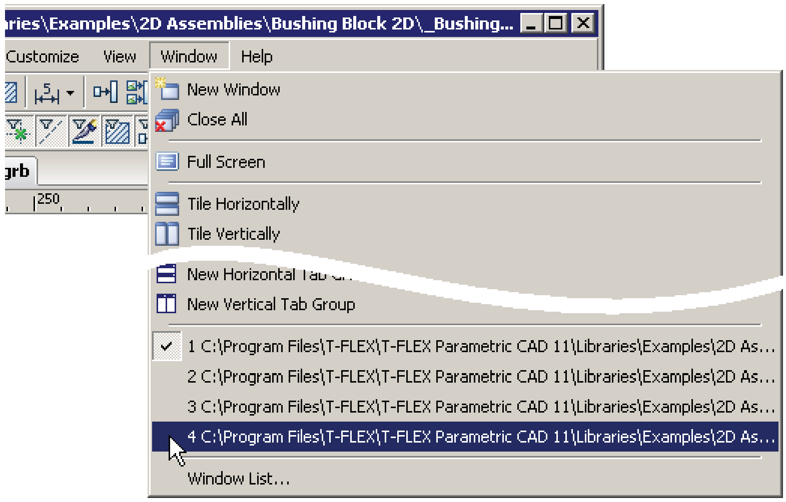


Selection of active window

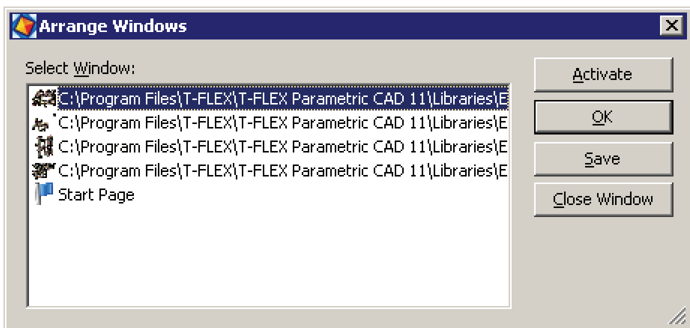
The active window (i.e. the window of the current document) can be selected in a number of different ways.

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.

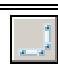


When the tabs are turned on, there is one more way for switching between the windows. In the upper right corner of the current window (with the document tabs arranged in a standard way – along the upper border of the window) there is a button ▾. When this button is pressed, the list of all open documents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

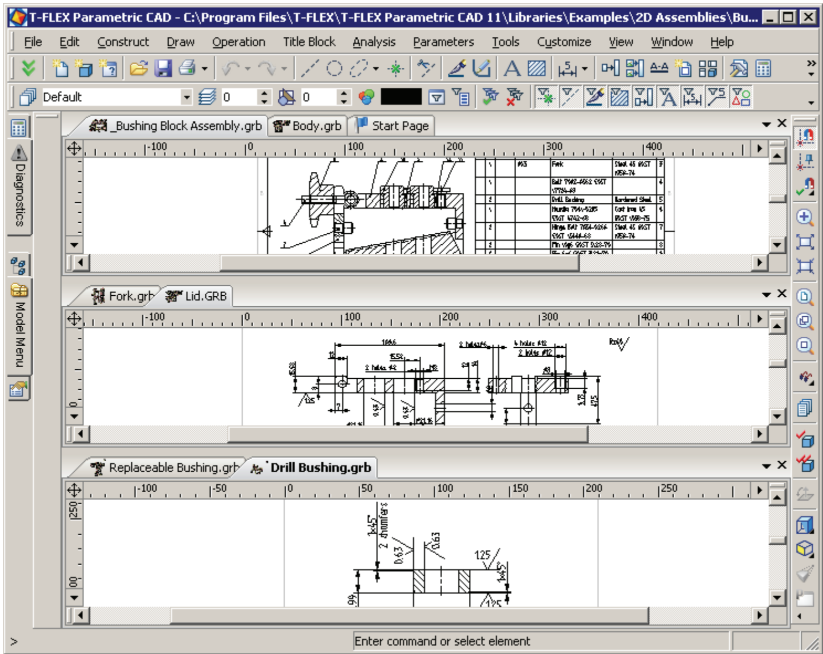
Calling the command **"WSS: Show/Hide Window Scrollbars"** toggles on/off visibility of the active window scrollbars:

Keyboard	Textual Menu	Icon
<WSS>	**"Window|Show Scrollbars"**	

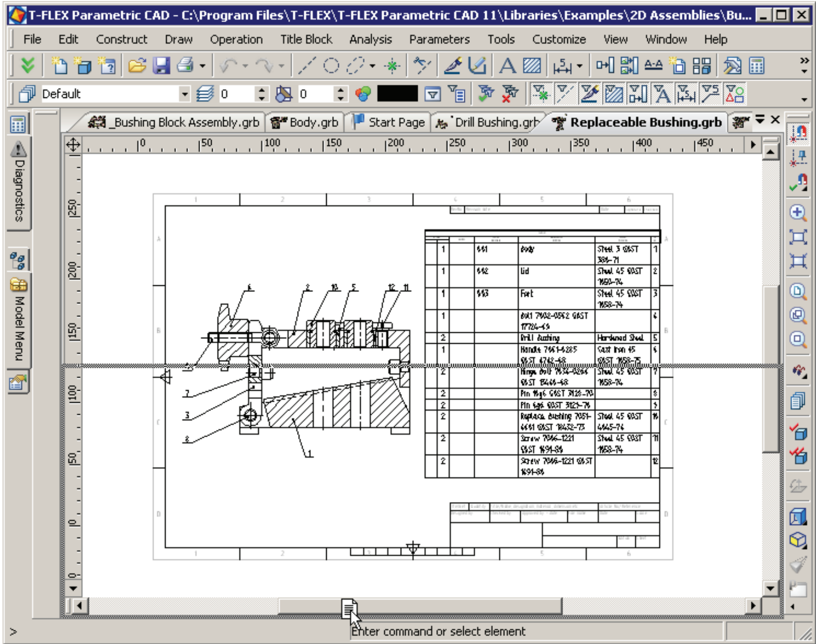
Hiding scrollbars increases the window working area. The function of the scrollbars is not available via keyboard input. However, there is a number of tools offering similar function, and in large variations. See, for instance, the rulers functionality described above, or the use of the mouse wheel.

Arranging document windows with turned on document tabs

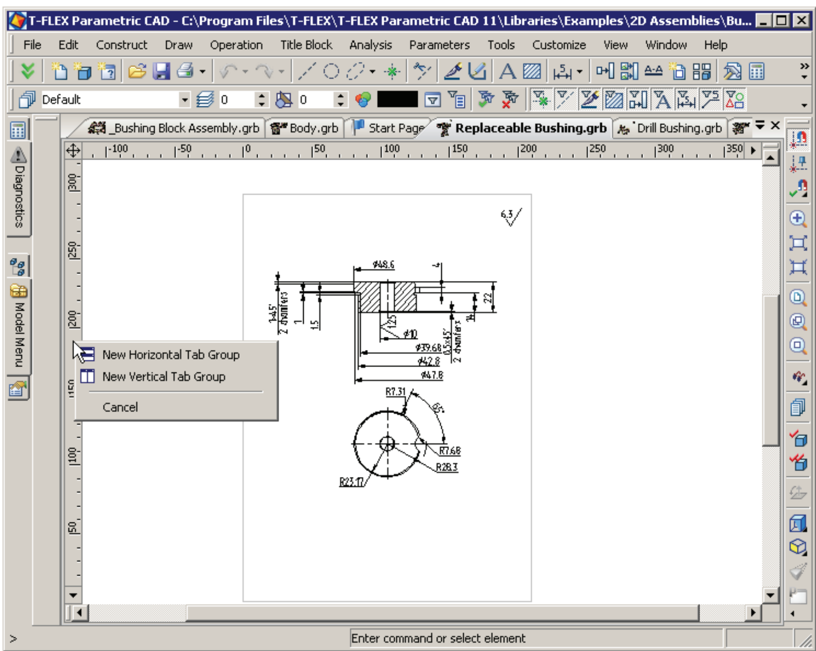
When the window tabs are turned on, the document windows can be grouped into horizontal or vertical groups. Any number of document groups can be created simultaneously, however all groups must be either horizontal or vertical.



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.

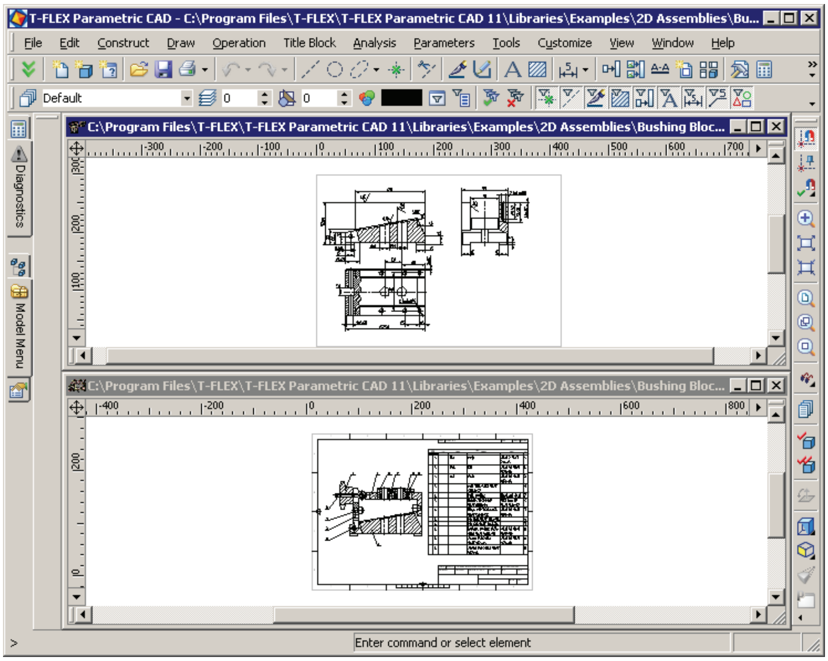
Arranging Document Windows with Turned off Tabs

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:

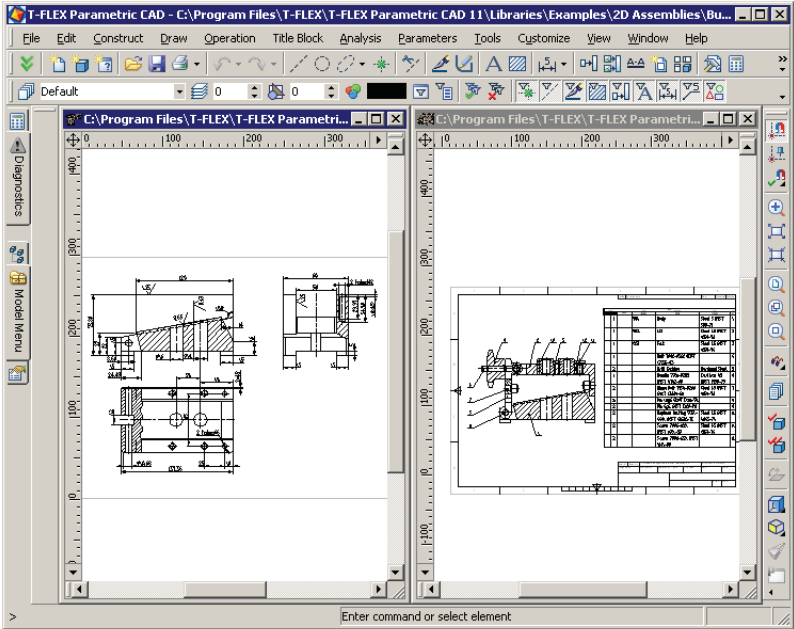
1. **Tile Horizontally**. Do this by the command **"WHT: Tile Windows Horizontally"**. Call the command using:

Keyboard	Textual Menu	Icon
<WHT>	**"Window|Tile Horizontally"**	



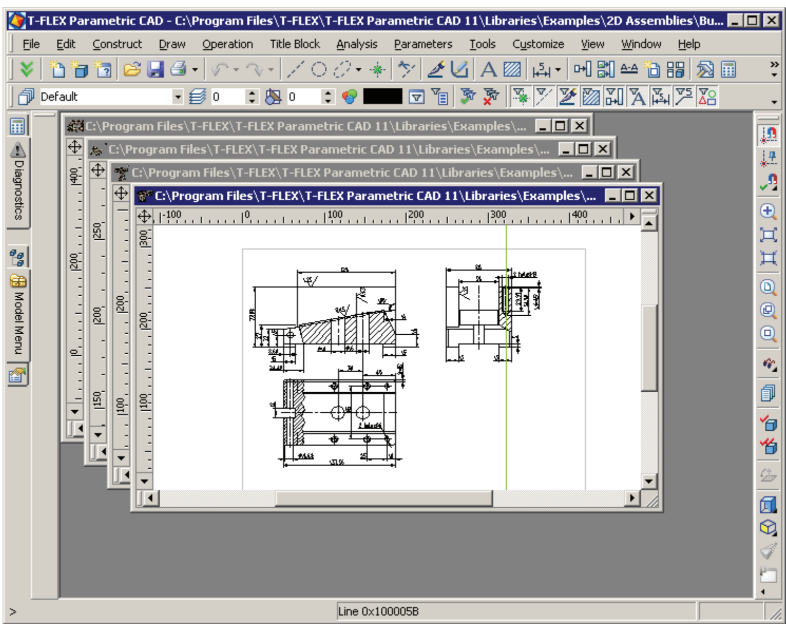
2. **Tile Vertically**. Do this by the command **"WVT: Tile Windows Vertically"**:

Keyboard	Textual Menu	Icon
<WVT>	**"Window|Tile Vertically"**	



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

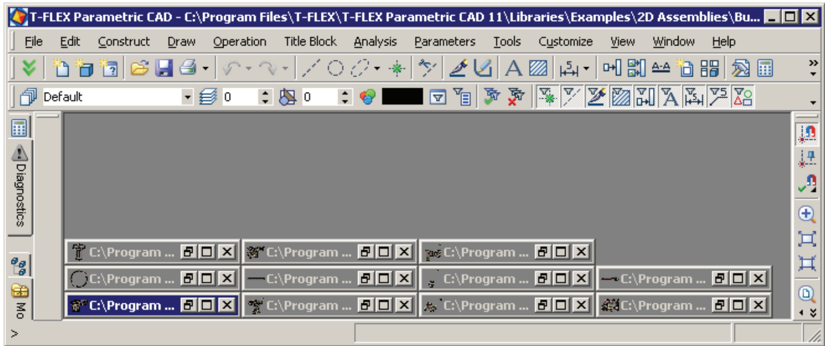
Keyboard	Textual Menu	Icon
<WCA>	**"Window|Cascade"**	



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.

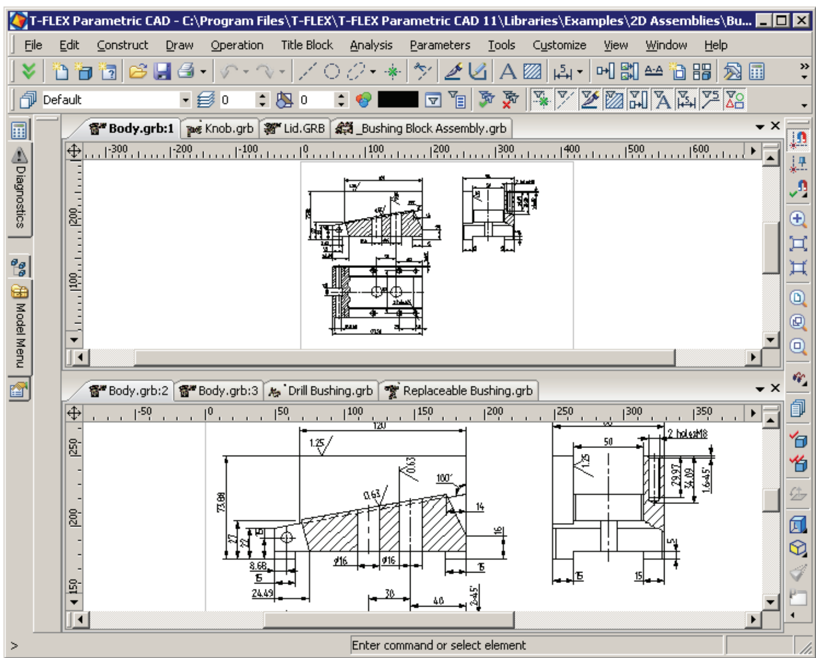
When the windows of all documents are minimized, they can be placed along the lower border of the working zone with the help of the command **"Arrange icons"**:

Keyboard	Textual Menu	Icon
–	**"Window|Arrange icons"**	-



Additional window of document

T-FLEX CAD enables to create additional windows for already closed documents. The name of document and the serial number of the given window appear in the title bar of such windows, for example: `"Drawing.grb: 1"`, `"Drawing.grb: 2"`.



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:

Keyboard	Textual Menu	Icon
<WO>	**“Window|New Window”**	

When a new window is created, it contains the currently active document. When creating a new window, the user needs to specify the window type, 2D or 3D.

Splitting Drawing Window

An active window can be split horizontally into two panes by calling the command **“WSH: Split Window Horizontally”**:

Keyboard	Textual Menu	Icon
<WSH>	**“Window|Split Horizontally”**	

To remove the horizontal split, toggle the icon off.

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

Keyboard	Textual Menu	Icon
<WSR>	**“Window|Split Vertically”**	

To remove the vertical split, toggle the icon off.

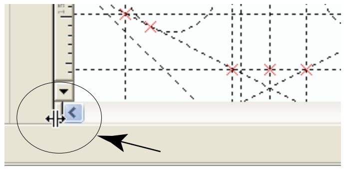
Consequent calling the two commands splits the active window into four panes.

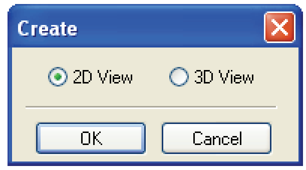
Current window	
1	2
3	4

When using the 3D version of the system, each call to any of the two splitting commands brings up a dialog box on screen for specifying the type of the drawing window to be created.

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.

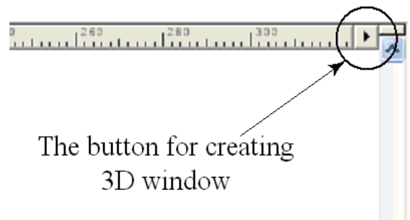
To split, move the cursor to the bottom-left corner of the drawing window of the T-FLEX application. Point the cursor at the box indicated by an arrow on the diagram below. As the cursor changes to “adjust split” arrows, press and hold  and drag rightwards to the desired location of the split. Then release the mouse button, and in the coming up dialog box select the new window type: “2D Window” or “3D Window” and press the **[OK]** button. The new window will be created on the right-hand side.





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.



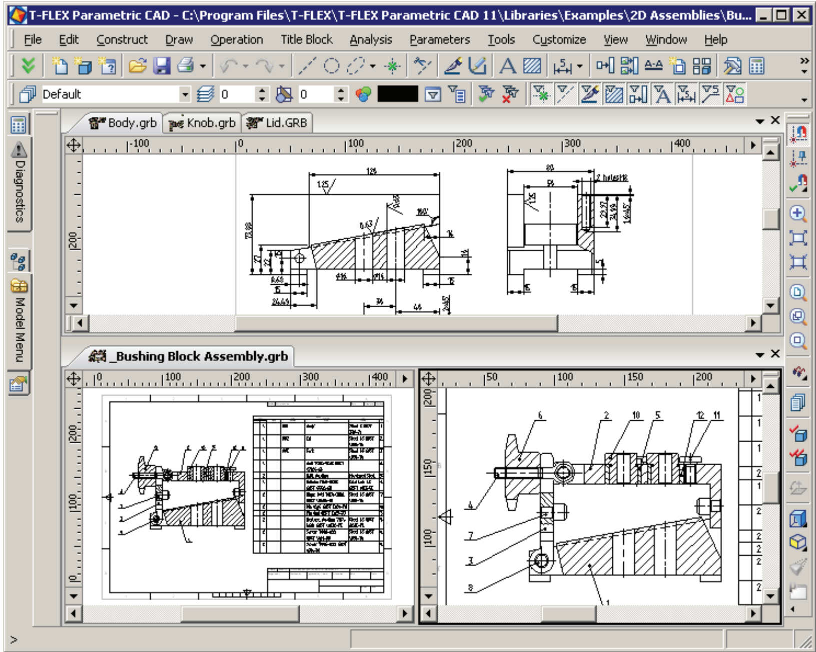
The button for creating
3D window

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.

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:

Keyboard	Textual Menu	Icon
<FCL>	**«File|Close»**	

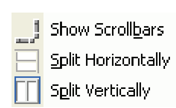
To close all windows of all open documents the command **"WCS: Close all Windows"** can be used:

Keyboard	Textual Menu	Icon
<WCS>	**«Window|Close All»**	

After calling these commands the windows in which the drawings were not modified will be automatically closed. To close the windows of the modified drawings it is necessary to confirm that they need to be saved.

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.

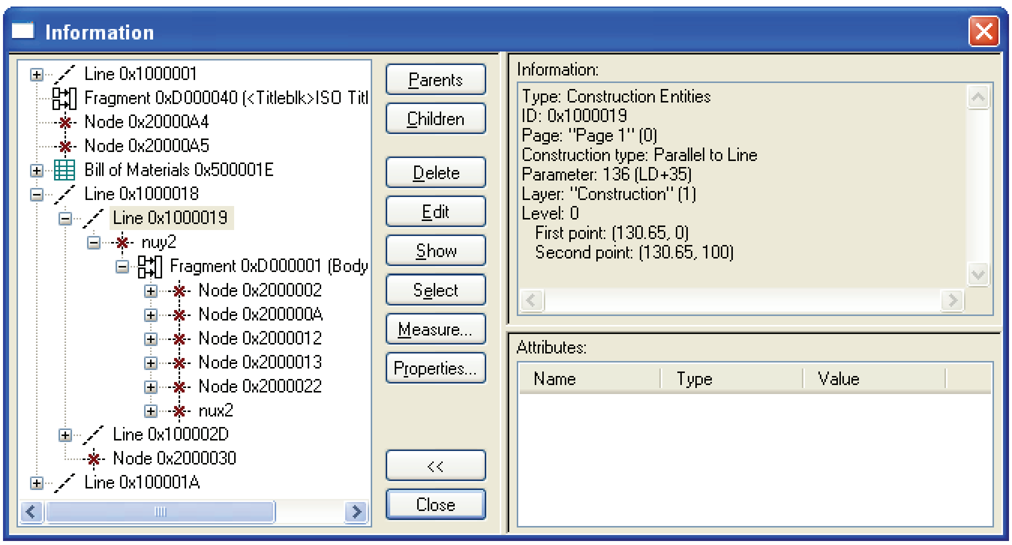
If a T-FLEX CAD document contains several pages, then the tabs with the names of the visible pages may be shown in the lower part of the drawing's window (with the default settings; the arrangement of the page tabs can be modified). One can switch from page to page using those tabs, by clicking them 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	Icon
<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.

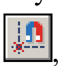
Attributes. This pane displays information about the attributes of the selected element.

Creating and Editing Drawing Elements

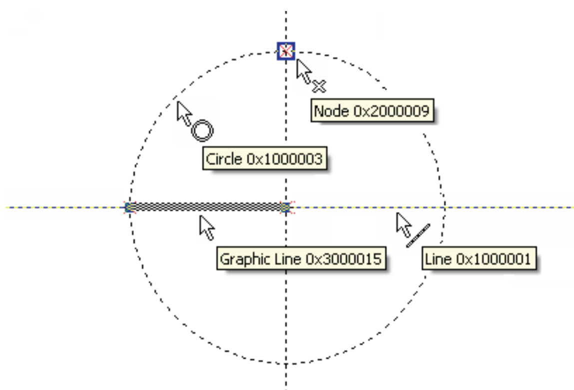
The system provides a specific command for creating and editing each type of model elements. This section describes main concepts of using these commands, as well as general principles of creating and editing a 2D drawing.

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.

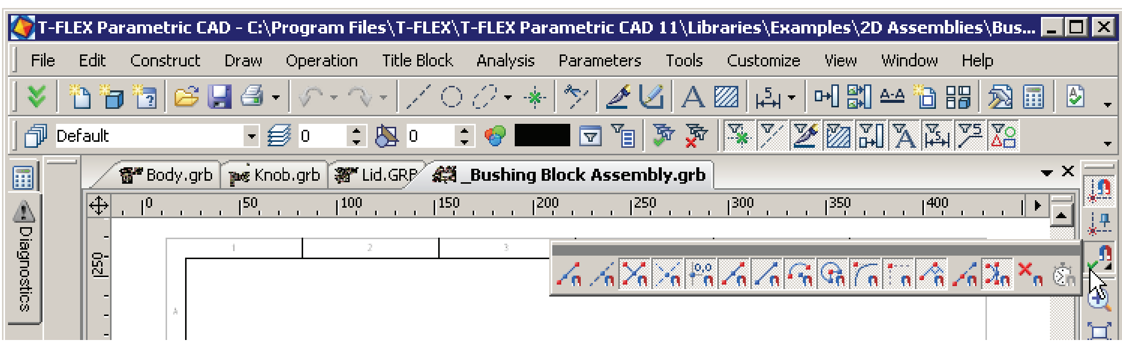
> Please note that the current documentation refers to the element selection mode with disabled object snapping when describing commands (implying only the use of automenu options).

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

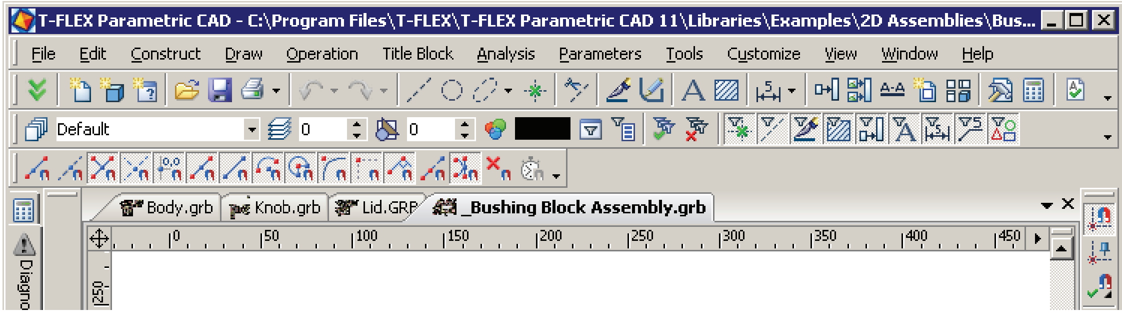
When defining positions of various 2D elements in their creation/editing commands, with the object snapping enabled, not only can you use the existing elements (construction lines, graphic lines, nodes etc.), but also select characteristic points defined by *object snaps*. Nodes can be automatically 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.

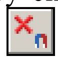
The most number of object snaps is used in the sketch-creating command **"SK: Create Sketch"**. Some of the object snaps may be unavailable in other 2D commands. Besides that, the use of snaps is affected by the settings made in the command **"SO: Set System Options"**. You specify what snap types can 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 .

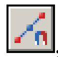
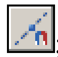
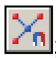
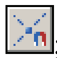
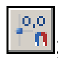
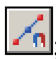
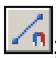
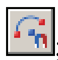
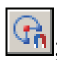
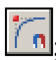
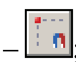
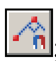
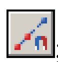
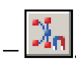


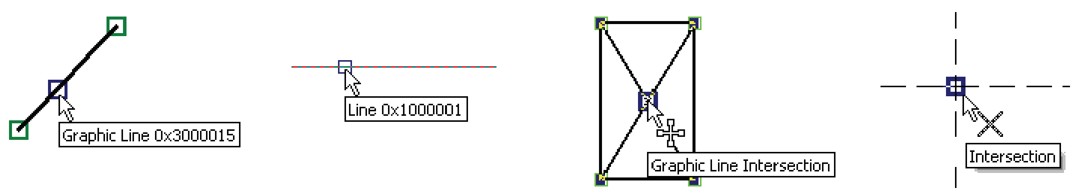
For displaying this toolbar in an "independent" mode, move the cursor to the title area of the toolbar, press and, without releasing the mouse, drag the toolbar into the desired location. In the future this toolbar can be left in the floating mode or snapped at any place of the T-FLEX CAD window.

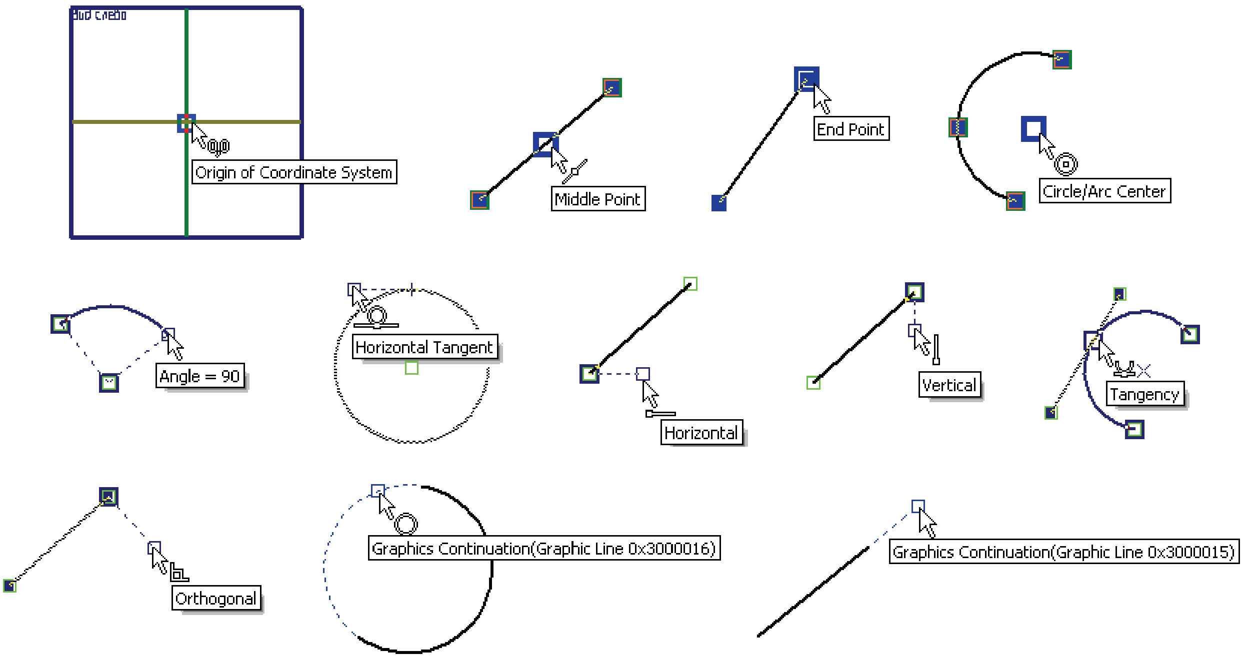


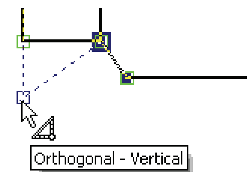
Using this toolbar, one can set and unset the snapping modes by clicking the desired icons with . All snappings can be simultaneously turned on or off by the button  - “Clear all sketch Snaps”. Also, all snapping modes except the required one can unset by clicking appropriate icon with **`<Ctrl>`** button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

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

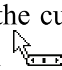
- Snapping to a point on a graphic line or construction line – , ;
- Snapping to graphic line intersection – ;
- Snapping to construction line intersection – ;
- Snapping to the coordinate system origin ((0,0) point) – ;
- Snapping to the midpoint of the graphic line – ;
- Snapping to graphic line end 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 – .

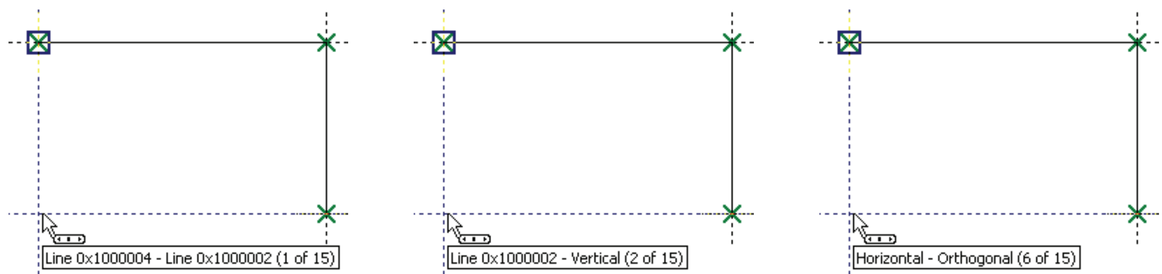




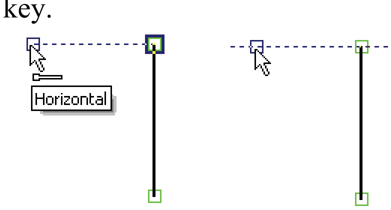


In the creation/editing process, the system automatically finds the allowed snaps and offers them to the user (by flashing a snap type next to the cursor). Besides that, the system monitors for a coincidence of two object snaps, for example, vertical – horizontal, perpendicular – horizontal, etc.

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.



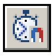
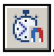
A system-offered object snap can be locked by the ***<Spacebar>*** function key.



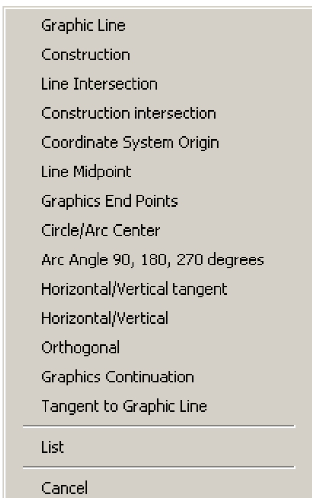
For example, let's fix horizontal snapping to one of the segment nodes. To do this, get horizontal snapping with this node and press the key ***<Shift>*** or ***<Spacebar>***. A temporary dotted line will be constructed through this node, the cursor sliding along as a free node.

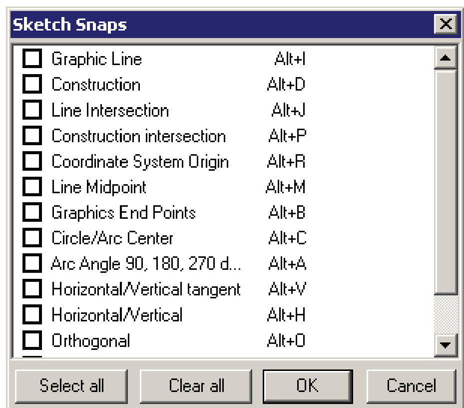
Snapping that are turned on on the sketch snapping toolbar stay active continuously throughout the sketch command session. If snappings are adjusted often, one can use temporary object snappings -the “one-action” snappings.

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.

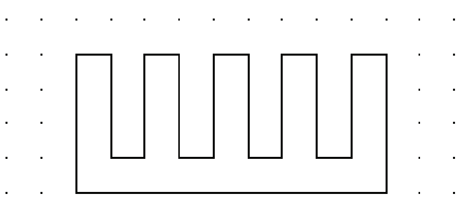
When temporary object snapping is turned on, all permanent snappings are ignored. The described temporary snappings act until the first click .

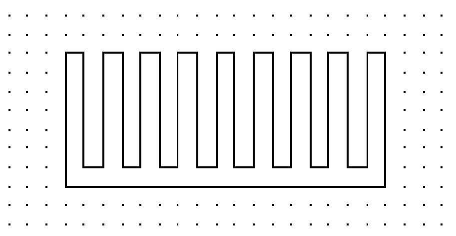




Using Grid

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.

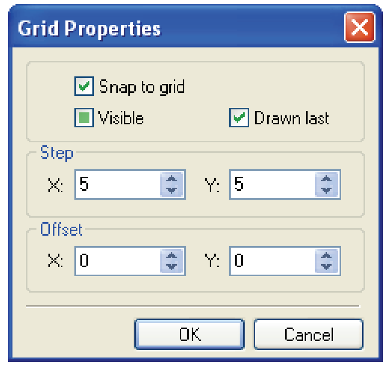




The grid can be turned on for the active page by the command **"QG: Change Grid Settings"**:

Keyboard	Textual Menu	Icon
<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.

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

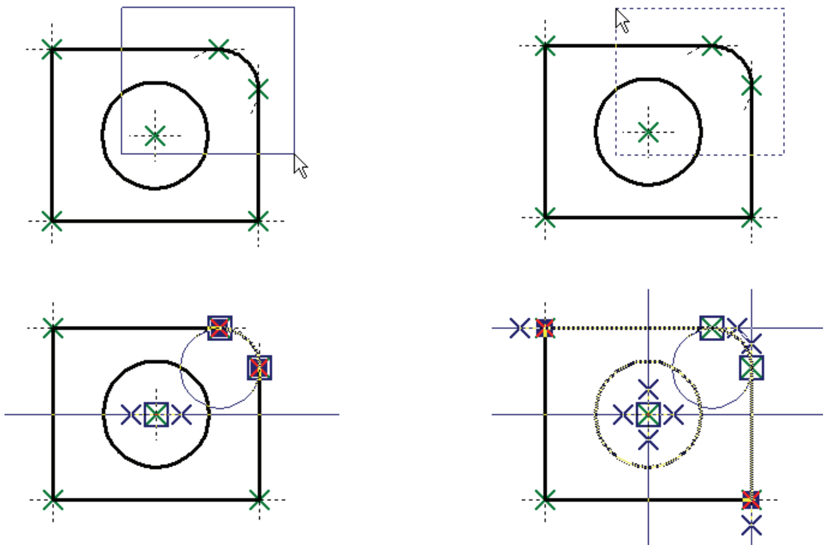
In editing commands, element selection is done by the cursor. To select, move the cursor to the element and click or press ***<Enter>***. Different elements are highlighted in different ways. Some are painted with colors, others surrounded by a frame. To relocate a selected element, move the cursor to 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, then all elements are selected that are fully within the specified region. In this case, the selection box is drawn on the screen in a solid line. If the cursor was moved right to left, then objects are selected by the crossing frame. That means, 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 drawn on the screen in a dashed line.



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 .

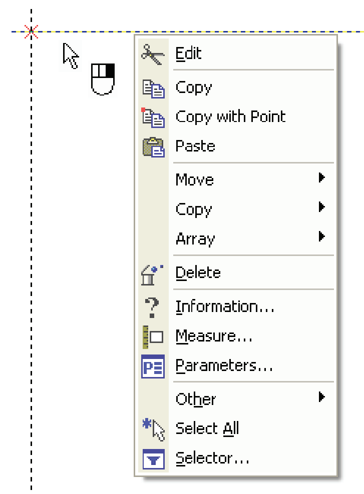
Selecting elements outside any command

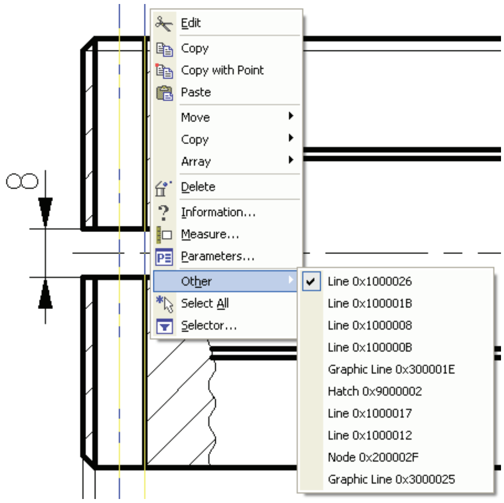
Elements can be selected for editing even outside any command, when the system is in the command-waiting mode.

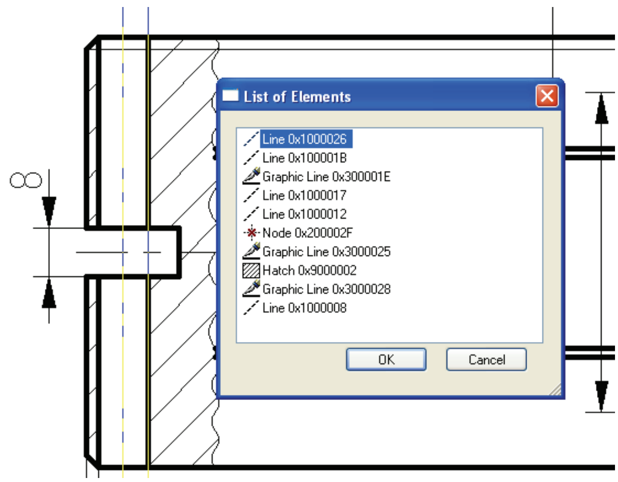
Selecting an element with automatically starts the given element editing command. Double-clicking will start the editing command and bring up the element parameters dialog box.

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.

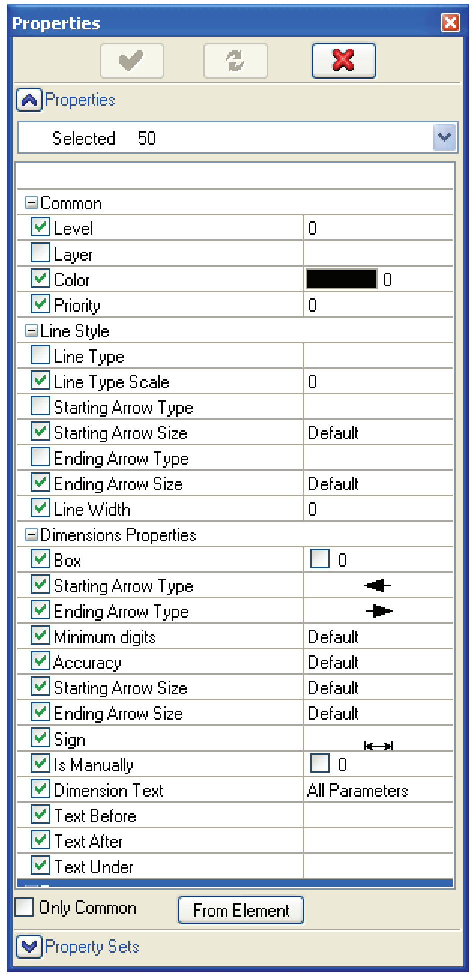
Changing various type element parameters outside any command

Use the property 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 properties of the selected element. The dialog is inactive by default. To activate the dialog, enter the property window and expand the group **"Properties"**. 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 **"Properties"** command in the context menu.

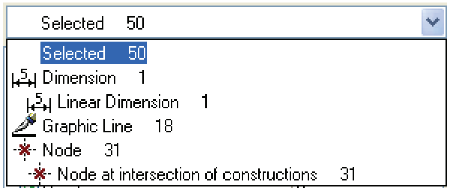
To turn off the active mode of the dialog, close the "Properties" group.

> Note: upon single element selection, the **"Properties"** 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.

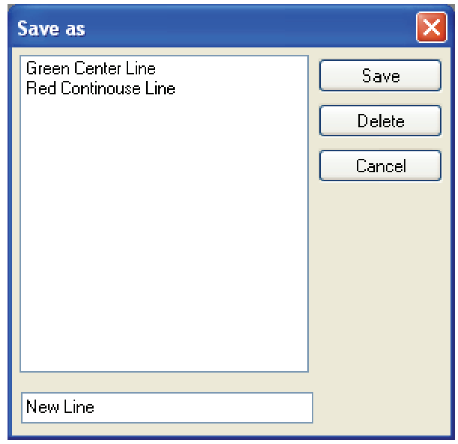
To change properties of the elements, check the desired properties in the table, enter the required values in the cells on the right-hand side, and press the  or  button in the upper part of the dialog box.

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.

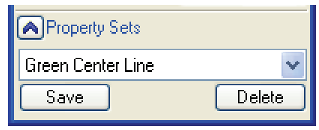
The  ("Cancel edit") button can be used to abandon the entered changes and finish processing the selected set of elements. Abandoning changes and finishing the selected element set processing can also be done by simply clicking  within the drawing area.

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.

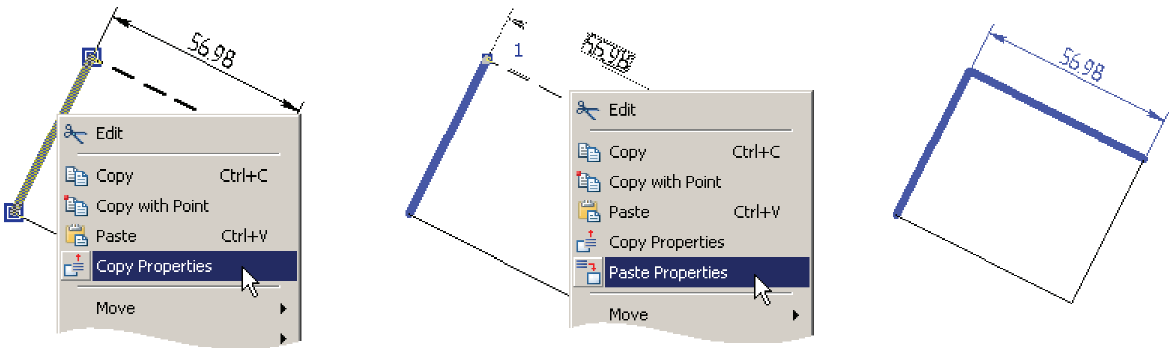


The name of the set to be saved is entered in a box in the bottom part of the dialog box. Upon entering the name, press the **`[Save]`** button. The "Save as" dialog box will close, and the saved set name will appear in the pull-down list. The **`[Cancel]`** button closes the window without saving the 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

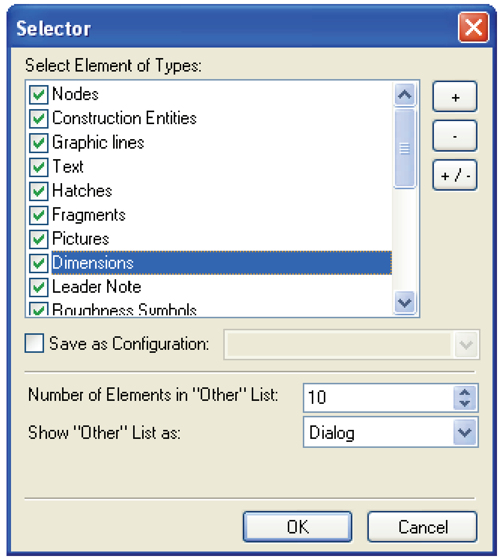
When working with a dense drawing, it is often difficult to select the desired element on the screen. In this case, it may be necessary to limit the list of the elements available for selection. This can be done in several ways. Some of them, such as using the level and layer mechanisms, were already 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:

Keyboard	Textual Menu	Icon
<FT>	**"Edit|Selector…"**	

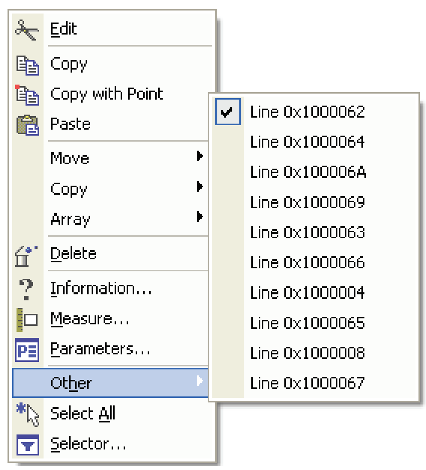


Upon calling the command, the selector configuration dialog box comes up on screen. The main field of this dialog, **"Select Elements of Types"**, contains the list of all system element types. The elements allowed for selection are checkmarked at the left of their type names. By default, all elements 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.

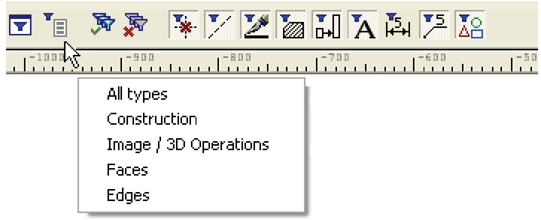
A specified combination of settings can be saved as a named selector **configuration**. To do so, check the **"Save as Configuration"** item and enter the name for the new configuration in the box on the right-hand side.

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.

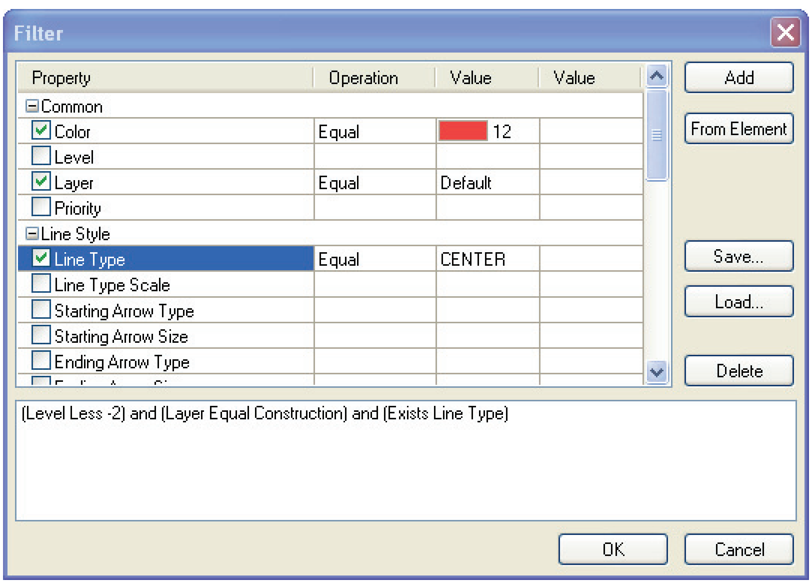
Filter

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:

Keyboard	Textual Menu	Icon
<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.

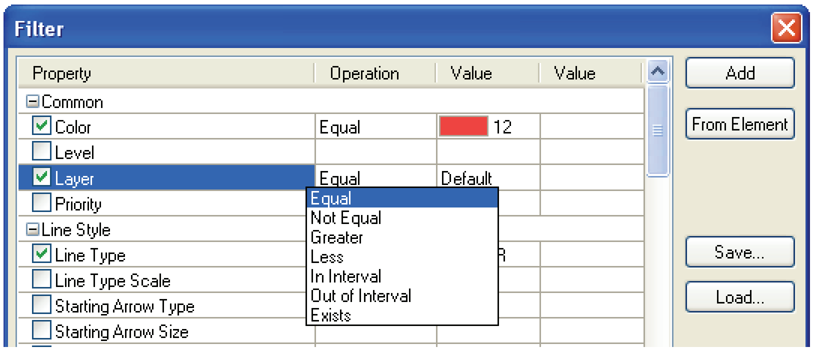
Calling the command brings up the filter parameters dialog box.



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, checkmark this parameter in the box before the parameter name in the table using . The limitation type for this parameter value will automatically appear in the **“Operation”** column, by default, **“Equal”**. If necessary, change the type by clicking on the type and selecting any other entry in the coming pull-down list, namely, **“Not Equal”**, **“Greater”**, **“Less”**, **“In Interval”**, **“Out of Interval”**, **“Exists”**.

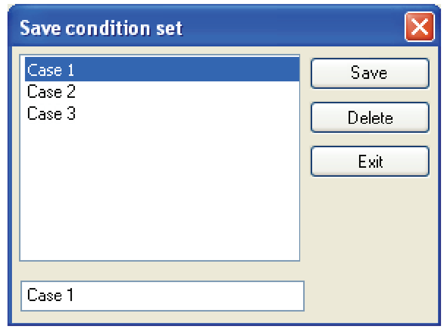


If the limitation type requires entering a value or values of the parameter, do this in the **“Value”** columns. Thus, for “Equal”, enter a value to compare against the element parameter value. Use the first “Value” column for this. For limitations requiring two values, as for “In Interval”, fill in both columns by entering the starting and the ending values of the interval.

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.

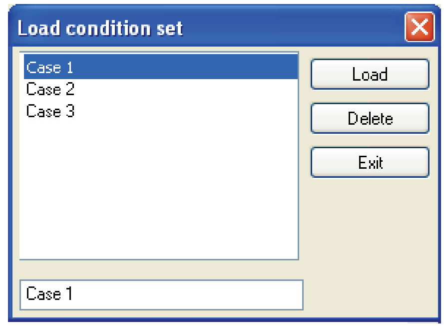
When creating a condition, the parameter values can be read from a specific element. To do so, checkmark the necessary properties, and then press the **`[From Element]`** button. The dialog box will temporarily disappear from screen, making possible selection of the desired element in the drawing window 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.



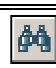
To use an earlier saved condition set, press the **[Load…]** button. After pressing the button, the "Load" dialog box will come up on screen for loading the named condition set. Working with this dialog is similar to the set saving dialog. The upper pane of the dialog contains the list of available sets. Use  to select the desired set from the list. The name of the selected set is displayed in the lower pane of the dialog.

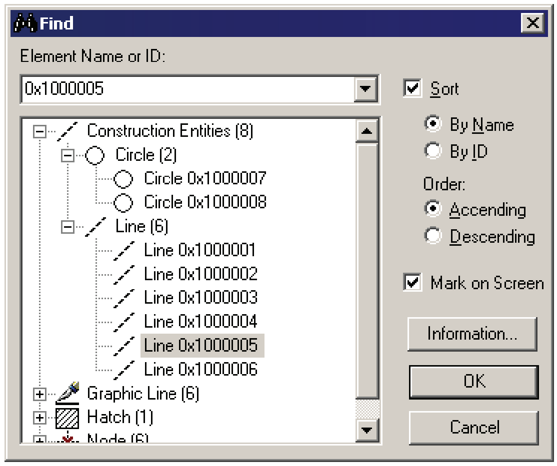
Once selection is done and the **[Load]** button pressed, the dialog closes and the contents of the selected set are added to the list of the existing sets of conditions. This dialog also allows deleting any of the existing named sets using the **[Delete]** button.

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"**:

Keyboard	Textual Menu	Icon
<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 (see the topic "Model tree"). 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.

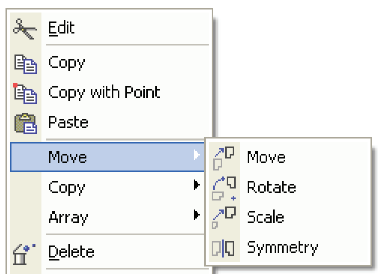
Another way of searching for an element is using the tree in the main pane of the dialog box that contains all model elements. When an element is selected in the tree, the upper input box displays its Id or name. The buttons in the right part of the dialog become accessible as well.

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 inavoidably 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 “Preferences” 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:

Keyboard	Textual Menu	Icon
<UN>, <Alt><BackSpace>, <Ctrl><Z>	**“Edit|Undo”**	

The **“RED: Redo Changes”** command is called via:

Keyboard	Textual Menu	Icon
<RED>, <Ctrl><BackSpace>, <Ctrl><Y>	**“Edit|Redo”**	

To cancel or repeat several actions at once, press the button  on the main toolbar 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 Model Status”**. 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 Model Status”** 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 Model Status”** 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 Model Status”** 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}`

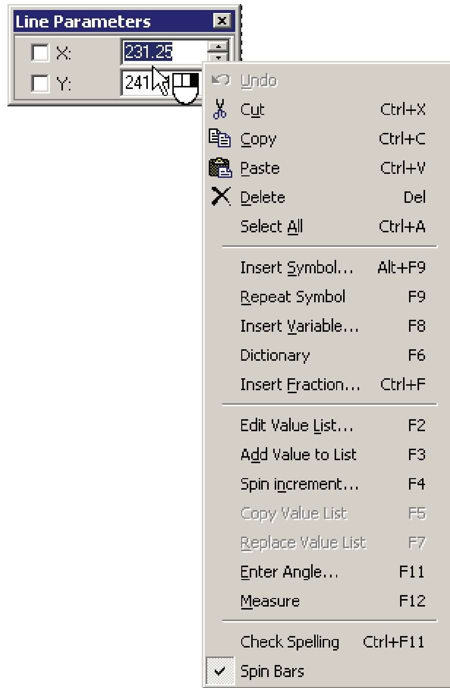
Note that the textual variable $MONTH begins with the ‘$’ character, as this is the prefix for all textual variables.

> 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.

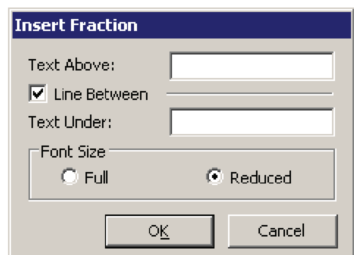
Insert Symbol… *<Alt+F9>*. Inserts a symbol from a special symbol table. The symbol code is actually entered in the input box instead of the symbol itself, for example, %%066 for the diameter symbol.

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.

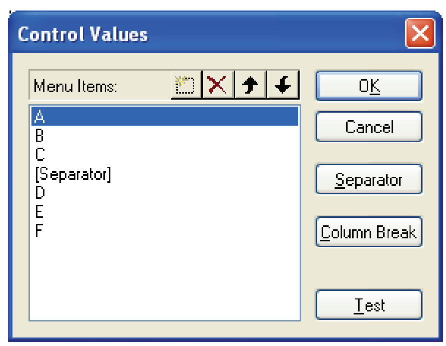
Insert Variable… *<F8>*. Inserts an existing variable from list. The variable name is inserted in the input box in braces. The drawing will display the actual value of the variable. The variable values can be changed in the variable editor or, in some cases, directly on the drawing (see the section **"Paragraph 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.

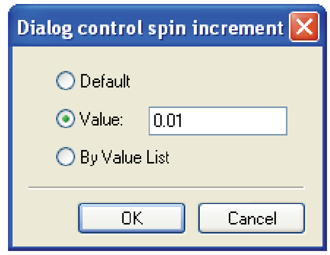
Insert Value to List *<F3>*. This command adds the current value from the dialog input box into the list. If the list did not exist, it will be created.

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"**.

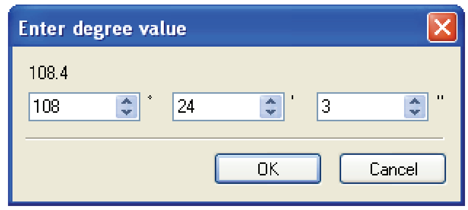
Spin Bars. This command enables the stepper – the way to modify the parameter in the respective field using the mouse wheel or the button .

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".

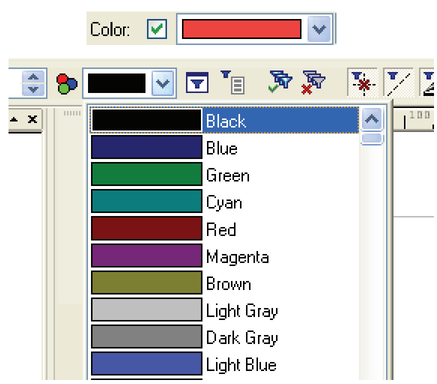
Check Spelling. ***<Ctrl+F11>***. Checking the spelling of the content of the dialog field, for which the context menu has been called.

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.

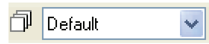
An element color can also be set using the system toolbar.

Setting colors via the system toolbar is available in creation and editing commands.

Layer

A layer is a parameter of any drawing element. It defines the element association with a particular group of the model elements.

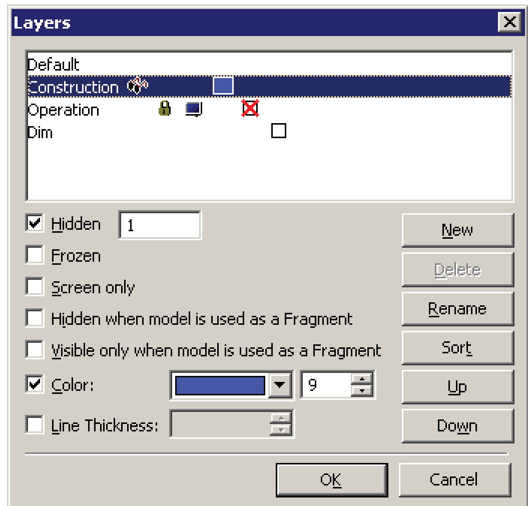
The user can define the layer name for each system element to belong to. A layer name is a string of up to 20 text characters.



An element layer can also be set on the system toolbar.

Layer parameters can be created, deleted and modified using the command **"QL: Configure Layers"**:

Keyboard	Textual Menu	Icon
<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 buttons **`[Sort]`**, **`[Up]`**, **`[Down]`** are used for changing the arrangement of the layers in the list. The layers arrangement is enforced in all dialog boxes of the system which allow selection of the 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. By entering layer parameters you define the properties of the elements belonging to this layer. 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.

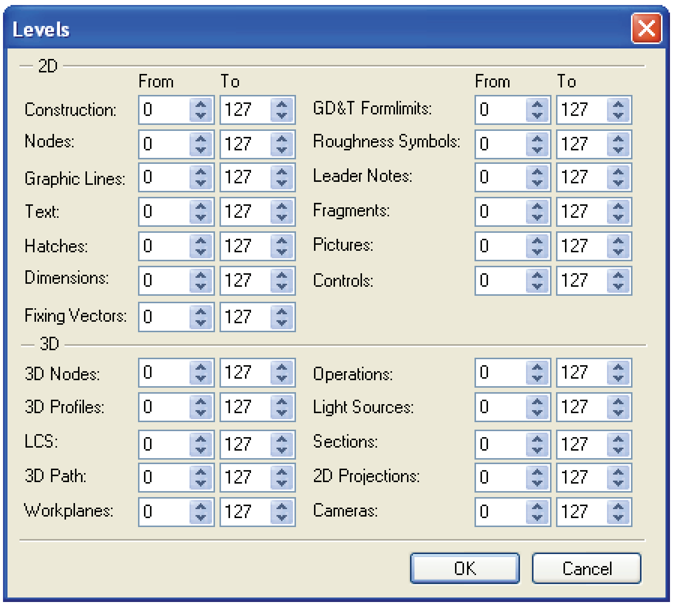
Level

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"**:

Keyboard	Textual Menu	Icon
<SH>	**"Customize|Levels…"**	

After calling the command, a dialog box comes up for specifying the ranges of element levels.



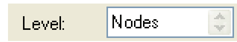
The level visibility range is defined by two numbers in the range from -126 to 127 for each element type. An element visibility upon redraw is defined in the following way:

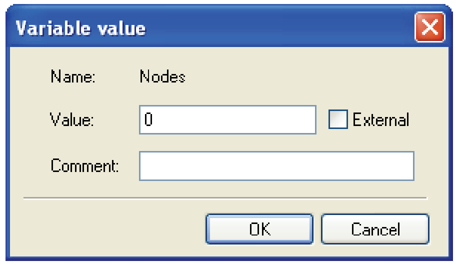
If the element level value is within the range defined for this type of elements, then the element will be displayed upon redraw.

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.



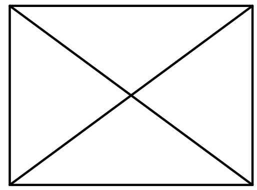


When using a variable for defining a level, enter the variable without braces, for example, `LEVEL1`

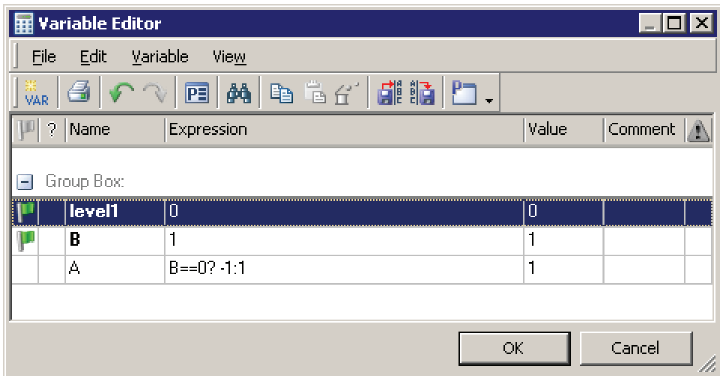
After exiting the parameters dialog box of the given element, another dialog box will come up on screen for setting the value of the variable `LEVEL1`.

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.

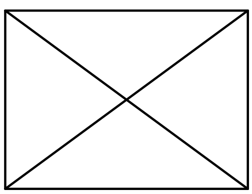


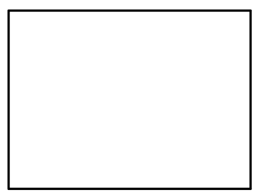
Set the level of the rectangle diagonals using the variable "`A`". Set the value of the "`A` variable" equal to "`1`". In the command **"SH: Set Levels"** set the visibility range for the graphic lines from `0` to `127`. In the variable editor create a variable "`B`" with the initial value "`1`". Enter 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.

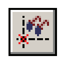
An example of using hidden line removal could be any assembly of co-axial parts, created from fragments. In this case, the fragment parts are created without hidden line removal required in the assembly. Simply set their correct priorities when assembling.

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"**:

Keyboard	Textual Menu	Icon
<SI>, <Ctrl><Shift> <C>	**"View|Hide Construction"**	

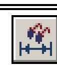
The command hides all construction elements in the current window (the 2D view or the 3D view). A second call to the command restores the construction element display on the screen.

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"**:

Keyboard	Textual Menu	Icon
<SN>	«-»	

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"**:

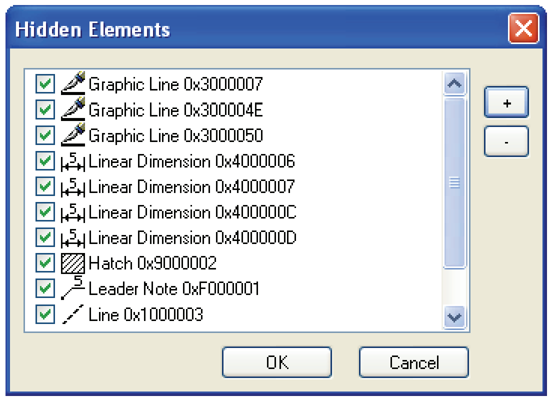
Keyboard	Textual Menu	Icon
<ESO>	**"View|Hide/Show Elements"**	

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.



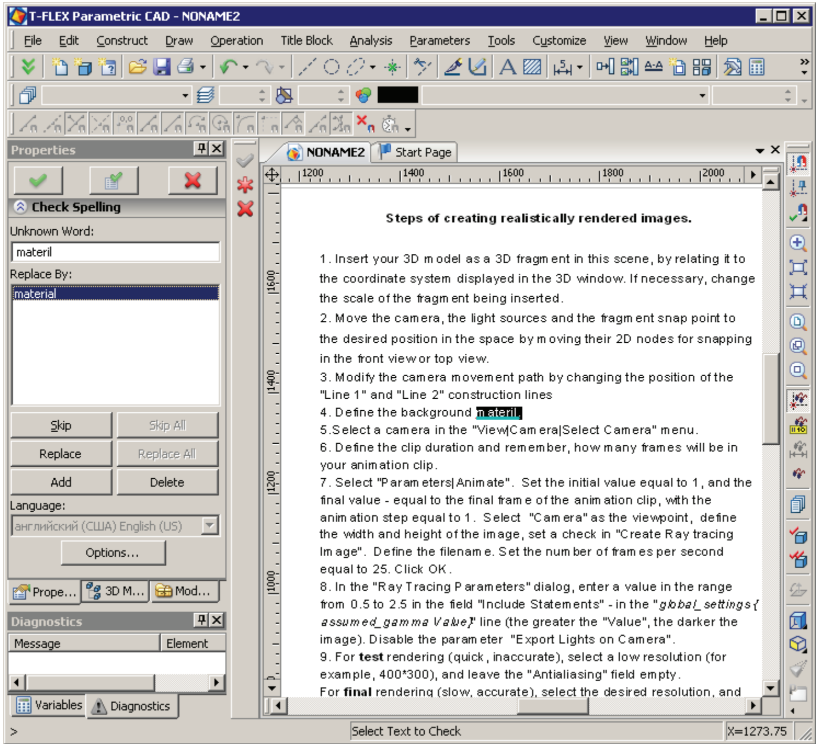
The option brings up a window with the list of all hidden elements.

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.

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:

Keyboard	Textual menu	Icon
<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.

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.

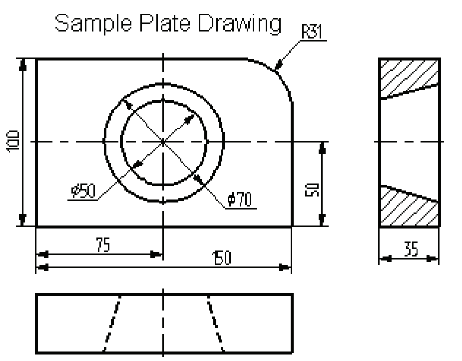
It takes somewhat more time resources to create a parametric drawing; nevertheless, later on such drawing will be easily modifiable as you desire. A nonparametric drawing (sketch) can be created faster. Its creation method is similar to the ways of drawing in some other CAD systems. However, nonparametric 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

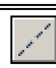
The following diagram shows a drawing to be created. It is a plate with a through hole of conical shape. The drawing will be defined parametrically so that any modifications will automatically reflect on all projections.

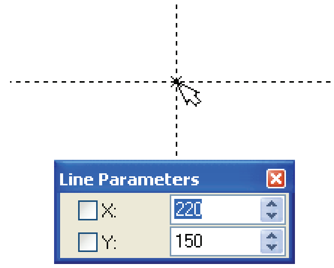


Let’s begin with the main (elevation) view of the plate. First, create the necessary “thin” construction lines, and then draw the graphic lines on top. Next, create the other two views using the construction lines of the main view. This creates a dependency between the views so that the two views automatically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

Keyboard	Textual Menu	Icon
<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.

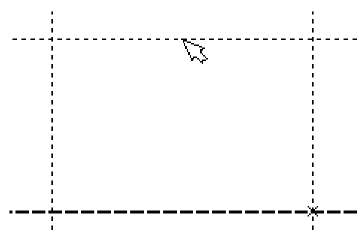
> Do not use more than two base lines on the main (independent) view, and more than one base line on the views defined by projections. This will insure freedom in placing the drawings.

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.

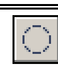
Pressing now starts rubberbanding of a line that follows the cursor while staying parallel to the selected one. We are now creating a line parallel to a vertical line. Such a relationship between the two construction lines, established at the creation time, is an example of an important feature of T-FLEX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

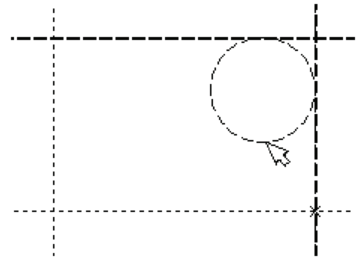
Place the new line at the left of the highlighted vertical line by pressing . The exact value of the distance can be specified in the property window or parameter dialog box. The newly created line will become the left-hand side of the part.



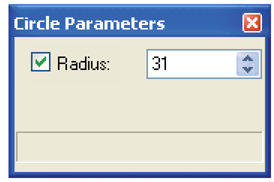
Pressing once cancels the parallel line creation mode, yet the line creation command stays active. (Otherwise, call it again.) Next, move the cursor toward the horizontal line and press . The line is selected as a reference for a parallel line to be created. Move the cursor up, specify an exact value 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

Keyboard	Textual Menu	Icon
<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,

Keyboard	Textual Menu	Icon
<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"**,

Keyboard	Textual Menu	Icon
<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"**:

Keyboard	Textual Menu	Icon
<PU>	**«Edit|Purge»**	

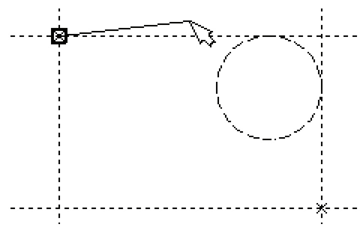
This command will delete all construction entities and allow to start drawing anew. A specific construction entity can be deleted using command **"EC: Edit Construction"**:

Keyboard	Textual Menu	Icon
<EC>	**«Edit|Construction|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

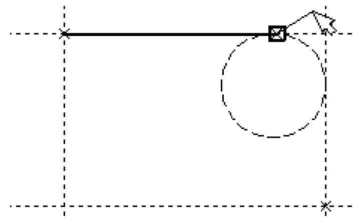
Keyboard	Textual Menu	Icon
<G>	**«Draw|Graphic Line»**	



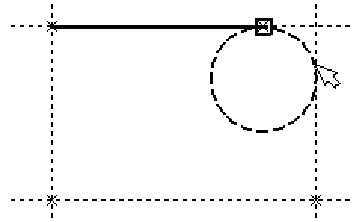
Note that the previous command is automatically terminated when calling another command via the toolbar icon button or the textual menu (no need to cancel the previous one explicitly).

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.

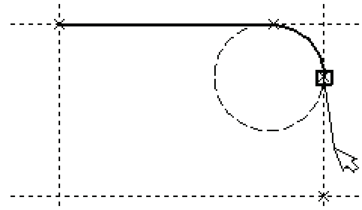


Move cursor to the tangency point between the top line and the circle, and press . What you see on screen now should be similar to the illustration at right. Note that T-FLEX system automatically adds nodes to the end points of the graphic lines, unless already created.

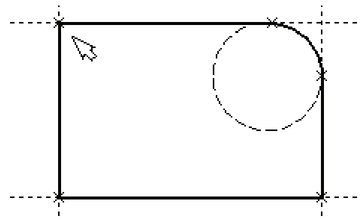


Now let's draw a graphic line along the circle to construct an arc between the two tangency points. To do so, move the cursor to the circle and press ***<C>*** key. The circle will then get highlighted. The direction of arc creation depends on the position of the cursor when selecting the circle. To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

Keyboard	Textual Menu	Icon
<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"**:

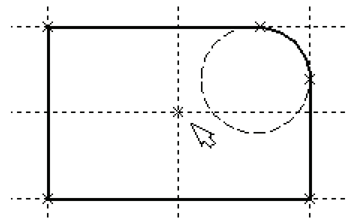
Keyboard	Textual Menu	Icon
<EC>	**«Edit|Construction|2D Construction»**	

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.

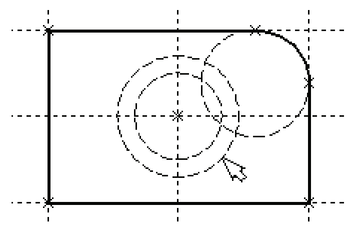
Call the command **"L: Construct Line"** and choose the icon in the automenu. Move the cursor to the right-hand side of the plate and select the vertical line by . A parallel line appears rubberbanding after the cursor. Move the cursor to the left-hand side of the plate without fixing the rubberbanded 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.



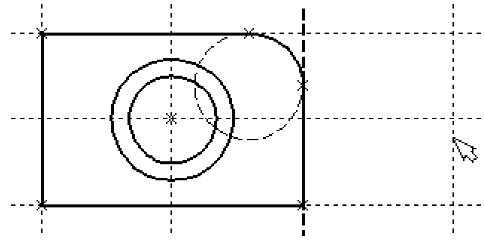
The second circle of the conical hole can be constructed as concentric to the first one. To do so, pick the icon in the automenu or type ***<O>***. Then select the existing circle with . The new circle starts rubberbanding after the cursor. Place the cursor so that the rubberbanded circle is slightly 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.

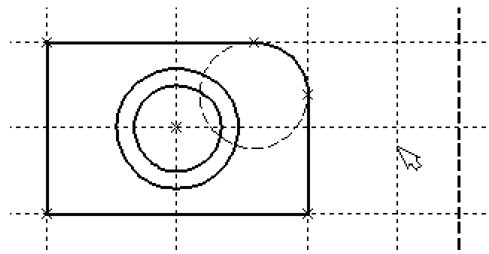
Activate the line creation command and move the cursor to the construction line defining the right-hand side of the plate. Select it with . This highlights the vertical construction line and starts rubberbanding of a new line parallel to the selected one. This line will be the right-hand side of the 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.

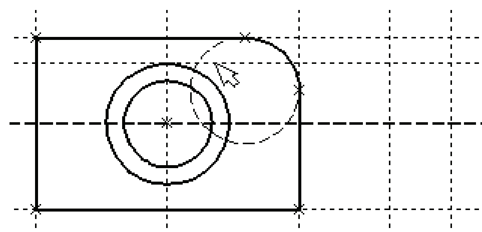
The next step is creating the line of the left-hand-side edge of the part on the side view. After completing one line, a new line rubberbanding began automatically.

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.

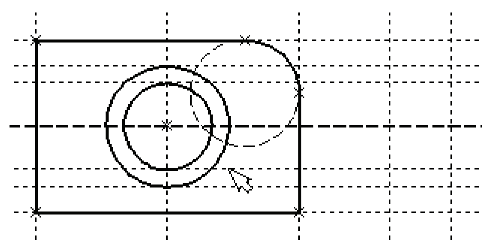


We recommend using specifically the right-hand side of the part main view as the base line, and construct all the rest vertical lines with respect to it. In this way, the line-to-line distances will be positive which is preferable in some situations.

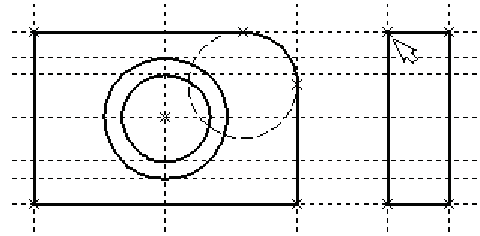
Now let's proceed with constructing the projection of the conical hole. First, let's create horizontal lines tangent to the top and bottom of the inner and outer circles of the hole. These lines will be used as guides for the side view of the hole.



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.

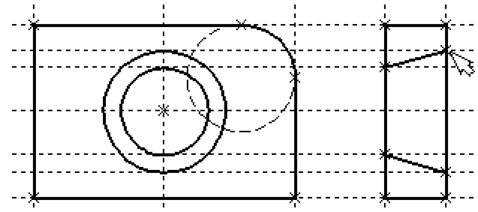


Repeat the same sequence of actions three more times: for the top tangency with the inner circle, and the two bottom tangencies.



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 .



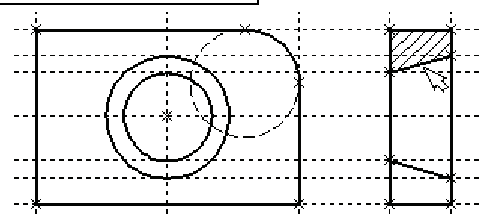
Next, apply the two lines representing the conical hole. The view is now almost complete, with only the hatch yet to be created.

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

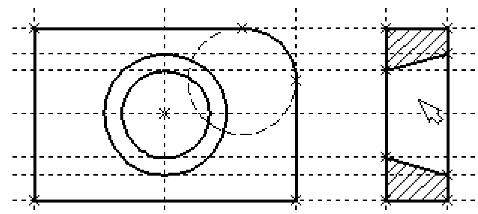
Keyboard	Textual Menu	Icon
<H>	**«Draw|Hatch»**	

Set the following option, unless on by default,

	<A>	Automatic Contour search mode



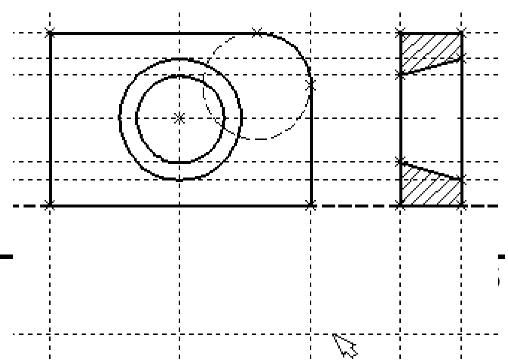
Then move the cursor to the top portion of the side view, place within the area to be hatched, and press . The top contour gets highlighted. Next, type ***<P>*** to invoke the hatch area parameters dialog. Specify the type and scale factor of the hatch. Pressing in the automenu completes hatching of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



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

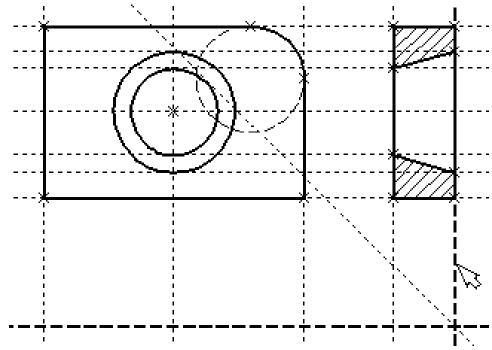
It is also possible to create a single hatch consisting of two contours, instead of creating two separate hatches. To do so, one could select the second contour right after the first one, and then press ***<End>*** key or pick icon in the automenu.

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 .

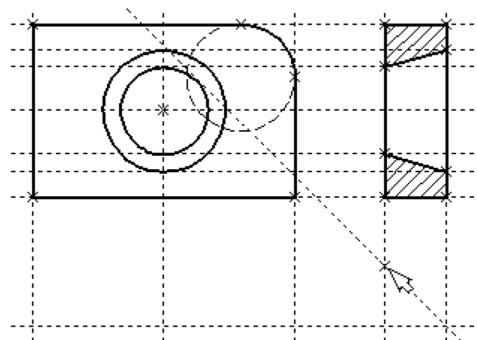


Let's try creating the plan view in such a way that modifications of other views propagated on the plan view via the established dependencies. The simplest way to create a dependency in projective drawing is creating a slanted line at 45-degree angle to the side lines of the side and plan view. The rest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

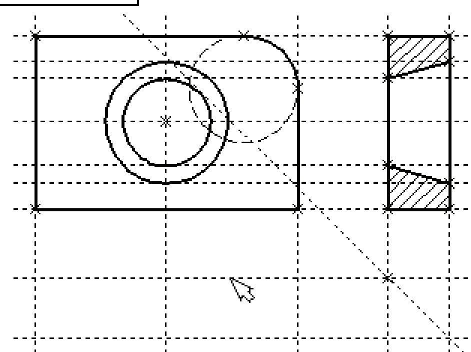


Let's create all necessary nodes at intersections while within the line creation command. The relevant intersection points are those on the right-hand-side line of the side view and the newly created slanted line. To create a node, place the cursor at an intersection and press the ***<Space>*** bar.



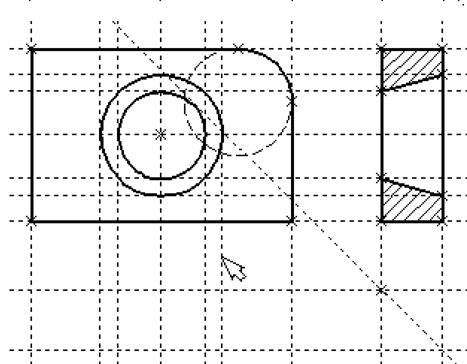
Another way of creating nodes is using command **"N: Construct Node"** via:

Keyboard	Textual Menu	Icon
<N>	**«Construct|Node»**	

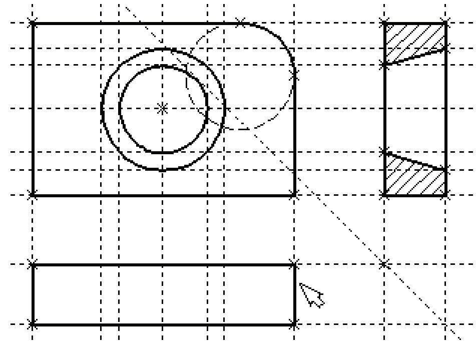


You should still be within the command **"L: Construct Line"**. Point the cursor at and select the bottom line of the plan view. This way we can create a line parallel to the bottom-side one. Now, move the cursor to the newly created node and type ***<N>***. This creates a line parallel to the selected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

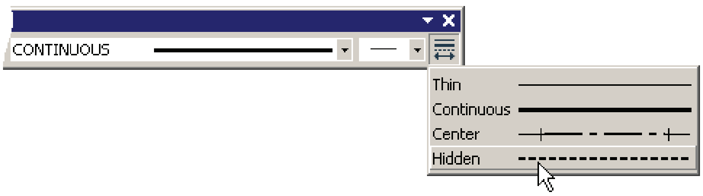


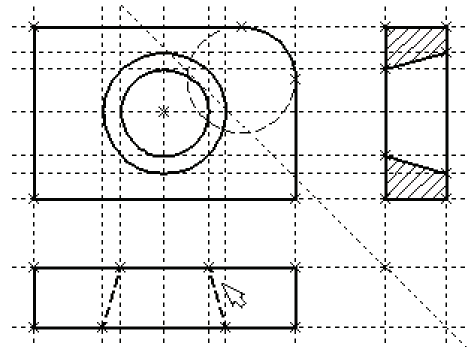
Construction of the conical hole on the plan view follows the same steps as on the side view. Select a vertical line while in the construction line creation command, and create four parallel lines tangent to the two circles.



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.

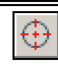
Next step is to apply the two dashed lines corresponding to the conical hole. Set the "HIDDEN" line type in the system toolbar.



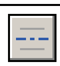


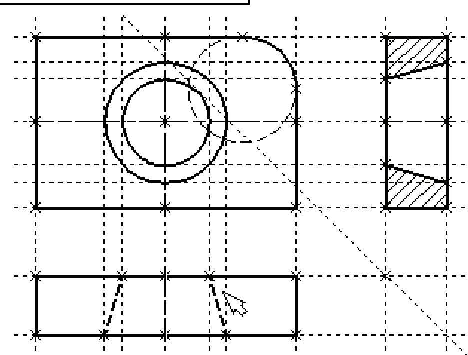
Then create the two dashed lines representing the conical hole.

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

Keyboard	Textual Menu	Icon
<AX>	**«Draw|Axis»**	

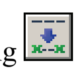
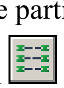
Set the automenu option:

	***<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.

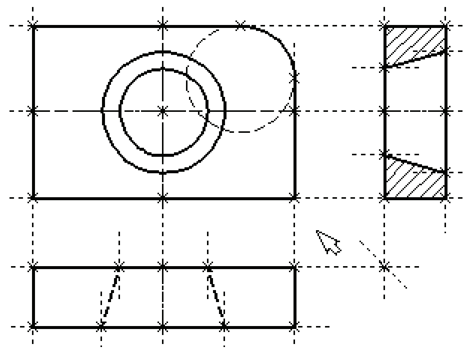
One could notice that all construction lines created so far were infinite. For convenience, an option is provided in the command **"EC: Edit Construction"** for trimming construction lines at outermost nodes. This works as follows,

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 Model Status”**:

Keyboard	Textual Menu	Icon
<ST>	**«Customize|Status…»**	

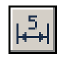
Go to the parameter **«Screen|Construction Lines|Length»** and set it to “Default infinite”. Alternatively, enter the command **“EC: Edit Construction”**, select desired lines, type ***<P>*** and specify an appropriate setting.



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:

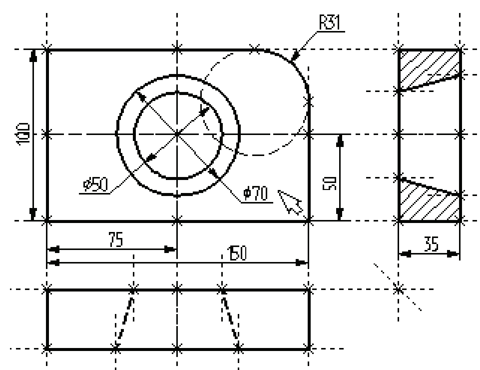
1. First, let’s create linear dimensions. Call the command **“D: Create Dimension”:**

Keyboard	Textual Menu	Icon
<D>	**«Draw|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 Model Status”**, the tab **“Font”**. 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.

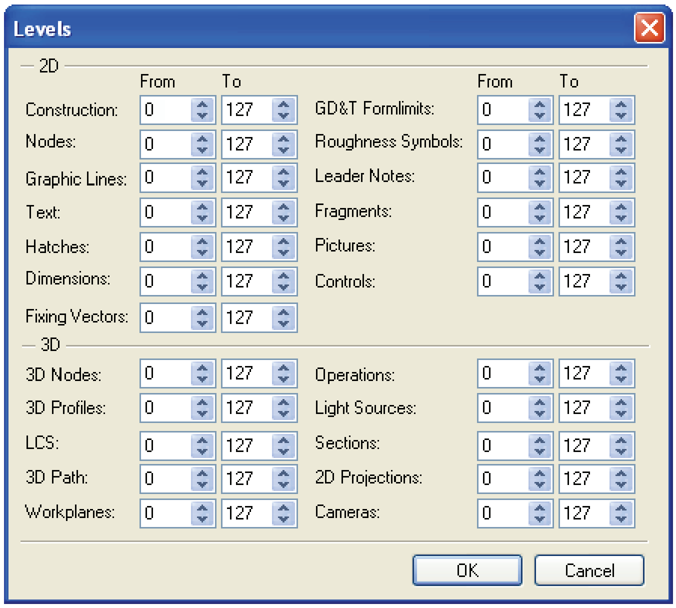


4. After finishing construction of all major elements, one can hide all construction entities using the command **"SH: Set Levels"**:

Keyboard	Textual Menu	Icon
<SH>	**«Customize|Levels…»**	

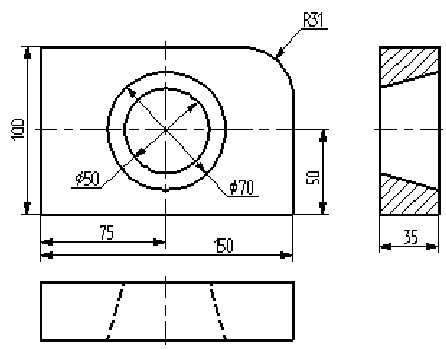
This command controls visibility of various elements. An element visibility depends on the "level" at which it is residing.

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

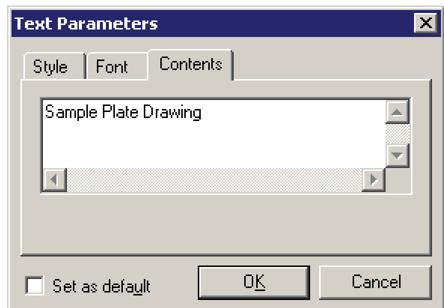
Keyboard	Textual Menu	Icon
<TE>	**«Draw|Text»**	

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

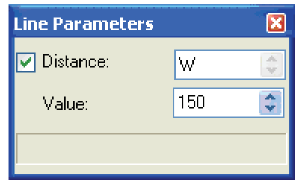
Keyboard	Textual Menu	Icon
<ET>	**«Edit|Draw|Text»**	

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.

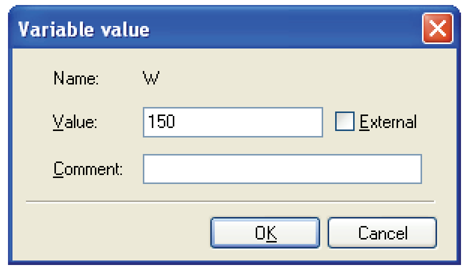
The drawing is now finished. One can try moving construction lines using construction editing command. When editing, fix line new placement by either using or specifying exact line location in the property window or parameters dialog (the latter accessible via the pick). Note that the whole drawing, 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 .



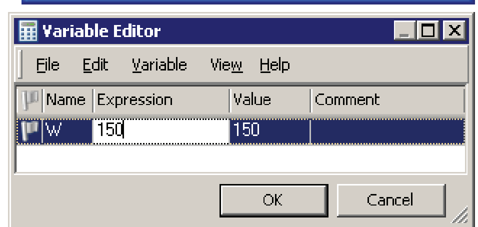
The line will get highlighted, along with the one it is dependent on by construction. Line editing command will automatically activate as well. The two parameters are displayed in the property window. The first one is the original distance, and the second is the current value according to the cursor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

Keyboard	Textual Menu	Icon
<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`".

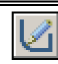
Check this on your drawing. Try setting "`W`" values greater or less than 100, and watch what's happening. Note that when the radius of the fillet equals `0`, then the radial dimension automatically disappears. The system does it for the user.

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.

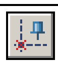
Creating Sketch, Non-parametric Drawing

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.

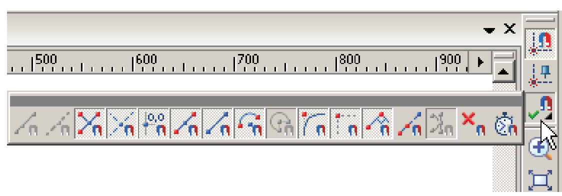
In this case, all construction is done using the command **"SK: Create Sketch".** Call the command via

Keyboard	Textual Menu	Icon
<SK>	**«Draw|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

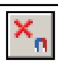
	Automatic parameterization on/off

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".

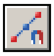
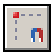
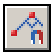
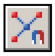
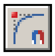


All snaps turned on by the present moment correspond to the toolbar icons which have been pressed.

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

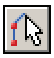
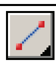
Unsetting this option sets all snaps on. In our exercise, the following snaps will be used:

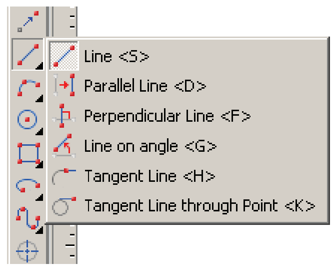
	Line Midpoint
	Horizontal / Vertical
	Orthogonal
	Line Intersection
	Horizontal/Vertical tangent

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

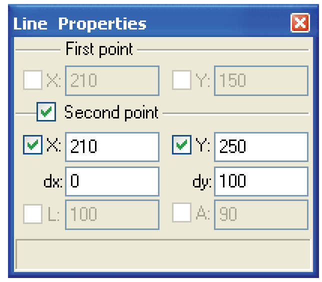
> Attention: the automenu may display any of the encapsulated option icons in the given position. Usually this is the icon of the last used option among the encapsulated set.

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.

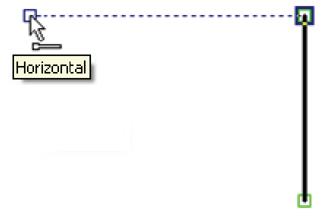
> While sketching, consider leaving sufficient margins. This space will later be used for creating dimensions.

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.

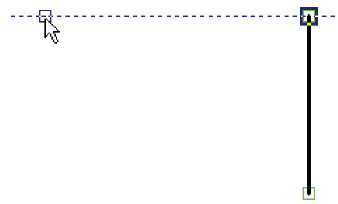


Let’s create the end of the segment by specifying its shift from the start point. Make “dx” value equal to 0, and “dy” parameter equal to 100. The parameters “X” и “Y” will instantly update with the absolute coordinates of the segment end and get checkmarked. Checkmarking locks the value of the respective 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”.

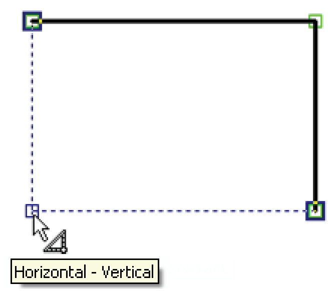


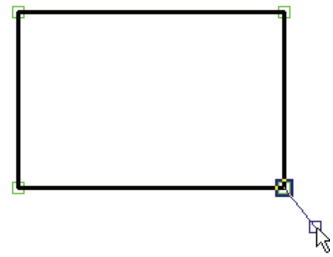
To lock this snap, press ***<Space>*** bar. Then, a horizontal helper line will be displayed passing through the node snapped to. The cursor will keep sliding along this helper line as an unattached node. The same effect can be achieved by setting the “dx” shift to 0 in the property window and locking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

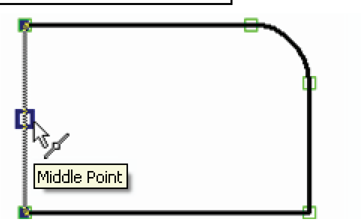
What is left now is to select two segments to be filleted. In our case, it is the top and the right-hand-side segments of the plate. Once the second segment is selected, the fillet is created, and the segments trimmed appropriately.

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

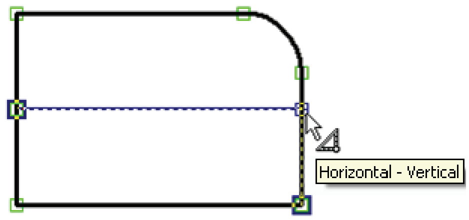
	<S>	Line

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

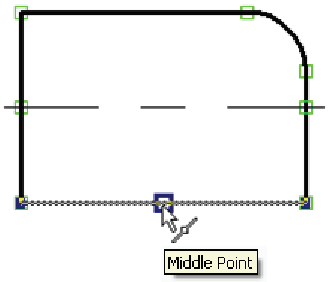


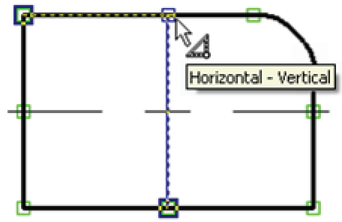
Then move the cursor to the left-hand-side graphic line segment to get vertical snapping to one of the segment nodes, and slide the cursor along the segment to its midpoint. When the cursor reaches the midpoint, the glyph beside the cursor will change to indicate this, along with the pop-up help.



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.

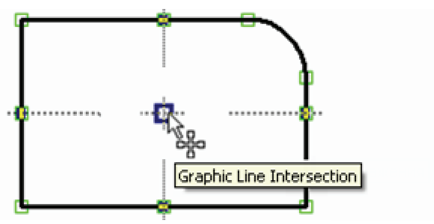




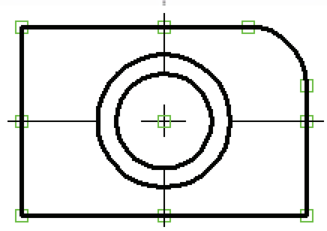
Now, let's create circles. First, reset the line type to CONTINUOUS by selecting in the system toolbar or in the command parameters dialog box. Call the dialog by typing ***<P>***. Then pick the option:

	<O>	Circle By Center And Radius

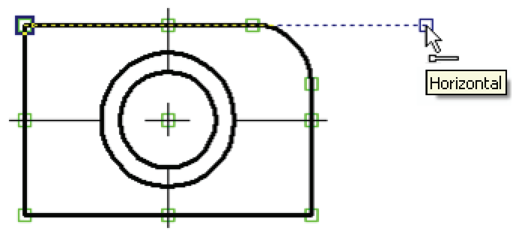
This is also an encapsulated option that may not be shown on the automenu, rather, be within a group of options.



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.

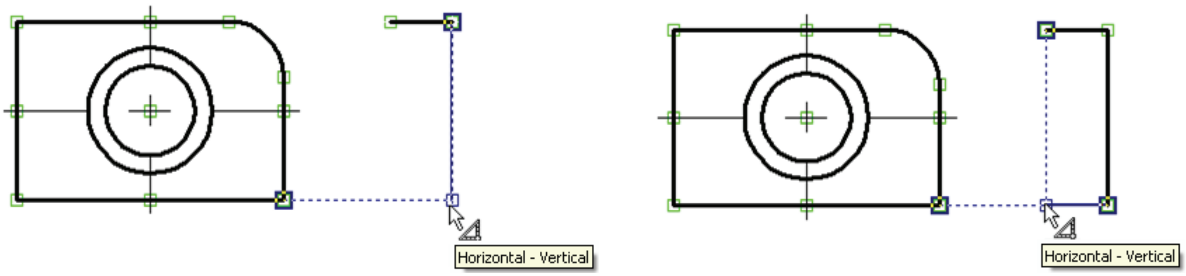


Enter the value of the radius of the smaller circle of the cone equal to 25 in the property window, and press **`[Enter]`** key. A full circle is now fixed on screen. Now you are still in the circle creation mode. Select the node at the two centerlines intersection and create a circle of a bigger radius,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

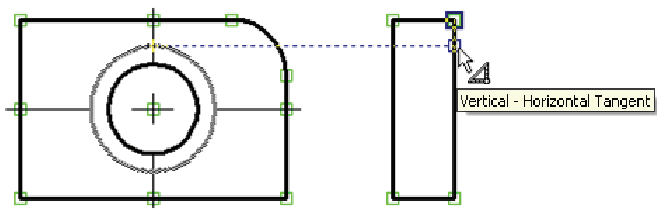
Click here , and move the cursor horizontally rightwards. In the property window, set the end node shifts. Set the X equal to 35, Y to 0. Press **[Enter]** or . The new segment will be fixed on screen, and rubberbanding resume from the last created node. Next, move the cursor vertically downward maintaining 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.



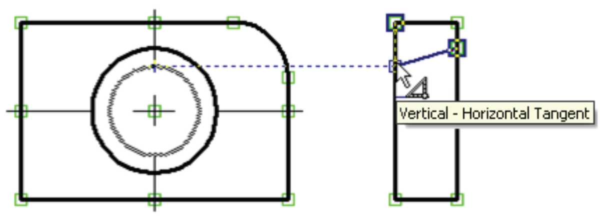
Click . Now close the perimeter of graphic lines by moving cursor to the first created node on this view, and clicking , and then .

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

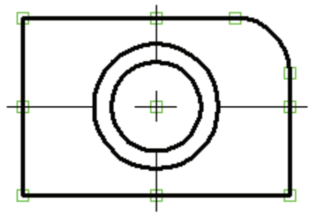
Next, we need to create the image of the conical hole on the side view. Without quitting the current command, move the cursor to the right-hand-side segment of the side view, and move along the segment until it snaps to horizontal tangency against the top of the bigger circle



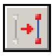
Click at this spot, then move the cursor to the left-hand-side segment of the side view and locate so to get it snapped against the smaller circle.



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.

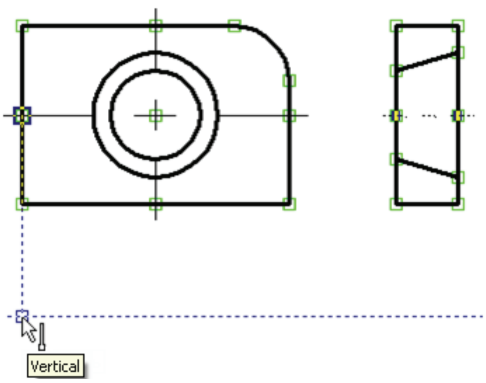


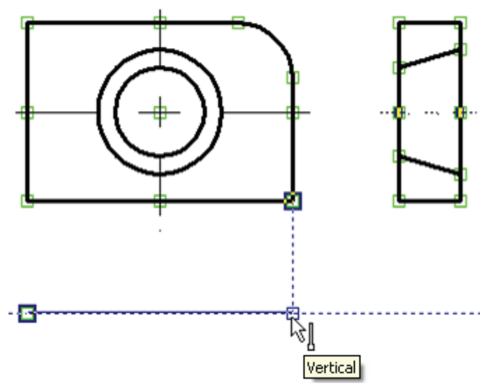
Proceed with the plan view. This view can be created in the same way as the side view. However, for deeper exploration of non-parametric drawing capabilities, we will follow a different approach. Set the option:

	***<D>***	Parallel Line

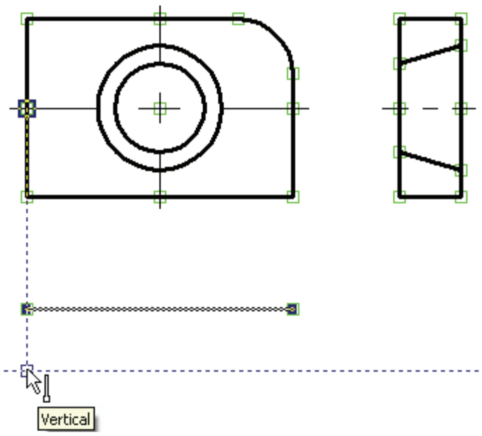
This is an encapsulated option in the segment creation group. If this icon is not displayed in the automenu, find it under one of the group icons marked with a black triangle (see explanation above).

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 .

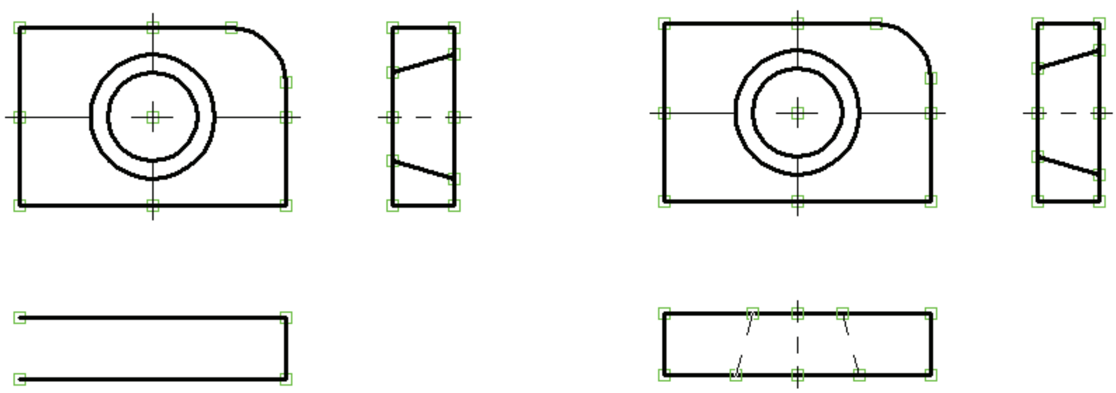




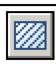
Thus, we have created the top segment of the plan view. A new auxiliary line starts rubberbanding after the cursor, parallel to the newly created graphic segment, as indicated by highlighting. Move the cursor down, and set the desired distance, equal to 35, in the property window, thus defining the thickness 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.



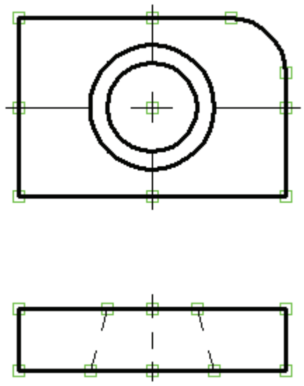
What is left now is to apply hatch on the side view. Call the command **"H: Create Hatch":**

	***<H>***	Create Hatch

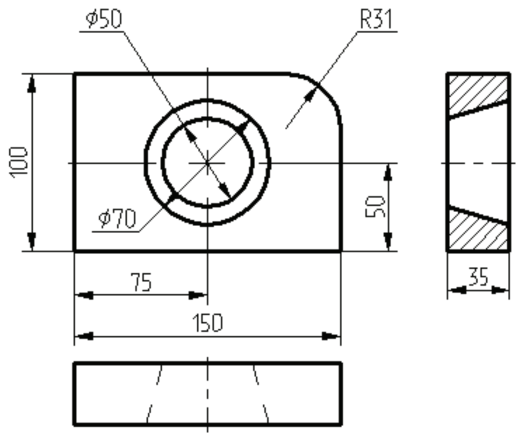
Set the option:

	***<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.

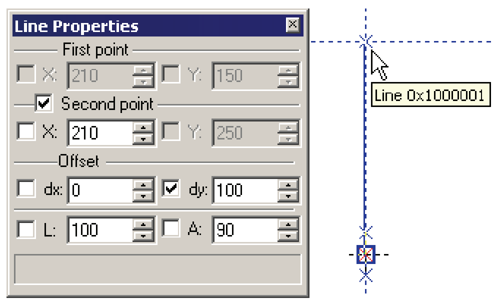
Call the command **"SK: Create Sketch"**. Make sure that the following snaps are enabled:

	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

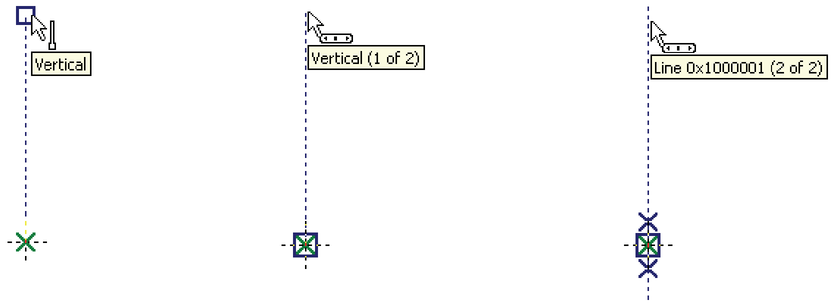
We will start with creating the main view of the plate. If necessary, enable the line segment creation option in the command's automenu (if desired). Create the first point of the segment, corresponding to the lower-right corner of the plate's main view. Please note that it was not a free node that was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

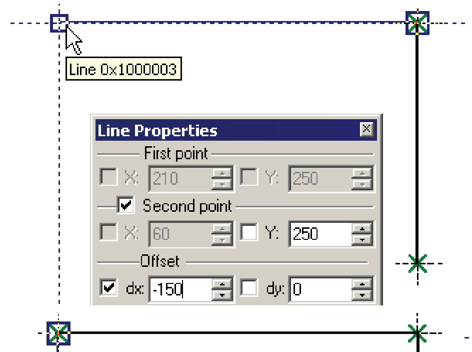


For the second node of the segment, define the Y-axis offset (100) in the properties window. The cursor will start moving along a horizontal auxiliary line. Move it to the vertical construction line going through the first segment node. When the snap to the latter line engages ("Line …"), click .

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.

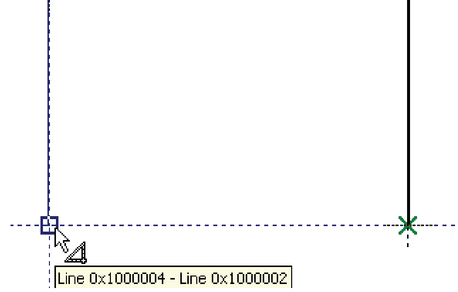
Please note, that, when selecting a snap, the system may offer the vertical snap to the first segment node rather than snapping to the line (the order of displaying snaps is determined by the settings in the command **"SO: Set System Options"**). To select the desired snap, do the following: briefly 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.



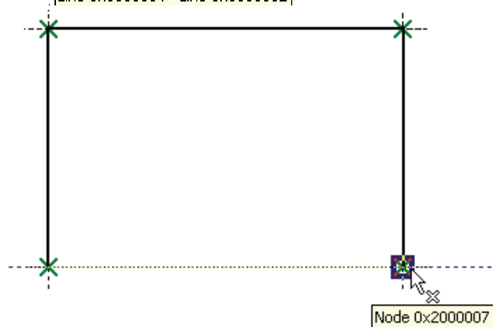


Create the second, horizontal, line segment with the length 150. When constructing it, specify the desired offset along the X-axis and use a snap to the line again.

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.

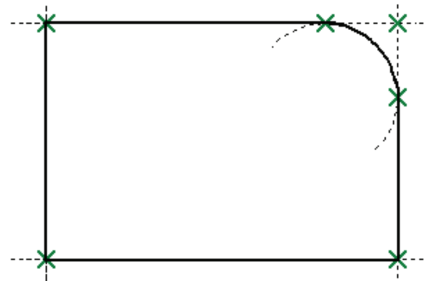


The fourth segment must be closing the described rectangle. After that, move the cursor rightward up to the first created node, as indicated by a special mark in the dynamic tooltip, and click . This completes the base for the part's main view.

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).

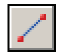
To create the fillet, let's use the option . After activating the option, set the fillet radius equal to 31 in the properties window. After that, select two segments, at whose intersection the fillet



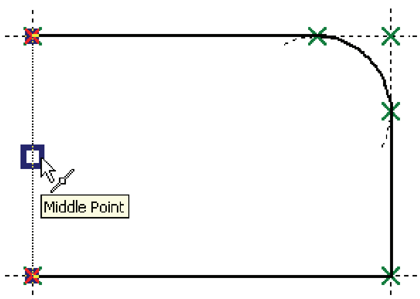
needs to be constructed (the top and the right-hand-side segments of the plate) or the node (the rectangle vertex) at their intersection.

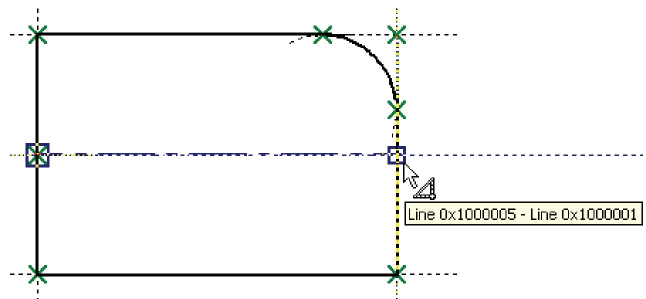
This will result in constructing a graphic line – a circular arc with a "slipped underneath" construction line-circle. Just like when creating a common sketch, the extra pieces of the fillet segments will be automatically trimmed.

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

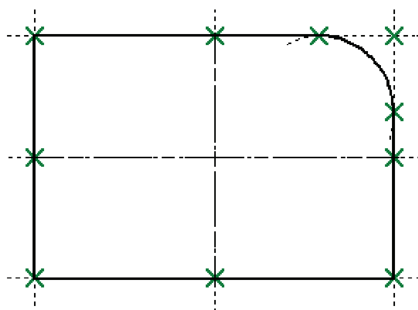
We will start with creating the axes. To create the axes, enable the  option again. Set the “Axis” line type in the system panel or in the graphic line parameters (the option ).

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.

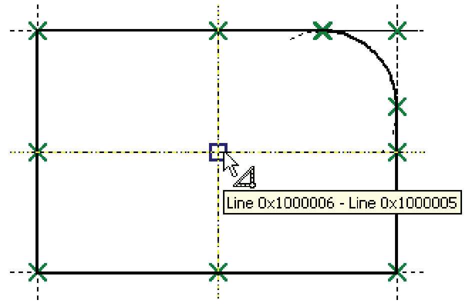




Similarly construct the vertical axis.

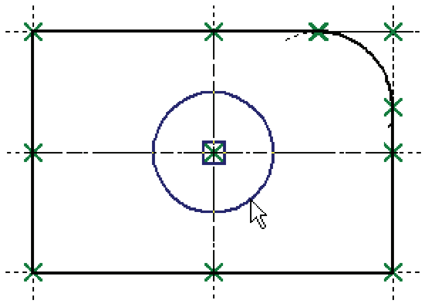


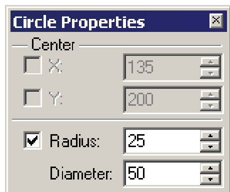
Now, let's create the circles. Set the normal graphic line type. After that, select the option . Move the cursor to the intersection of the axial lines, wait until the tooltip appears indicating the available snapping to the intersection lines. Click right there. A rubberbanding circle will appear on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

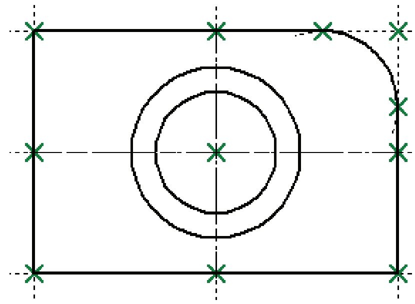


Set the radius value equal to 25 for the smaller circle of the conical hole and click or press the button **`[Enter]`**. Similarly construct the second circle with the radius equal to 35.

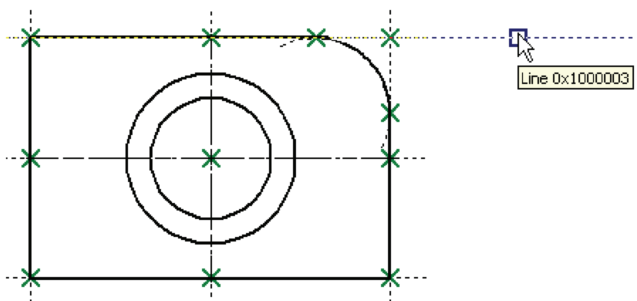
Please note that the construction result is not just free graphic lines representing the circles. The system constructed them as lying on the construction lines-circles.





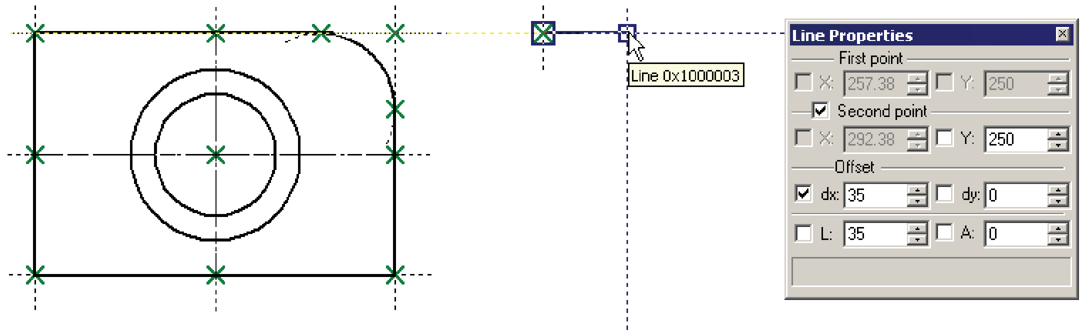


Now let's create the left view. To do that, enable the segment creation mode again (the option). If the system offers creating a segment from the last created node, refuse that by right-clicking . Move the cursor to the right-hand side of the drawing and set it so as to maintain the snap to the top line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

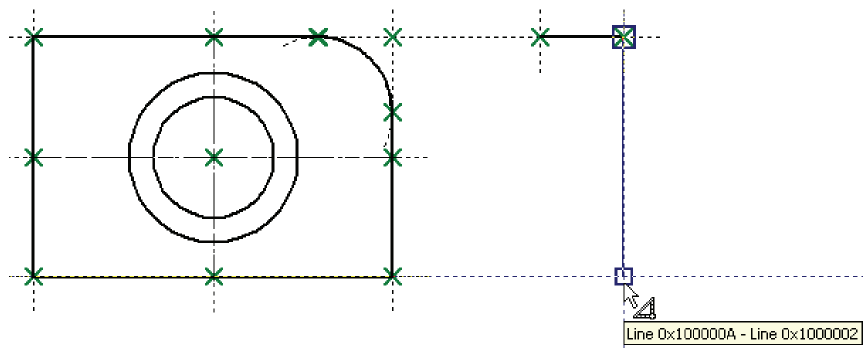


Click there . The first node of the new segment will be constructed as lying on the selected construction line.

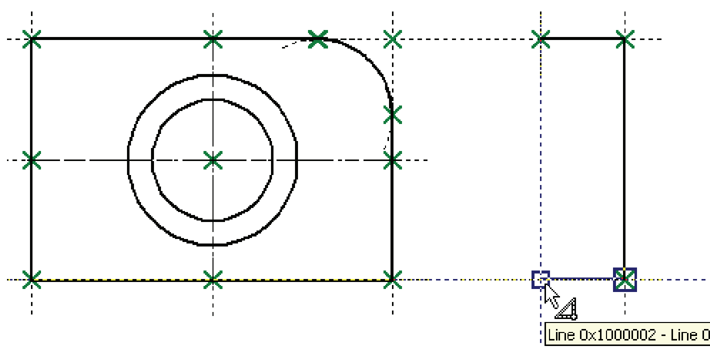
Move the cursor horizontally rightward. In the properties window set the offset of the second point along the X-axis equal to 35. Then move the cursor so as to pick the snap to the top line of the main view. Click . As a result, the second segment node will also lie on the top line of the main view at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

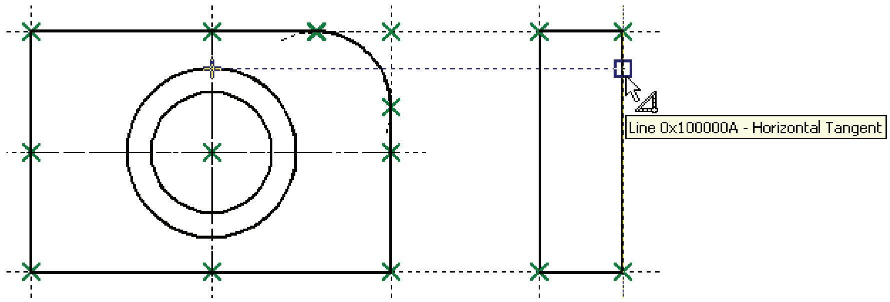


Click and move the cursor leftwards till snapping to two other lines.

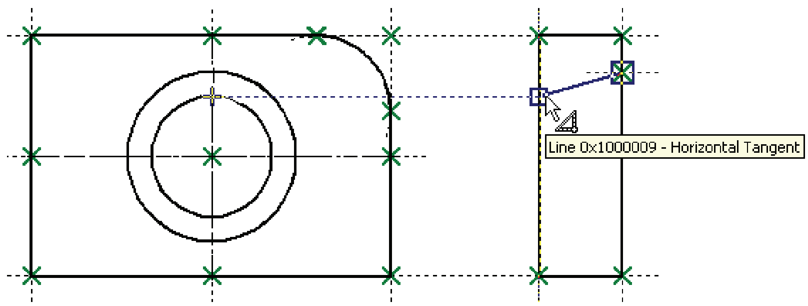


Click . Now, close the created graphic lines by moving the cursor to the first created node on this view, and click , then (to cancel the mode of continuous line input).

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.

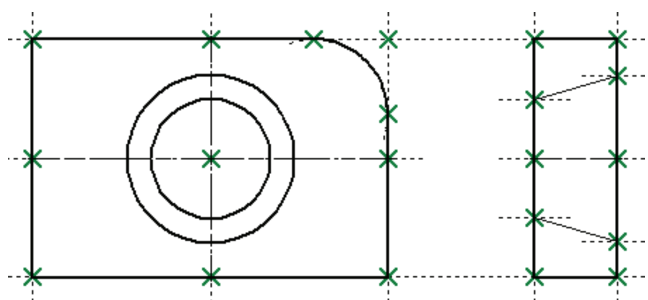


Move the cursor to the left-hand-side segment of the same view so as to establish the relation between the smaller circle and the line underlying that segment. Click .



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.

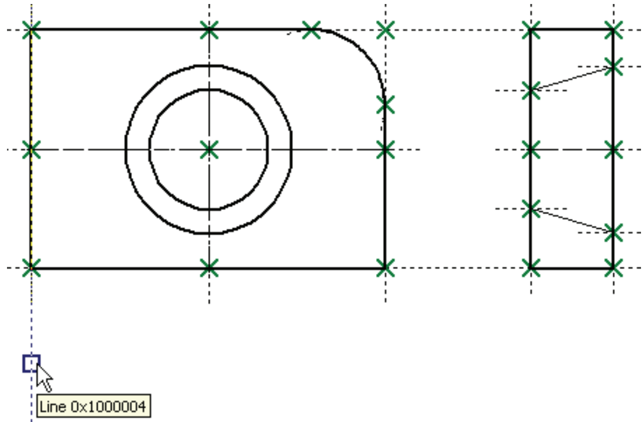
Next, use the same method to construct the lower line of the conical hole. Then create the centerline by snapping to the middles of the lateral sides of the left view. Do not forget to also set the dash-dot line type in the graphic line parameters (the option) or on the system panel.



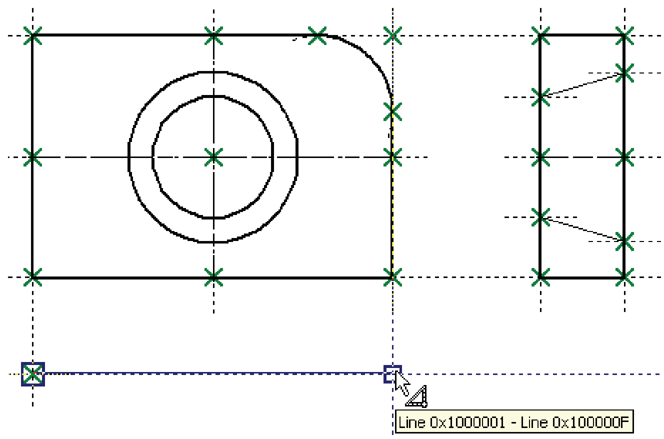
Let's proceed to creating the top (plan) view. We will create it a little bit different then when creating a simple sketch. We will not use the option of constructing a parallel segment. Now there is no practical necessity in defining that particular relation. When using the automatic parameterization 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.

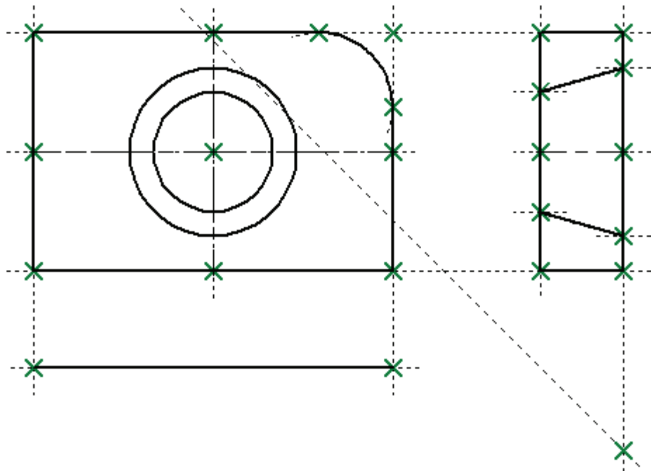


Next, move the cursor rightward up until hitting the snap to another line of the main view. Click again. We have just created the upper segment of the top view.

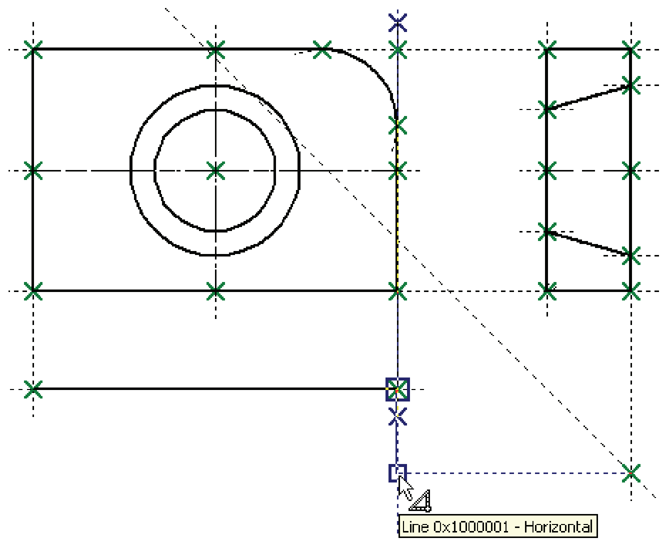


Next, we will have to temporarily quit the sketching command. The reason for that is that it is impossible to create a relation between the left view and the top view by the common means of automatic parameterization. Such relation can be achieved only by various workaround methods (for example, introducing 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).

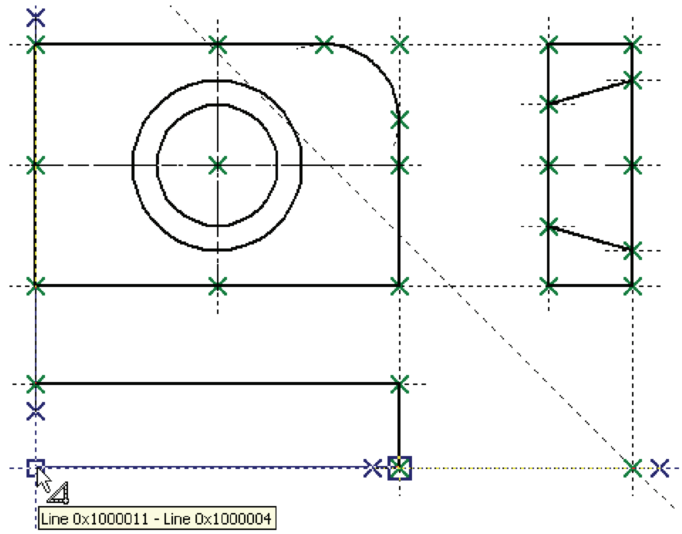
So go ahead and call the command **"L: Construct Line"**. With the help of the option , construct a line – the symmetry axis between the left-hand-side vertical line of the left view and the horizontal line of the top view. Place the cursor at the intersection point of the created line and the right-hand-side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



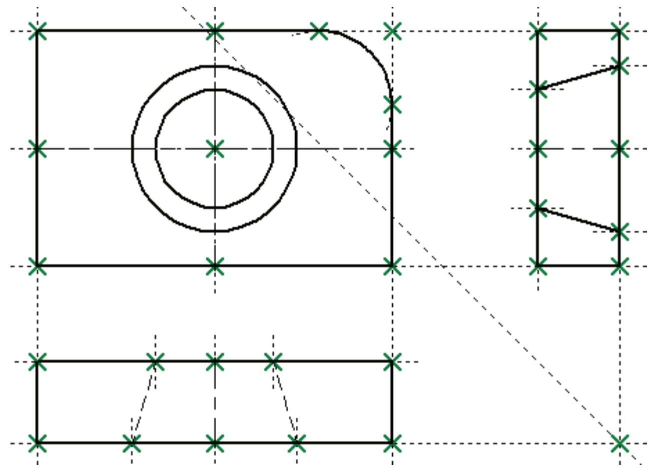
After that, call the command **"SK: Create Sketch"** again. We will create the next segment of the top view. Select the end node of the last created segment as the first node of the next segment. Then move the cursor up until reaching the intersection between the line and the horizontal through the node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



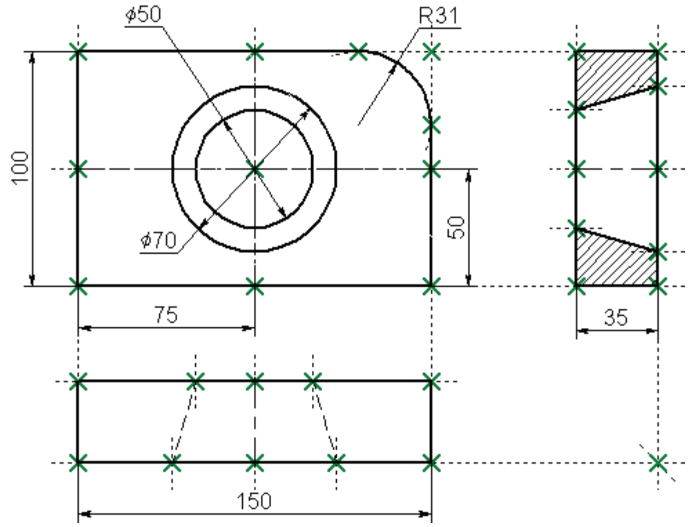
The next segment is constructed by snapping to two lines.



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).

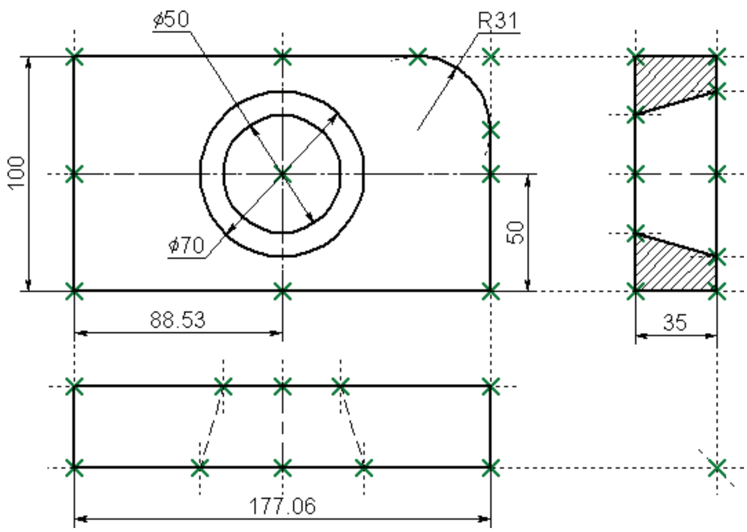


Create the hatch on the left view and the dimensions in the same way as in the previous cases.



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.

BASIC 3D TERMS AND CONCEPTS OF MODELING WITH T-FLEX CAD

Introduction to 3D Modeling

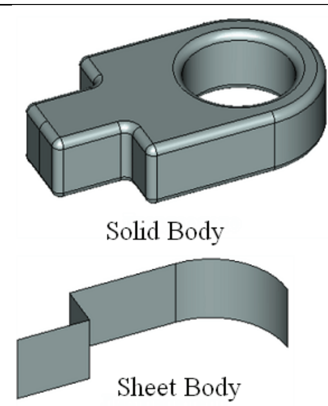
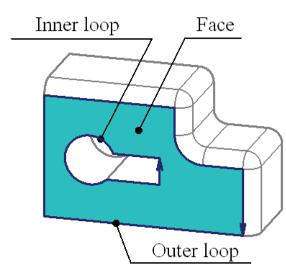
T-FLEX CAD 3D is a parametric solid and surface modeling system. It is equipped with most up-to-date tools for creating models of various complexities. Supported exporting and importing geometric data in common formats facilitates interoperability with most other CAD systems. T-FLEX CAD 3D also includes a complete line of instruments for 2D modeling and drawing compliant with various national and international drawing standards.

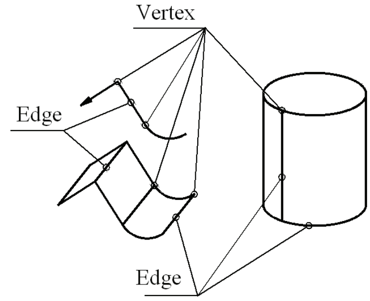
Before beginning with 3D modeling, it is recommended that users familiarize themselves with 2D drawing techniques in the appropriate sections of the documentation. This will help embracing the general modeling principles with T-FLEX CAD.

This chapter presents a classification and a brief description of all elements in T-FLEX CAD 3D, and basic methods and techniques of 3D modeling. The following chapters of this volume will describe each of these elements in detail.

Basic Topology Elements

A model in T-FLEX CAD 3D is a set of connected or disconnected geometrical components. Shown below are the basic topology elements that form any geometrical object in T-FLEX CAD.

<table>
<tr><td>Solid Body (a solid) is a set of geometrical objects – vertices, edges and faces that comprise a closed connected volume. A simplest solid can be defined by sweeping a bounded surface or a contour.</td><td rowspan="2">
</td></tr>
<tr><td>Sheet Body (a surface) is a set of geometrical objects – vertices, edges and faces that comprise a closed connected area and do not constitute a volume.</td></tr>
<tr><td>Face is a bounded patch of a surface. The boundary is represented by loops. One face may have an unlimited number of loops. A face may be without loops. This is the case of a full face comprising a closed volume, such as a full sphere.</td><td rowspan="2">
</td></tr>
<tr><td>Loop is a set of edges making one closed contour. A loop is an element defining the boundary of a face. Each vertex of a loop connects no more than two edges.</td></tr>
</table>

<table>
<tr><td>Edge is a curve segment bounded by two vertices. A closed edge may contain only one vertex.</td><td rowspan="2">
</td></tr>
<tr><td>Vertex is a point in space serving as a bound for edges. One vertex may belong to several edges.</td></tr>
</table>

Basic Geometrical Terms in T-FLEX CAD 3D

All geometrical objects in T-FLEX CAD can be classified into four general groups by type of the object geometry:

- The simplest object in the three-dimensional space is **3D point**. A point has only one property, namely, the placement coordinates. It can be defined by a 3D node, a 3D vertex, a “placement” parameter on a curve or surface, a characteristic location property within an object, such as on an axis of revolution, at the center of an arc or sphere, at objects intersection, etc.
- The second group of objects is “wire” geometry, the objects that possess the basic property of length. These include all linear objects, as are edges, 3D paths, loops and 3D profiles.
- The next group includes all objects that are characterized by area. These objects will be referred to as **“sheet”** geometry, and include all kinds of surfaces, sheet bodies, faces, and closed 3D profiles. A sheet object can be produced by most 3D operations.
- The forth group includes all **solid bodies**.

3D Entities and Operations

A 3D model creation implies building solid or sheet 3D objects describing a certain volume or surface in the 3D space. The creation and further modification of such objects is done by means of ***operations***.

Operation is any step of creating a 3D model that leads to emergence of a new or modifications in an already existing solid or sheet geometrical object. A separate command is provided in T-FLEX CAD for each operation. The names of operation creating commands correspond to an operation’s purpose.

The operations that result in new solid/sheet 3D objects will be referred to as ***basic operations*** (extrusion, rotation, sweep, loft, etc.). The operations that alter geometry and modify already existing solid/sheet objects will be referred to as ***modifier operations*** (blend, shell, Boolean, etc.).

The geometry base for most basic operations is provided by ***3D construction elements***. 3D construction elements are auxiliary elements of a 3D model that are used for creating three-dimensional contours, defining spatial orientation, determining directions, vectors, axes, trajectories, etc. A separate command is provided for creating each such element.

> Some operations (for example, the operations for creating 3D arrays) can be both basic and modifier operations, depending on the source data and parameter settings.

Any solid or sheet 3D object in the 3D scene corresponds to a special element in the 3D model structure, which is ***Body***. The “Body” element is introduced for user convenience: once a new geometrical object is created by the first (basic) operation (solid or sheet body), it can be handled in the future as a permanent element in the 3D model structure.

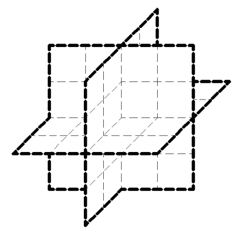
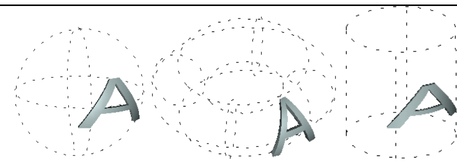
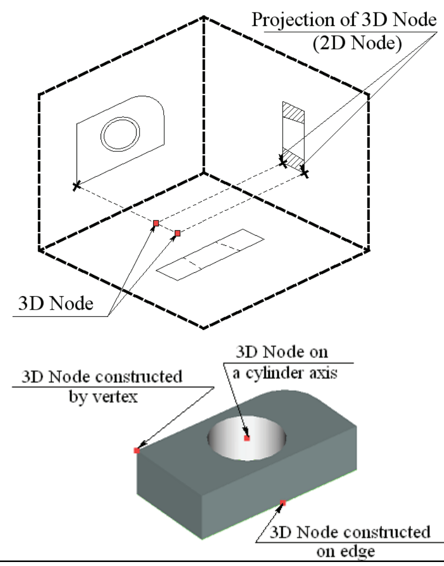
Body is created automatically upon creating a solid or sheet 3D object by a basic operation, and is maintained as long as the given object exists. The geometry of the original volume or surface may change (as a result of applying modifier operations), but it always refers to the same Body that defines parameters of this geometrical object: name, material, color, rendering settings (mesh density, wireframe display).

A 3D model can contain an unlimited number of Bodies.

In some commands, Bodies can be used as separate elements. In such a case, the source object will be the body of the last operation in the creation history of the given Body. For example, when creating a 2D projection, a specific Body can be selected for projecting. This would be convenient, if the first design stage was making the drawing of the part's workpiece or a set thereof, while the second - modifications of this model by additional operations. In this way, the drawing reflects on all future modifications to this part.

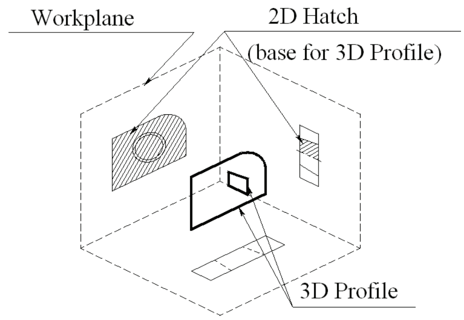
> In this manual, the term "Body" (with capital "B") means specifically a 3D model structure element. The lowercase inscription "body" will be used for a quick reference to the geometrical object, which is a volume or surface in the 3D scene.

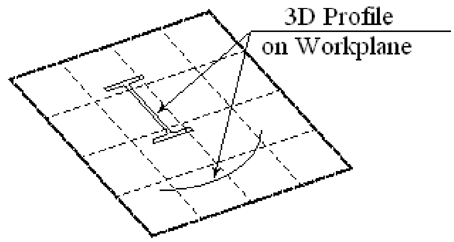
3D Construction Entities

Workplane is an entity that helps defining the required data for 3D operations, and, first of all, creating 3D profiles. A 3D model cannot be built without creating a workplane. Workplanes can be specified in 2D or 3D window based on various references, such as the 2D drawing views, projections of entities in a 3D model, or the 3D entities themselves, including other workplanes.	
Work Surface serves similar functional purposes as the workplanes, providing a non-planar geometrical basis for further design. A work surface can be a cylinder, a sphere, or a torus.	
3D node is one of the basic construction entities used for representing a point in the three-dimensional space. There are several ways of creating 3D nodes. A 3D node can be specified, for instance, as a characteristic point on a body, referencing a vertex, an edge, or a face. It also can be located by using absolute coordinates or offsets from other 3D nodes. 3D nodes can also be specified using the nodes from the two-dimensional drawing, and workplanes. In this way, the 3D node will be defined by selecting one node on a workplane, or two nodes on two different workplanes that are related by having a common projection.	

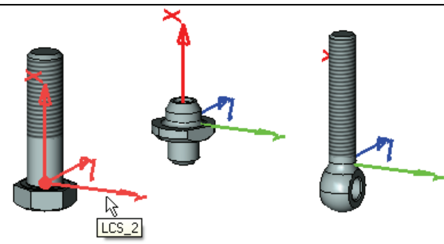
3D profile is a construction entity used for defining a patch on a surface. A 3D profile is one of the basic entities because it is used as an original reference for many operations. A 3D profile may be defined by an open or closed contour. A closed contour defines a fixed area on the surface that can be used as a base for various operations creating solid bodies. An open profile can be used as a base for creating sheet bodies only as it consists of wire geometry.

There are many ways of creating a 3D profile. Thus, it can be drawn on a workplane, or get anew by a variety of modifications to an existing profile, and so on. One 3D profile may contain several contours of the same type. An example of a multi-contour profile is a text entity. A 3D profile can be obtained from hatches and graphic lines (when drawn on the active workplane).

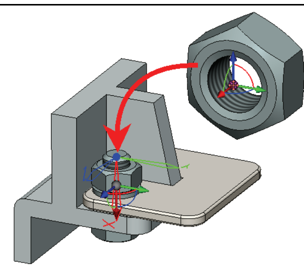




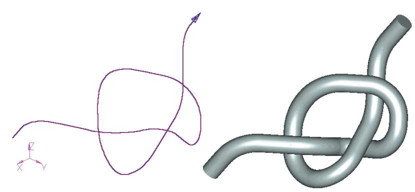
Local Coordinate System is an entity for referencing three-dimensional objects in space. It is used for inserting 3D fragments and other elements, copying, exploding of assemblies, etc. This entity is defined by the origin point and the axis directions. Objects snap to a coordinate system by making the object and the target coordinate systems coincide.



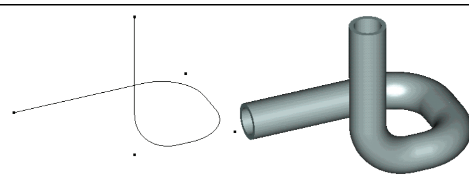
3D connector – is a special type of local coordinate systems which allows a user, apart from performing the functions of snapping, to automatically tie external variables for jointed parametric elements of the 3D assembly models. This significantly simplifies positioning of parts and parameters assignment when designing the assemblies.

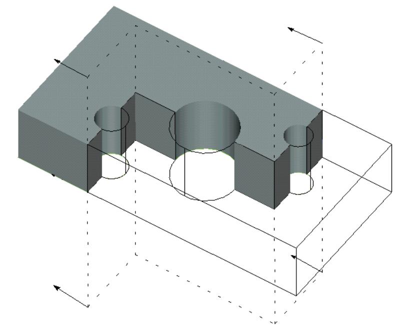
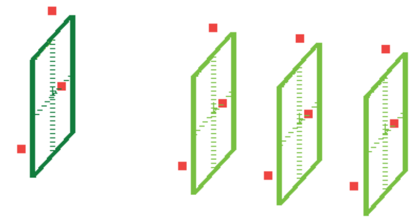


3D Path is a bounded three-dimensional curve with a defined traversal direction. The 3D paths are used in the operations “Sweep”, “Pipe”, “Loft”. A 3D path can be defined by a hatch, 2D paths, as a chain of a body edges, as a curve passing through a sequence of 3D nodes, by modifying existing 3D paths, etc. A 3D path can be closed.

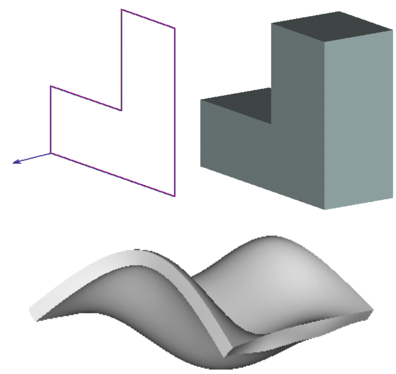
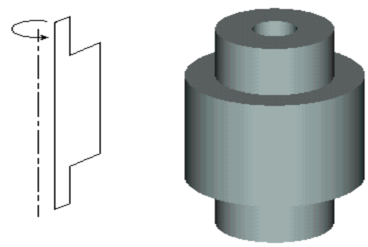
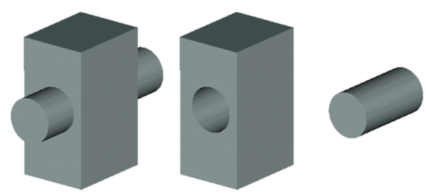


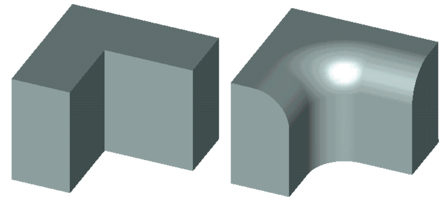
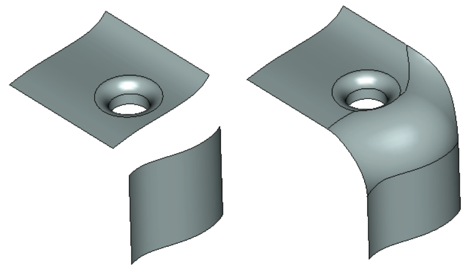
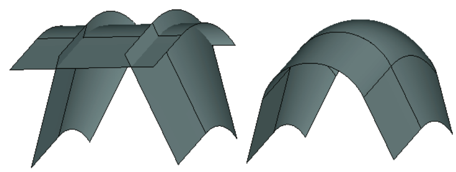
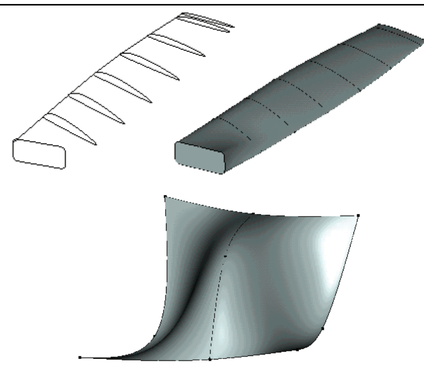
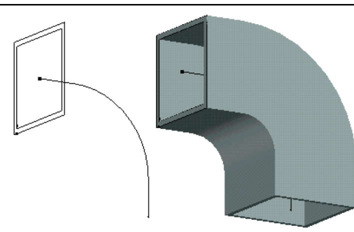
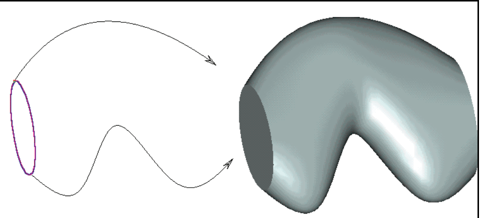
Pipe Path is a 3D path consisting of straight line segments smoothly connected by arcs. This command is mainly used for modeling pipes. A large variety of options and controls provided by this command help quickly and easily solving this sort of tasks.

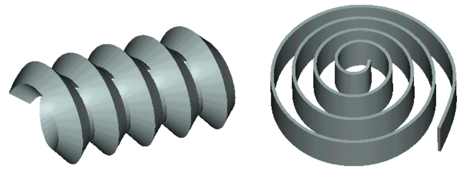
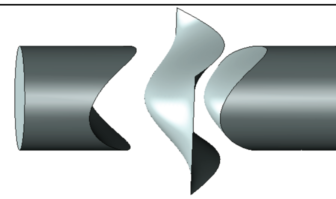
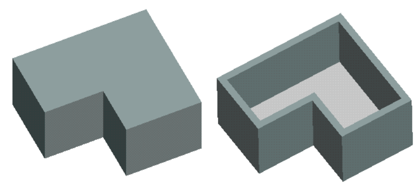
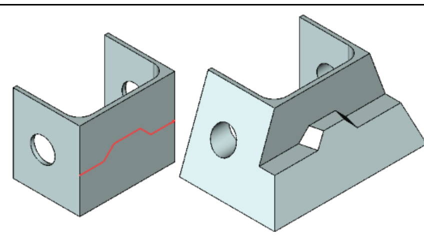
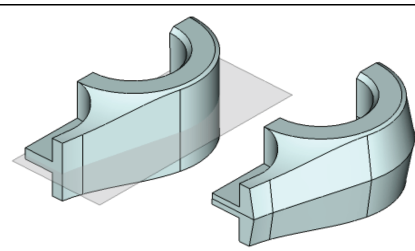
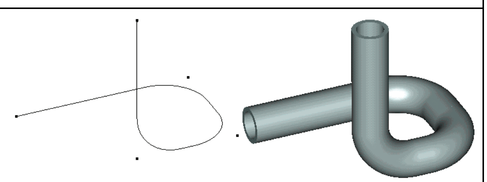


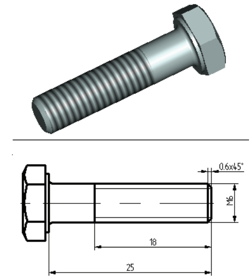
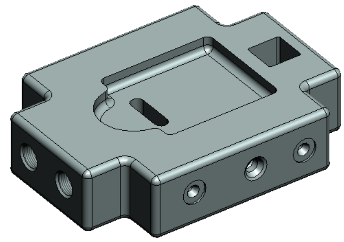
3D Section is an entity that, generally, results from extruding a planar curve or polyline through all geometry in the direction normal to the plane of the curve. In some cases, a 3D section consists of one or several planes. In this case, it can be used for creating two-dimensional cuts. A 3D section may be included in visualization of the objects of a 3D scene, and can be used in “cut” operations.	
Construction array is a special composite construction element. A construction array is a particularly organized set of copies of an arbitrary construction element, except for sections, light sources and cameras. The copies contributing to such an array can be used as conventional construction elements. Construction arrays are created and edited using any of the 3D array creation operations.	 Source construction elements Construction array (a set of copies of original elements)

Basic Three-Dimensional Operations

Extrusion yields a body formed by straight propagation of a contour along a specified direction. This operation creates both solid and sheet bodies. Extrusion can be performed along not only the extrusion vector, but the normal to the contour in either or both directions as well. Thus, it provides a means of thickening an arbitrary face, even a non-planar one.	
Rotation yields a body formed by revolving a contour around an axis located in space for a given angle. The original contour can be located in an arbitrary orientation with respect to the axis, but it should not intersect with the axis. This operation creates both solid and sheet bodies.	
Boolean operation is intended for creating a new body by combining two existing bodies. The type of the operation is specified as addition, subtraction, or intersection.	 Addition Subtraction Intersection

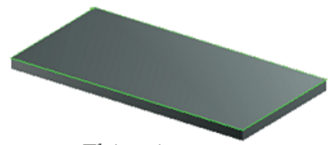
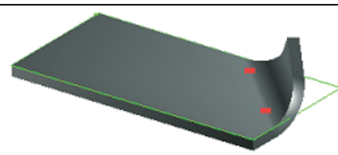
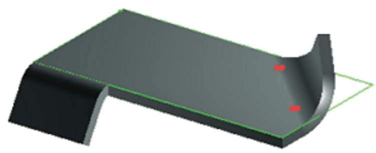
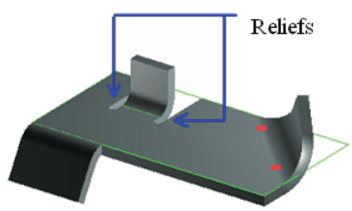
Blend Edge is an operation that modifies an existing body by smoothing or merging its vertices, edges and faces. The main difference of this operation from other types of blending is in that the new surface is created from the selected edge and merges with the adjacent faces only. The operation permits creating chamfers, edge rounding with a variable radius and elliptic rounding.	
Blend Face-Face is an operation for creating smooth transitions from one set of smoothly connected faces to another. The sets of faces to be blended may not have common edges (intersect). The command has numerous options for controlling the blending surface shape, trimming conditions, bounds, etc. This operation should not be considered a substitute for blending edges. Each of the two approaches has their own advantages and well complement each other.	
Blend Three Faces is a special case of face–face blending separated into another command. The operation creates a transitional surface between a “right” and a “left” wall while tangent to a “middle” wall.	
Loft is an operation for creating new bodies of complex geometrical shapes. The resulting spline surfaces are constructed on the wire guides in one or two directions and according to the specified boundary conditions. The base elements for spline definition can be practically any entities of the three geometrical types, “point”, “wire” and “sheet”. Depending on the base element type, the result will be either a solid body or a set of surfaces.	
Sweep is an operation that creates bodies whose surface is defined by an arbitrary-shape profile propagated along a space curve. The operation provides controls over scaling of the profile and its twisting with respect to the trajectory axis as it moves along.	
Parametric Sweep – this operation extends the capabilities of the “Sweep” operation. The base profile is defined in such a way that its geometry and orientation are driven by a variable. The driving variable can assume values within the specified range. The body is created as a result of recomputing the geometric shape and orientation of the profile across the whole range of the variable values.	

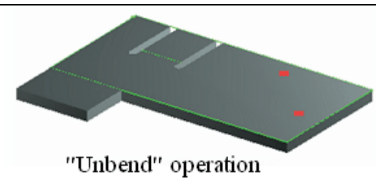
Spiral is an operation for creating spiral-like bodies. The generatrix contour can be defined by an arbitrary-shape profile. This operation may be used for modeling actual geometry of a thread. However, in most cases, when only cosmetic representation of a thread is needed, the “Thread” operation is recommended for use instead.	
Spring is a specialized operation for creating spring-shape bodies. It differs from the “Spiral” operation in the capability of forming the end cycles of the spring. The generatrix of a spring is defined as a circle.	
Cut is an operation that divides a body in two, or cuts a portion off a body. The cutting surface can be defined as a set of connected faces, or sections, or workplanes.	
Shell is an operation that removes the inside material of a part by removing the selected faces and leaving the rest of the faces just thickened by the specified width in the direction where the material used to be, removing all the rest of the material. A shell can also be created without removing any faces. Another special functionality allows creating new bodies offset from a given one.	
Face taper is an operation that tilts selected faces at the specified angle while automatically adjusting other affected faces.	
Body taper is an operation that creates 3D bodies by tapering faces of a selected body at a specified angle with respect to selected edges of that body in the taper direction. This operation significantly simplifies the design of cast molds. Unlike “Face Taper” command, this operation allows creating two-sided tapers and tapered body faces without having a clearly defined “fixed” edge.	
Pipe operation makes a pipe along a space trajectory (a 3D path). The user specifies the diameter of the pipe and the diameter of the outlet. The outlet itself is optional.	

Thread is an operation for creating cosmetic representation (imitation) of threads on cylindrical and conical faces of a 3D model. When the projections of the model are created on a drawing, the thread is automatically drawn according to drawing conventions	
Hole is an operation for creating standard apertures. It relies on a provided parametric library of holes satisfying current standards. The command supports creation of patterns of holes, holes through multiple bodies, and threaded holes. When a threaded hole is created on a face, a cosmetic thread is displayed.	

Sheet Metal Operations

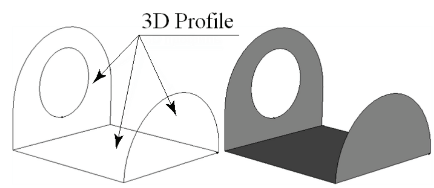
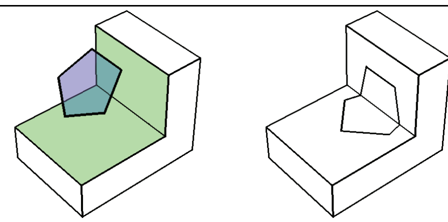
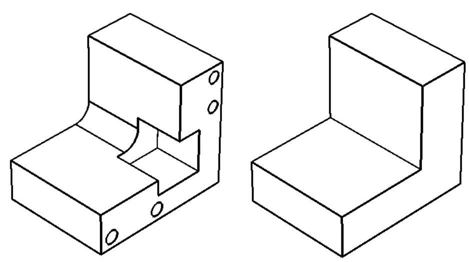
The specialized sheet metal operations make a separate group.

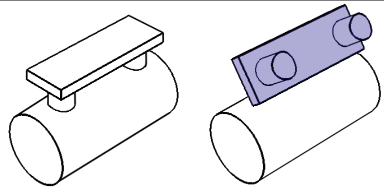
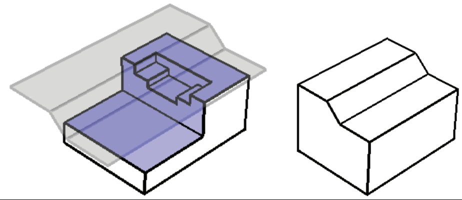
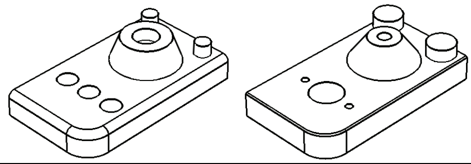
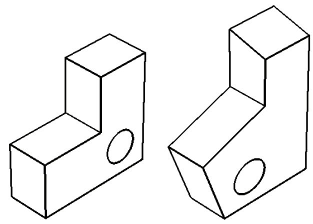
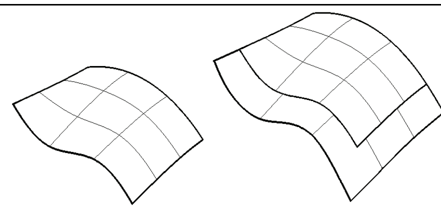
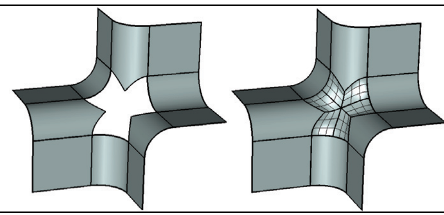
Create Base Part operation creates a solid body – a flat part. The base element for the operation is a flat closed 3D profile. The body is created by thickening the profile by a specified amount. The part material will be added along the normal direction to the profile at either or both sides of the profile.	 Flat part
Bend operation supports three types of bending, "Bend", "Attach Flange", "Cut and Bend". • The first type is for bending an existing body, such as a part, around an infinite axis defined by a pair of 3D nodes or a straight-line entity. • The second type supports attaching a flange to the part according to specified length, width, bending radius and the offsets from the ends of the bending axis. • The third type is for bending a portion of the part around a finite line segment. This is done by making appropriate cuts in the original body. The operation allows introducing reliefs defined by their type, depth and width.	 "Bend" operation  "Attach Flange" operation  "Cut and Bend" with customized reliefs

Unbend operation can be used after a sheet metal model has been defined, to get the flat part and proceed, for instance, with its drawing. **Re-bend** operation repeats bending of all surfaces that were previously “unbent”. It provides a means of keeping the final bent shape with the model data. This is important, for instance, when creating a drawing with both the flat part views and a final bent shape view.	
Forming Feature operation is specifically provided for performing common sheet metal stamping manipulations. It uses a library of typical elements for creating ribs, beads, flanges, embossings, louvers, etc. The geometry of these features is defined by 3D profiles.	

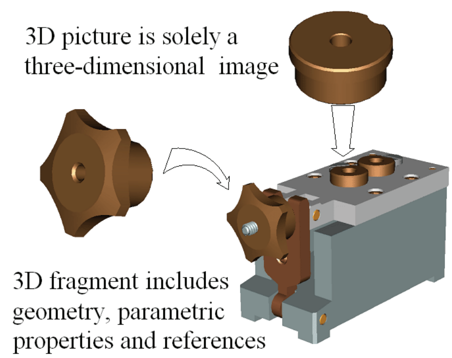
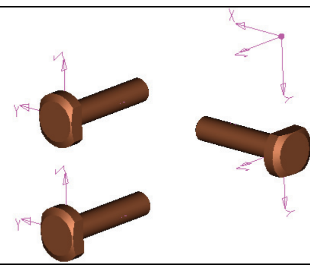
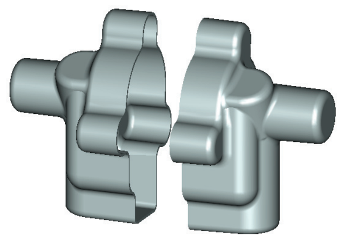
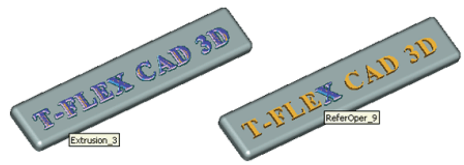
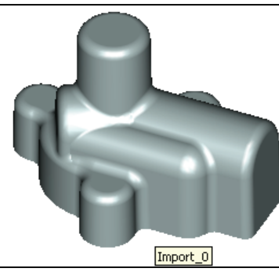
Face Handling Operations

The set of commands for direct handling of body faces are united into a separate group.

Sew is an operation for creating solid bodies or sheet models from a set of disjoint surfaces adjacent to each other, as faces, 3D profiles, sheet bodies. When sewing into a solid body, the operation may add simple surfaces as necessary.	
Imprint Elements is an operation that forms patches of a specified shape on already existing faces. Depending on the dividing option, the shape of the new patch is determined either by the shape of the dividing element or by the geometry of the element being divided.	
Delete Faces is an operation that allows deleting one or several selected faces. Deleting faces breaks the solid body topology. Gaps are introduced that invalidate the closed volume. If necessary, the system may attend to mend those by various means.	

Separate Faces is the command for excluding selected faces from an already existing body and using those for creating a new body. The gaps in bodies caused by this process can be closed by one of several means.	
Replace Faces is the command for substituting a geometrical surface underlying the selected faces by different surfaces. Sheet bodies can be used as a replacement surface.	
Change Faces is the command for modifying face parameters when the underlying surface is analytical (a cylinder, cone, sphere or torus), as well as parameters of faces created by the blend operation.	
Transform Faces is the command that applies a transformation to one or multiple selected faces.	
Expand Faces is the command for expanding the selected face (or multiple faces belonging to a sheet body) in the specified direction by the given length. The direction of growing the face is defined by selecting side edges on the faces being expanded. 3D profiles can also be expanded.	
Fill Hole is the command for creating one or multiple faces closing an area bounded by a closed loop of edges. Depending on the initial geometry, the system may fill the area by an analytical or a ruled surface, or by an explicitly specified sheet body.	

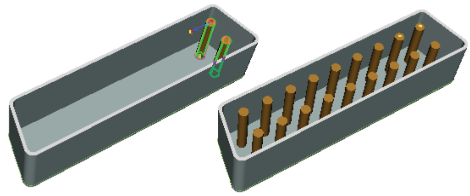
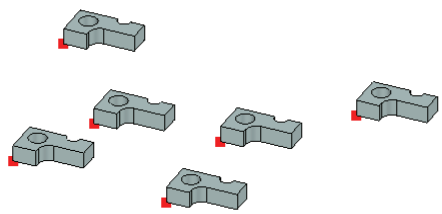
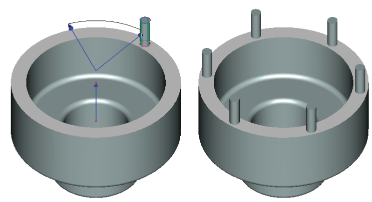
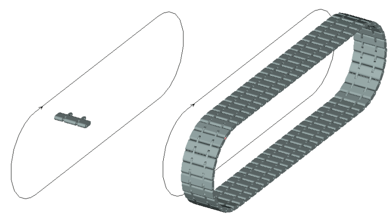
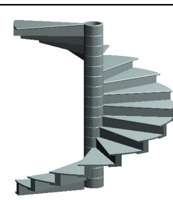
Copy and Insert Operations for 3D Elements

Insert 3D Fragment is an operation for using the geometrical data of a standalone 3D model in creating an assembly model. Any T-FLEX CAD document containing a 3D model can be used as a 3D fragment.	
Insert 3D Picture is similar to inserting a 3D fragment, except that the 3D picture has neither associativity among elements nor parametric modification capabilities. It is merely a shaded 3D image of a part that appears exactly as a 3D fragment. 3D pictures are handy at a final stage of design when the element modifications are no longer expected. Since 3D pictures are not subject to regeneration, the overall model regeneration time is reduced. 3D pictures cannot be used as references for other elements. Thus, faces of such objects can't be selected, and projections cannot be constructed.	
Copy is an operation for creating a transformed copy of a body defined by various transformation parameters. Copy operation utilizes a "local coordinate system" entity.	
Symmetry is an operation for creating new bodies as copies of the original of bodies by reflection at a specified symmetry plane.	
Divide is an operation for separating numerous bodies obtained from various operations. The resulting bodies can further be processed separately. For instance, the bodies created by an "Array" operation, will be separated into standalone elements. This command can also be used with an imported from other systems model that consists from several bodies.	 An extrusion of a multi-contour profile based on a text (Extrusion_3) is divided into separate bodies (letters)
External Model is an operation for importing models created in other systems based on Parasolid format (*.x_t and *.xmt_txt). Just like 3D pictures, these objects do not possess parametric properties, however, their elements (vertices, edges, and faces) can be used as references for further design.	

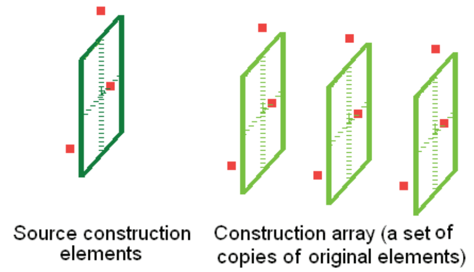
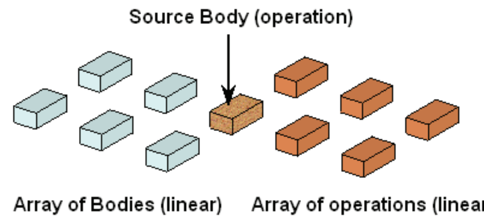
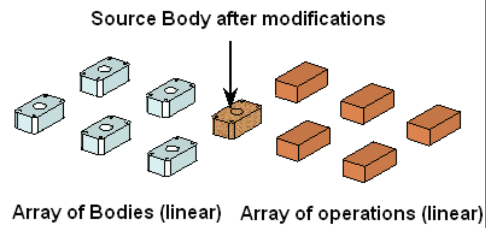
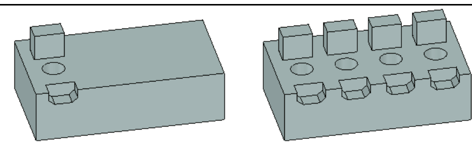
Operations for Creating 3D Arrays

The array creating operations allow simultaneously creating multiple copies of source 3D objects. The source objects for creating arrays (objects for copying) can be not only operations and Bodies, but also 3D construction elements and faces.

Placement of the copies being created (the array elements) depends on the array type: linear, circular, array by points, array by path, parametric array.

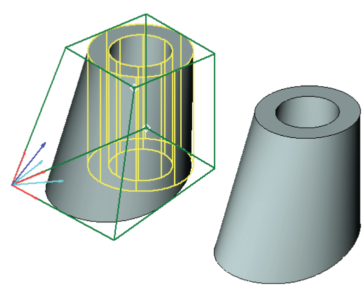
Linear array is the one in which the copies of the source objects are placed along one or two direction vectors at a specified step. The copies can be placed not only in the forward, but also in the reverse direction along each direction vector.	
Array by points is the one in which the positions of the arrayed copies are defined by 3D points.	
Circular array is the one in which the copies are placed on a circle around the array axis. Reverse rotation is also possible, including creation of copies simultaneously in two rotation directions. It is also possible to create copies in a second direction - either along the array axis or in the radial direction.	
Array by path is the one in which copies are placed along one or two spatial curves. One can set different ways of orienting copies on each guide curve – by the cord, by the minimal twist, by parallel translation.	
Parametric array is the one in which the spatial positioning and parameters of copies are defined by a specified parametric law.	

Depending on the type of the copied objects, distinguished are the following array types: arrays of construction elements, arrays of operations, arrays of Bodies and arrays of faces. All arrays of one element type, regardless of their properties, share the specifics of creating and editing.

Array of construction elements makes copies of any 3D construction objects, except for sections, light sources and cameras. It results in a special 3D construction element – a construction array.	
Array of operations copies only the result of the selected operation. If the body created by the operation is later transformed by another modifying operation, the array is not affected. **Array of Bodies** copies a whole Body. If that Body is modified in the future, the array will regenerate, accounting for the new operations added in the Body's history.	 
Array of faces is used for adding holes and protrusions repeating existing design elements, to Bodies already existing in the 3D model. Any array of faces is always based on one model Body: all copied faces must belong to this Body.	

Deformation Operations

The operations of deformation allow a user to carry out modification of solid and sheet bodies by various means. When applying these operations on the basis of parameters specified by a user, an internal function producing the volume deformation of the deformed body is generated in the model. Applying this function in a continuous manner deforms the entire volume of this body (or its part). Topology of the deformed part of the body is not changed. The number of faces, edges, and vertices, etc. is preserved. If necessary, the faces and edges of analytical type (planes, segments, cylinders, arcs of circles, etc.) are automatically replaced with spline surfaces and curves.

Skew – this type of deformation assumes existence of the original body and the coordinate system in which a bounding parallelepiped is calculated. Deformation law is specified by displacing the vertices of this parallelepiped in different directions. The displacement can be realized along any of the axes of the coordinate system, along the edges of the bounding parallelepiped, and along the diagonals of the faces of the bounding parallelepiped.	

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Sculpt deformation – in this type of deformation a regular mesh of points is defined on one of the faces of bounding parallelepiped. Any point on this mesh can be displaced relative to its initial position by a specified value. As a result, a flat face of virtual bounding parallelepiped is transformed into a space spline surface which forms the required transformation law for the body.

The operation of sculpt deformation has 3 modes:
• One side – only the points located on one face of the parallelepiped are displaced;
• Both sides – the points located on opposite faces of the parallelepiped are displaced. The points on the face opposite to the selected one are displaced in the same direction by the same distance;
• Symmetric – points on the opposite face are displaced symmetrically with respect to symmetry plane of the parallelepiped.
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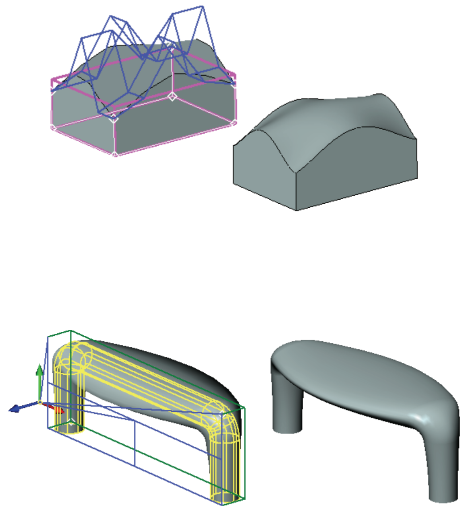
<td></td>
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Scale/Twist – this operation allows a user to specify various scales and angles of twist in different sections along the axis of the selected coordinate system. This deformation can be carried out either for the entire body or within the borders of the user-defined region.

In addition to scaling and twisting sections, it is possible to entirely stretch or compress the deformation region in the direction of the selected axis of the deformation. For sections, the scales along different axes can be different.
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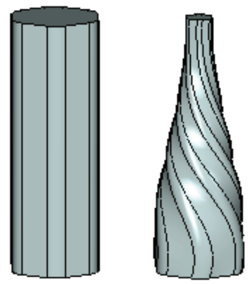
<td></td>
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<td>
Bend – this operation allows a user to bend the selected body by a specified angle with respect to the selected axis. In addition to bending angle, the distance defining the location of the «neutral» surface is specified.
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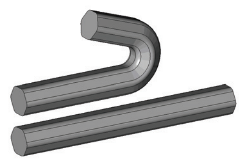
<td></td>
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Deformation by curve – for the deformation by curve, the overlap of the source curve, associated with the deformed body, with the target curve takes place. The deformation function constructed as a result of this is applied to the deformed body. A user can control the location of the body with respect to the curve. Upon deformation, one of the three body orientation control algorithms and the ways of using the source curves can be chosen.
•3D curve – 3D curve. This method uses one source and one target curve. In most of the cases in practice, the source curve is a line. As an example, consider one of the typical problems solved by the present algorithm – wrapping the source body into a ring.
• Curve – Spiral. The spiral-like curve is chosen to be a target one and the axis of the spiral is specified.
• Pair of curves – Pair of curves. The source pair of curves and the target pair of curves are specified. Additional curves perform the function of controlling the twist of the body with respect to the main curve.
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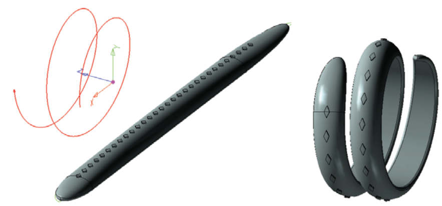
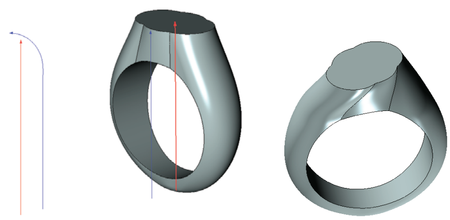
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<td>
Deformation by surface – this type of deformation forms the transformation law of certain single surface into another surface and applies this law to the source body. As the initial data from the source and target sides, the surfaces and the three points on each surface are chosen for the subsequent overlap. An additional displacement of the resulting surface from the source surface can be also specified.
This transformation works in two modes – «By parameters» and «With minimal distortion». In the former case, the exact overlap of the parametric surface spaces by the selected points is carried out. In the latter case, the proportions of the geometric distances measured between certain points on the source surface are preserved.
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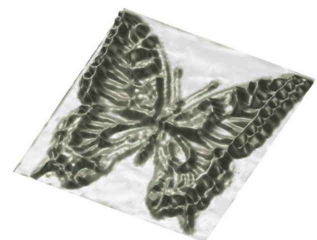

</td>
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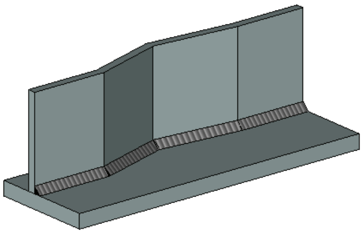
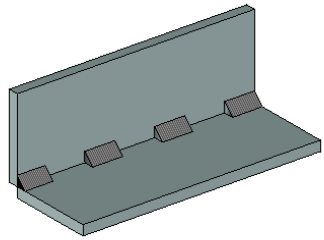
Commands to Create Welds

A group of commands in the **“Tools|Weld”** menu is provided to design welded parts. Those serve to create various types of standard and nonstandard welds on a 2D drawing or in the 3D model. Once a weld is created, its symbol can be automatically applied, and tables of welds can be built.

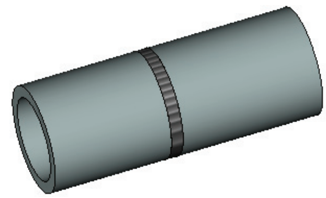
There are the following 3D weld types: fillet, intermittent fillet, butt, composite. Fillet, intermittent fillet, and butt 3D welds are denoted in the 3D scene with a special “cosmetic” body with a characteristic texture.



Fillet 3D weld



Intermittent fillet 3D weld



Butt 3D weld

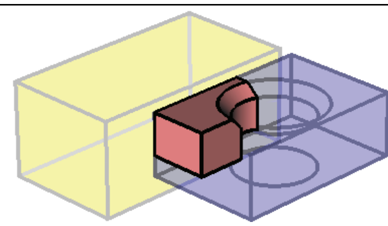
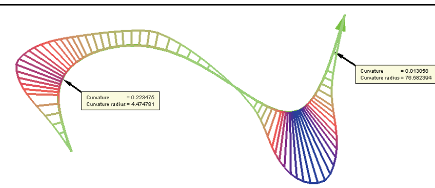
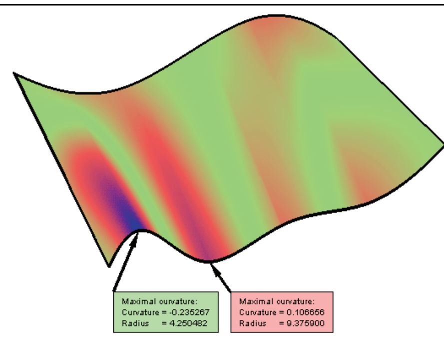
The composite weld is a variation of the 3D weld. It can be used to designate some 3D model elements (bodies, edges, 3D profiles, 3D paths, or a combination of several existing 3D welds) as a 3D weld. In this case, no image of the weld will be created in the 3D scene.

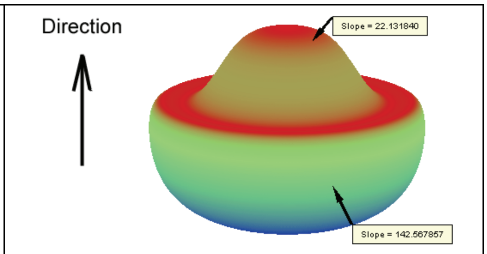
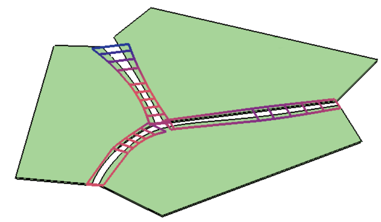
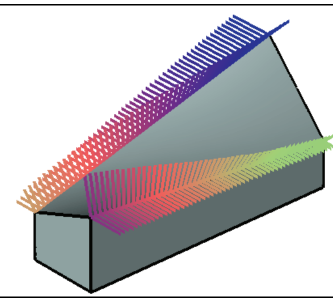
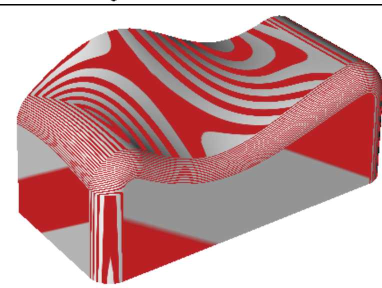
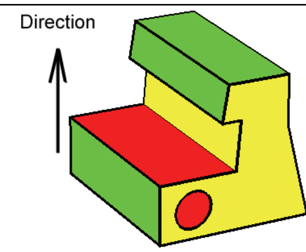
Geometry Analysis Commands

Measure is a command for defining mutual situation of objects on the 3D scene, whether one body penetrates another one, what is the minimum distance between elements. Besides that, various geometric characteristics can be computed for selected elements, such as length of an edge, area of a face, coordinates of nodes, etc. One can introduce variables that will be gaining specified characteristics from the elements of interest, using special functions. Thus, these characteristic values can be used as input for further construction.

Mass-Inertia Properties is the command for calculating the mass-inertia properties of the selected operations. If necessary, the calculations can be done with respect to a specific coordinate system.

Model check is the command for diagnosing the selected body for errors in its geometry.

Intersection Check is the command for examining the model against intersections and contacts between selected bodies. The command is particularly helpful when working with assemblies.	
Curve curvature is the command for measuring the curvature and the radius of curvature on selected curves. The curvature is displayed in the form of the porcupine quill. To measure curvature, you can select edges and 3D paths.	
Surface curvature is the command for measuring curvature and the radius of curvature of one or multiple selected faces. You can watch the overall curvature distribution on the face (the model is colored appropriately for this purpose), or read the curvature value at a specific point.	

Surface deviation is the command for defining the selected face’s normal deviation from the specified direction. The colored face display allows watching the deviation all over the face. You can also measure the deviation at a specific point.	
Surface gap is the command for evaluating the gap between two or multiple selected faces. The command is used for analyzing models suffering import/export deficiencies.	
Normal deviation is the command for measuring deviation between the normals of the neighboring faces evaluated on the specified edges. The quills displayed in the 3D window help watching normal deviation across the overall edge extend. Besides that, the angle of the normal deviation can be measured at a specific point.	
Surface smoothness is the command for evaluating the model regularity. On entering the command, a special pattern is applied to the body's faces, usually made by long light and dark stripes. This pattern helps usually determine whether the adjoining faces make a smooth tangency transition, or smooth curvature transition.	
Model Separation Check is the command for identifying faces of the areas preventing a die cast or injection mold from opening. Additionally, the faces are highlighted, whose slant angle with respect to the selected direction is less than specified.	

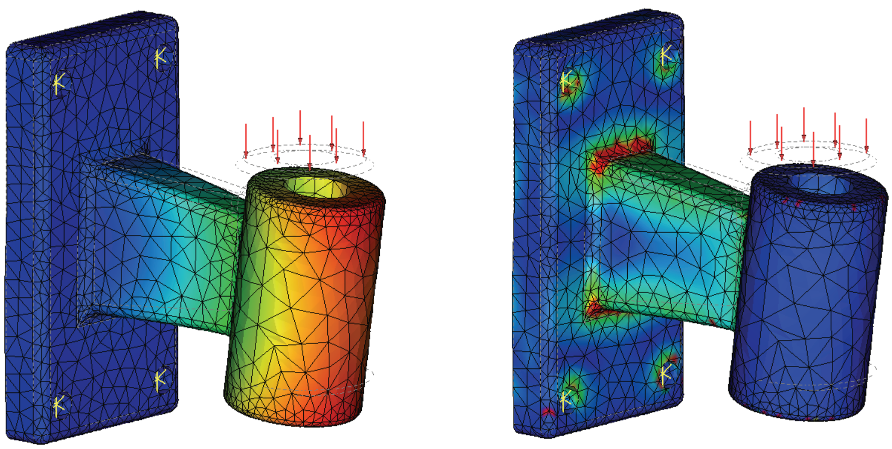
Engineering Analysis

Besides the geometry analysis commands described in the previous paragraph, the T-FLEX CAD complex provides additional modules serving to conduct advanced model analysis: **Finite Element Analysis** and **Dynamic Analysis**.

Finite Element Analysis – this is a "**T-FLEX Analysis"** module that serves to perform various types of finite element calculations:

- **Static analysis** allows calculating the state of stresses and strains in a structure under the impact of constant in time forces applied to the model;
- **Frequency analysis** allows calculating natural (resonant) frequencies of a structure and the respective vibration modes;
- **Buckling analysis** is important when designing structures, whose operation implies lasting influence of loads ranging in intensity;
- **Thermal analysis** is the module providing the capability of evaluating a heated product behavior under the impact of sources of heat and radiation.

T-FLEX Analysis is oriented at solving physical problems in the three-dimensional formulation. All calculations rely on the finite element method (FEM). The product's mathematical approximation uses its equivalent replacement by a mesh of tetrahedral elements. At the same time, an associative relationship is maintained between the three-dimensional model of a part and the finite element model used in the calculations. Parametric modifications of the original solid model are automatically propagated into the meshed finite element model.



Results of finite element modeling (displacements and stresses)

The standard T-FLEX CAD 3D distribution kit includes only a limited trial version of the finite element analysis module – **Express Analysis**. The Express Analysis is a light version of the "T-FLEX Analysis" module, specifically tuned for running simplified yet qualitatively sound strength studies. The user is provided with the necessary selection of load and restrained types. Based on the T-FLEX CAD model geometry, the automatic mesh generator creates a quality finite element discretization within the Express Analysis. Once the calculations are completed, the stress, strain, displacement and strength safety factor are output in a graphic form.

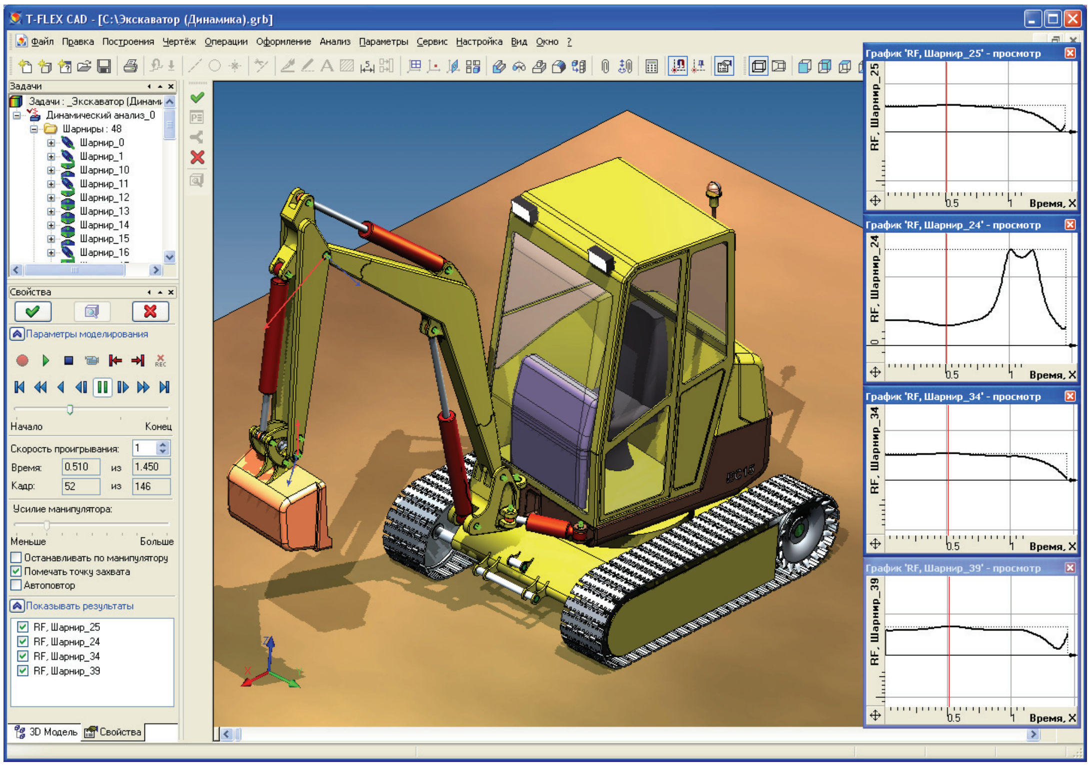
The fully functional "T-FLEX Analysis" finite element analysis module is sold separately.

> For details on working with the finite element analysis module, please refer to the "T-FLEX Analysis" manual.

Dynamic Analysis – this is the module that serves to run studies on dynamic behavior of various three-dimensional mechanical systems.

The dynamical analysis module is capable of solving the following studies:

- Analysis of the motion trajectory, velocity and acceleration of any point of a component in a mechanical system due to applied forces;
- Analysis of performance times of a mechanical system (the time to the target point, the time for vibrations to settle, etc.);
- Analysis of forces building up in components of a mechanical system during motion (reaction forces at the supports, joints, etc.).



A mechanism's model is defined as a system of solid bodies, joints, and loads. The data for the analysis is automatically accessed directly from the geometrical model created in the T-FLEX CAD system. The familiar tools of T-FLEX CAD are used in modeling, with mates and degrees of freedom employed to define relations between three-dimensional bodies. The system also provides the means of modeling the contact between arbitrary solid bodies, being capable of processing a simultaneous contact of hundreds and thousands of solid bodies of arbitrary shape.

Loads on solid bodies are defined as the initial linear and angular velocities, forces, moments, springs, gravity, etc. To read the results, special sampling elements are used. Numerous values are available for the analysis: coordinates, velocities, accelerations, reaction forces in joints, forces in springs, etc. The user can observe the model behavior from any viewpoint immediately during the actual calculation. One can create animation clips from the obtained results of a dynamic calculation.

The T-FLEX CAD 3D standard distribution kit includes only a limited trial version of the dynamic analysis module – the Express Dynamic Analysis. The Express Analysis has certain limitations on load types and the ways of rendering dynamic analysis results (the tools to obtain numerical calculation results are not available). In the commercial module, the calculation results are displayed as graphs, dynamic vector arrows and as an array of numbers (graph points).

The fully functional dynamic analysis module can be purchased separately whenever needed.

> More details on working with the dynamic analysis module are provided in the "T-FLEX Dynamic Analysis" manual.

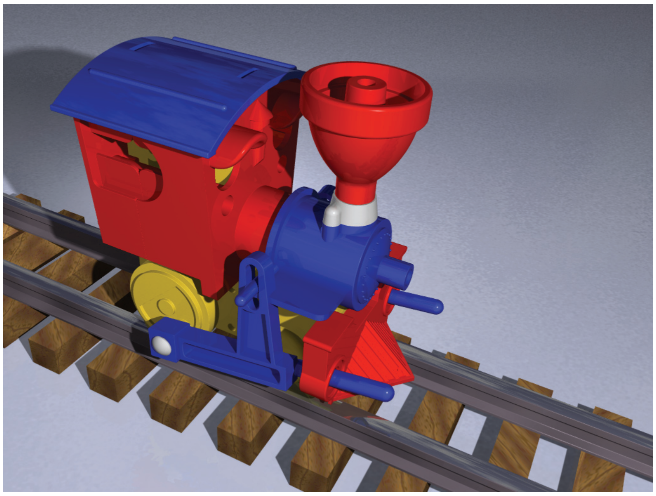
Auxiliary Elements and Operations

Material is a system element assigned as a parameter to each created body. Material helps rendering computer models so that they look like real objects. It contains a list of characteristics of the real material used in actual production. Material has the following parameters: density, luminosity, ambience, etc. The whole body material can be assigned within parameters of any operation.

Apply Material is an operation that assigns a material to specific faces of a body.

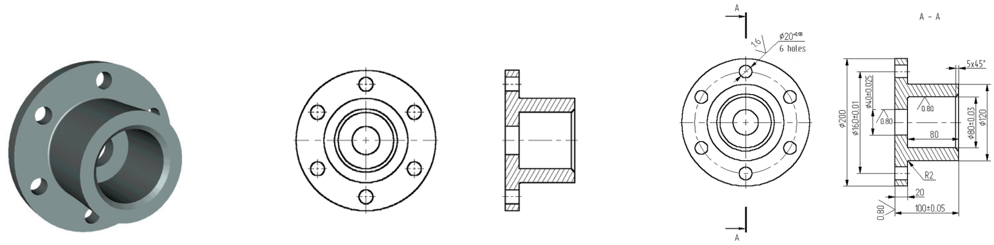
Transformation is a command for defining translation and rotation of an object that change its location and orientation on the 3D scene. This command works with all operations and most 3D construction entities.

Photorealistic Imaging is a command for creating a file in BMP format containing a photorealistic image of the objects on the 3D scene. This is done with POV-RAY application that is shipped together with T-FLEX CAD 3D. Photorealistic images are also used for creating animated clips.



2D Projection

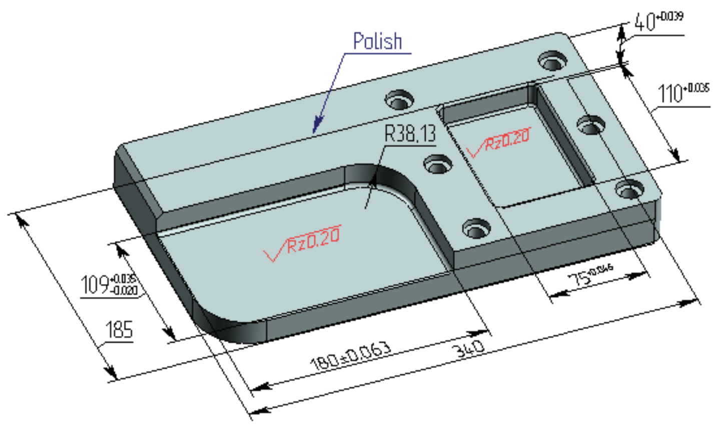
2D Projection projects all bodies of three-dimensional scene or selected bodies or specified elements on a plane. The resulting image is displayed in 2D window. Projection creation accounts for sections for instance when creating cuts. This approach helps avoiding additional constructions in 2D window. Simply create a three-dimensional body, and all necessary views in 2D window will be obtained by projection. Further, 2D projections can be used for laying out drawings.



3D Annotation

T-FLEX CAD allows creating drawing annotation (dimensions, leader notes, roughness symbols) directly on faces, edges and vertices of a 3D model.

The capability of creating 3D annotation allows introducing in a 3D model not only geometrical, but also technological process and other information that can later be used when creating drawings by 2D projections, as well as in other applications, for example, in the process design modules or when creating CNC control sequences.



3D Object Rendering

View is a combination of 3D window parameters, such as view point, distance to object, visualization parameters, the type of projection, etc. Particular sets of these settings can be saved in order to quickly set up 3D scene in required configuration later.

Rendering type is a way of rendering 3D bodies in the 3D window.

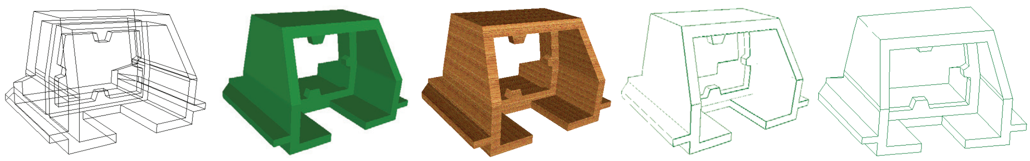
First type is **Wireframe model**. This way is convenient in that the front elements do not obstruct the ones behind, and the objects inside the body can be seen as well.

Second type is **Shading**. The faces of the bodies are displayed in a specified color.

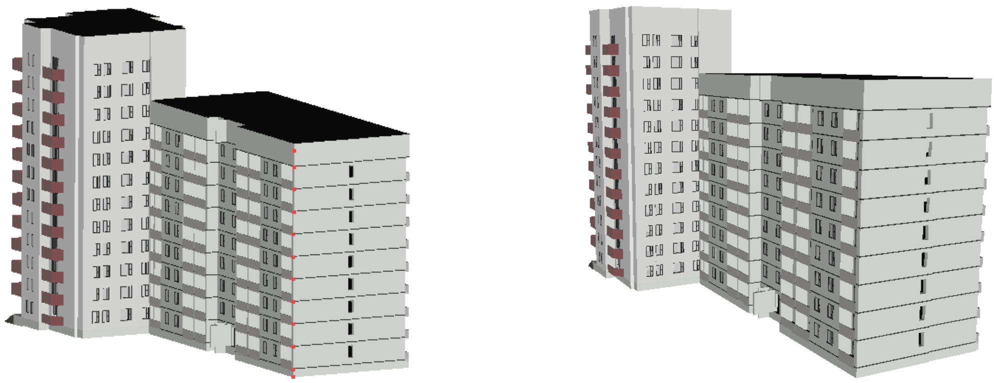
Third type is **Shading with Material**. The faces display accounts for the material specified for the whole body, as well as applied to particular faces.

Fourth type is wireframe with **Hidden line removal**. A fast algorithm is used for determining line visibility.

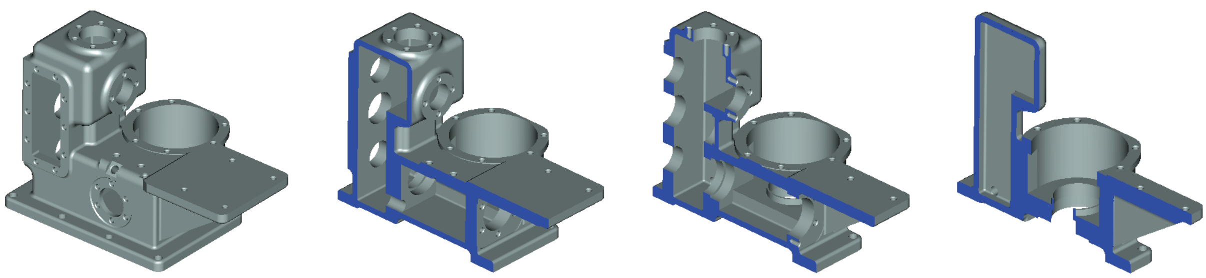
Fifth type is wireframe with **Precise hidden line removal**. The wireframe is displayed with invisible lines removed per the current orientation.



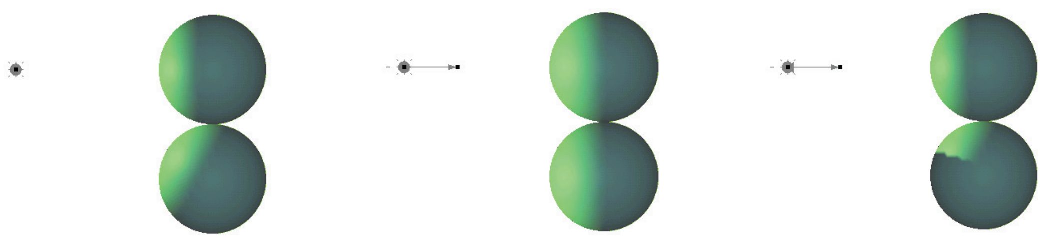
Projection method defines the way objects are represented on the 3D scene. It may be parallel projection without perspective that does not account for the distance from eye to object and the perspective angle, or projection with perspective that does account for the above parameters.



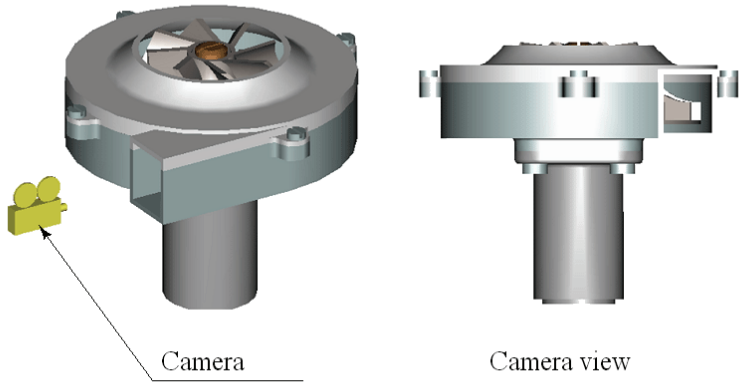
Clip plane is a plane situated parallel to the screen when originally defined. It can move back and forth along the fixed direction and clip out the portion of the objects on the scene before the plane. It is used for visual analysis of inner elements of the bodies, and for selecting objects on the 3D scene located inside the bodies.



Light Source is an element for controlling the illumination of the 3D scene. Originally, there is a single light source coinciding with the viewing point. A light source can be a Spot Light, a Direction Light, and a Projector. The user can create additional lights as needed, vary their intensity and direction, turn on or turn off any of them. Light source is used for creating a photorealistic image.



Camera is an element that defines the viewing point and direction on the 3D scene. Each 3D window has one default camera. One can create additional cameras, select an active camera. The newly created cameras are bound to a local coordinate system, and can be relocated together with it or relative to it. The viewing direction of a camera can also be changed. This helps surveying inner elements of the scene and create animated clips.



Three-Dimensional Model Animation

Animation of a three-dimensional model is done by the same means as drawing animation, that is the variable-driven model modification. The user assigns the range and the step for the variable modification. Since the real-time regeneration of a three-dimensional model driven by a variable modification may be slow, the animation may not look well in interactive session. This is why a good strategy is to create a multimedia file in *.AVI format (a video clip) for watching the modification animated.

A video clip may be created using photorealistic imaging. In this case, each frame of the clip will be processed by the POV-RAY application.

Approaching Solid Modeling with T-FLEX CAD 3D

General Recommendations before You Begin with 3D Model Creation

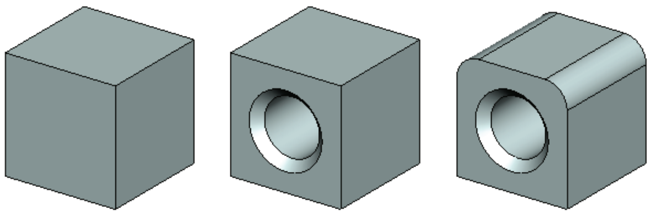
Analyze the design intent before starting 3D model creation. The degree of design automation depends on how well the engineer thought through of the model layout. Parametric design requires additional time investment at the initial stage as compared to non-parametric approaches. However, this brings an unmatched advantage at the later stages, as, for instance, in preparing documentation for various modifications of a product.

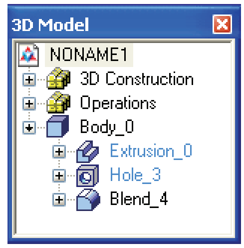
> Due to high flexibility of the system, the same result can be achieved by different means. One of the foremost goals of the designer is to find an optimum solution. Naturally, this depends on how well the designer manages to operate the instruments provided by T-FLEX CAD 3D system.

First, decide what operations you will use for creating the elements of a part. Important is what kind of references and interdependencies will be imposed on various elements. Define the appropriate boundary conditions. Determine what model parameters should be represented by variables. Divide a complex assembly into a set of fragments, 3D pictures, copies, and library elements. Begin with a model creation after realizing a rough plan of the design layout.

Parameterization and Model Regeneration

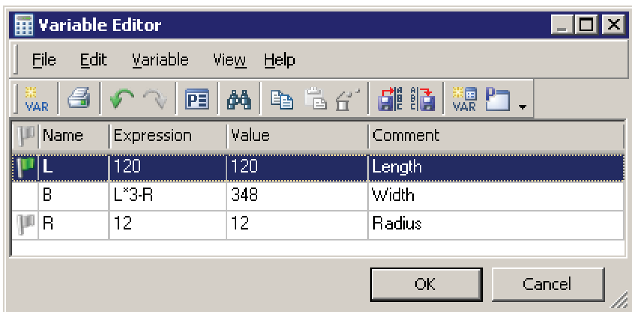
A T-FLEX CAD model is constructed as follows. First, create a new Body using 3D operations based on construction and other auxiliary 3D elements. Next, a base body is used to create other Bodies, which, in turn, undergo modifications and alterations, become subject to certain dependencies, boundary conditions, and so on. Each construction element or operation is recorded in the model history. It is possible to identify parent-child relations between certain elements. The model hierarchy is represented by a tree structure.



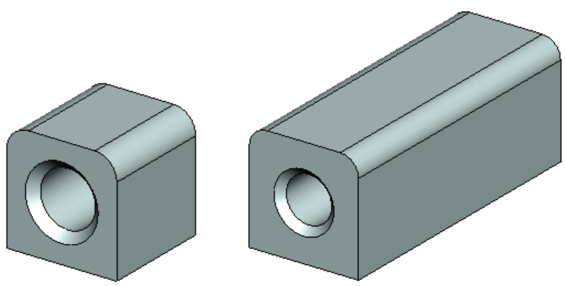


Now, suppose, some parameter of a parent element operation needs to be modified that would affect its geometrical configuration. This would cause the children to adjust their locations according to the parent geometry and the model parametric dependencies. T-FLEX CAD is capable of handling such issues due to its architecture that supports through-model parameterization.

The word “parameterization” implies a wide range of capabilities. It is possible to modify virtually any parameter of any operation at any time. Besides, parameters are not limited to numerical or textual values, as they also can be represented by variables. The variables can be driven by certain expressions, and be dependent on other parameters and variables.



Again, suppose, we changed a parameter of a parent element. Next, the model regeneration is launched manually or automatically, traversing the model structure and recalculating the model with the updated variable values.



T-FLEX CAD supports full and partial regeneration. Full regeneration is necessary for updating the whole drawing and the model. This causes recalculation of all objects from scratch. Partial regeneration saves time. The system automatically analyzes what objects were modified since the last regeneration, and recalculates these objects and their children only.

To update a model use a selective regeneration command,

Keyboard	Textual Menu	Icon
<Alt><F7> or <3G>	**“Tools|Regenerate”**	

The full regeneration is invoked by the following command,

Keyboard	Textual Menu	Icon
<3RG>	**“Tools|Full Regeneration”**	

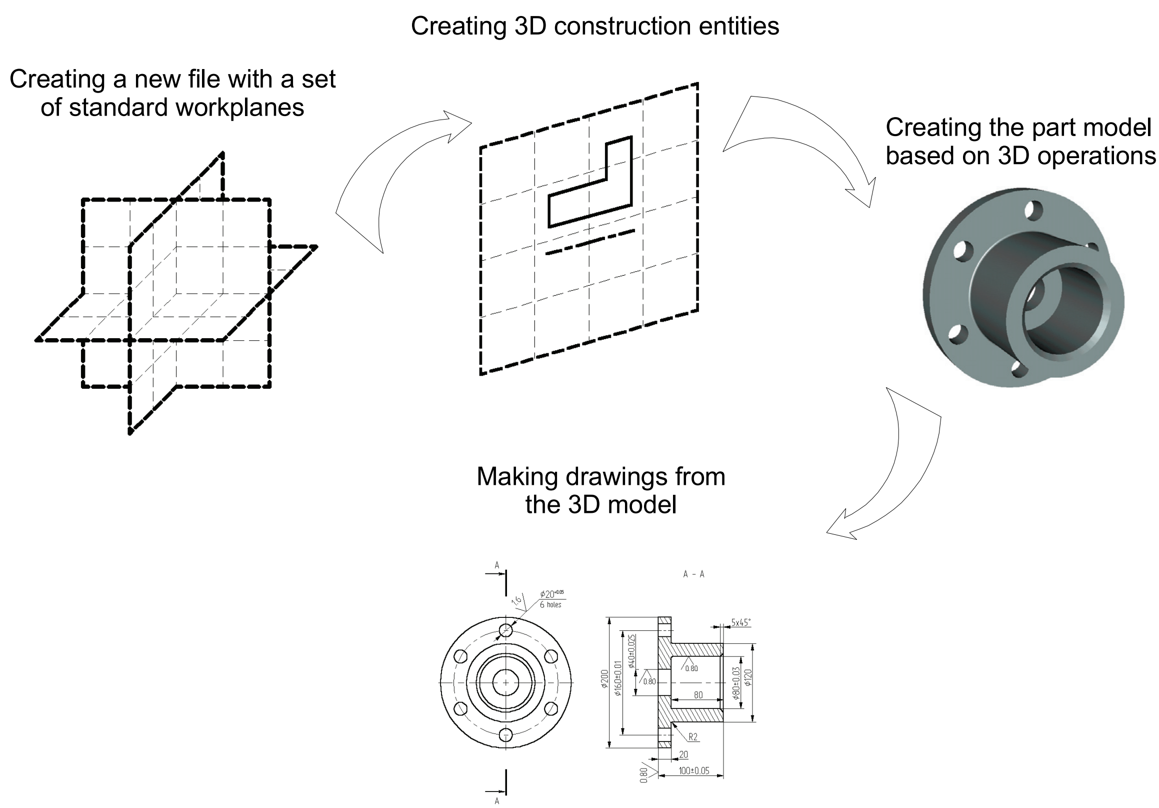
Three-Dimensional Model Creation Techniques

T-FLEX CAD supports various techniques of 3D model creation. The **mainstream** design principle implies direct approach to a 3D model creation “from scratch to workplane to sketch to model”. Another approach implies use of pre-designed 2D drawings or auxiliary 2D construction. This technique can be called “**from 2D to 3D model”**.

Whichever technique is used in a 3D model creation, the same operations form the body geometry. The difference is in the ways of creating the 3D construction entities.

The Mainstream Approach

Charted below is the mainstream design approach workflow.



Begin a document creation by selecting an appropriate prototype of a 3D model (`3D model.GRB`). Use the command **“F3: Create New 3D Model”**. This brings up a 3D window with a standard set of workplanes. Now all is set for starting with a 3D model.

> The mainstream 3D modeling approach does not require the 2D window. All auxiliary elements, including profiles, nodes, 3D paths, etc., can be constructed directly in the 3D window using the same 2D drawing tools.

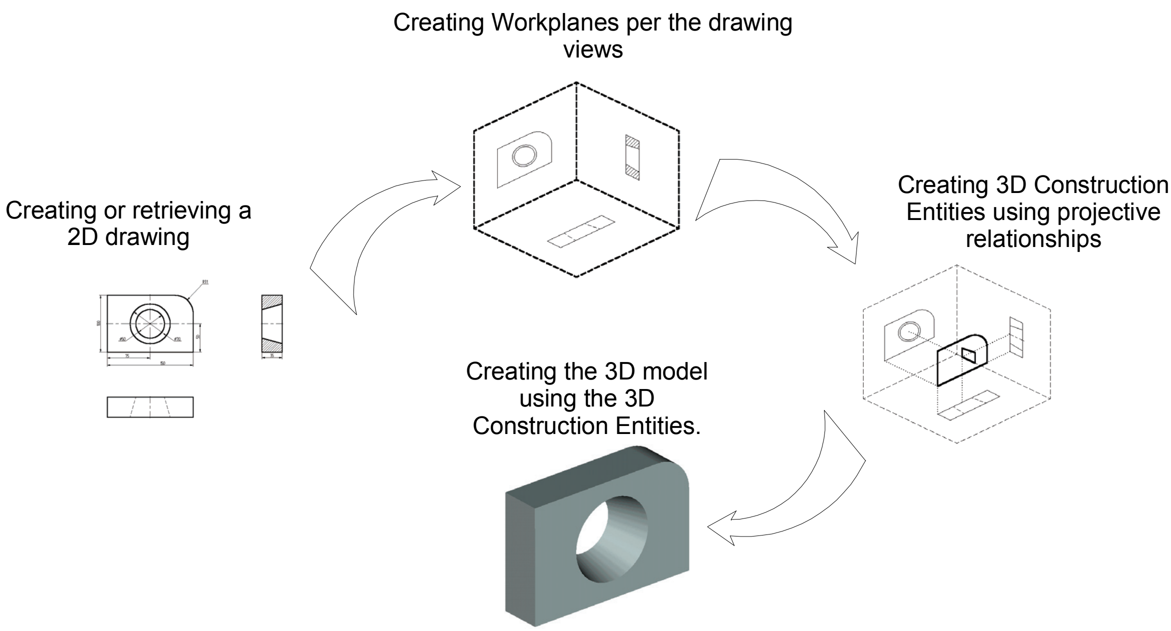
Normally, an operation requires a certain set of auxiliary 3D construction entities to be completed. To create a 3D profile, first select a workplane or a flat face as a base, then all 2D drawing commands become accessible. Just like in 2D, one can draw on the active workplane new lines, contours, etc. Parametric dependencies can be automatically introduced into the model at this early stage. A 3D operation, such as, say, an extrusion, can be invoked without leaving the drawing mode. The system would automatically create a 3D profile based on the just created graphic lines. This minimizes the number of user actions required for completing an operation. Consider, for instance, the “Rotation” operation. The required axis (a dashed line) and the contour can be created on the fly directly on the workplane. Preview is always available to evaluate the result of an operation. Preview activates automatically for the operations that do not require extended time for computations. In other operations, it is invoked by a special command.

The bodies created at an early design stage may later be used in other operations, such as in Booleans, blends, tapers, etc.

The completed 3D model can be used for generating drawings, if desired. Create required projections, sections and cuts that can be used as references for the necessary elements of the drawing layout, dimensions, etc.

“From 2D to 3D” Technique

The following chart illustrates a design technique “From 2D to 3D”.



Often, a 2D drawing of a part may already be available before beginning with the 3D model creation. The proposed approach to model creation is preferable in such a case. At the initial stage, it is good to have a 2D drawing of the part that contains all necessary views. The part views are better be laid out in accordance with the projection rules, although this is not a strict requirement. Certain parametric relationships of the pursued parametric 3D model can be introduced at the 2D drawing stage. The first step in 3D model design is creation of workplanes. Often, two or three orthogonal workplane are sufficient that represent the front view (the front plane), the top view (the horizontal plane), and the view from left (the side plane).

Next, the necessary 3D construction entities are introduced, such as 3D nodes and 3D profiles. 3D nodes are used as references for 3D profiles, vectors of extrusion direction, rotation axes, etc. In case a workplane can't be activated, the required profile can be defined based on a 2D hatch. The hatch must reference an actual drawing.

Besides the profiles, 2D views can be used for constructing 3D nodes from the 2D nodes, and 3D paths from the 2D paths and other construction entities.

> It is thinkable of combining the two techniques described above. Note that three-dimensional modeling is a creative process that does not necessarily implies a sole way towards a particular solution. The user is provided the necessary set of tools, while it is up to him to make a choice of a most effective approach to realizing the design intent.

Creating Assemblies

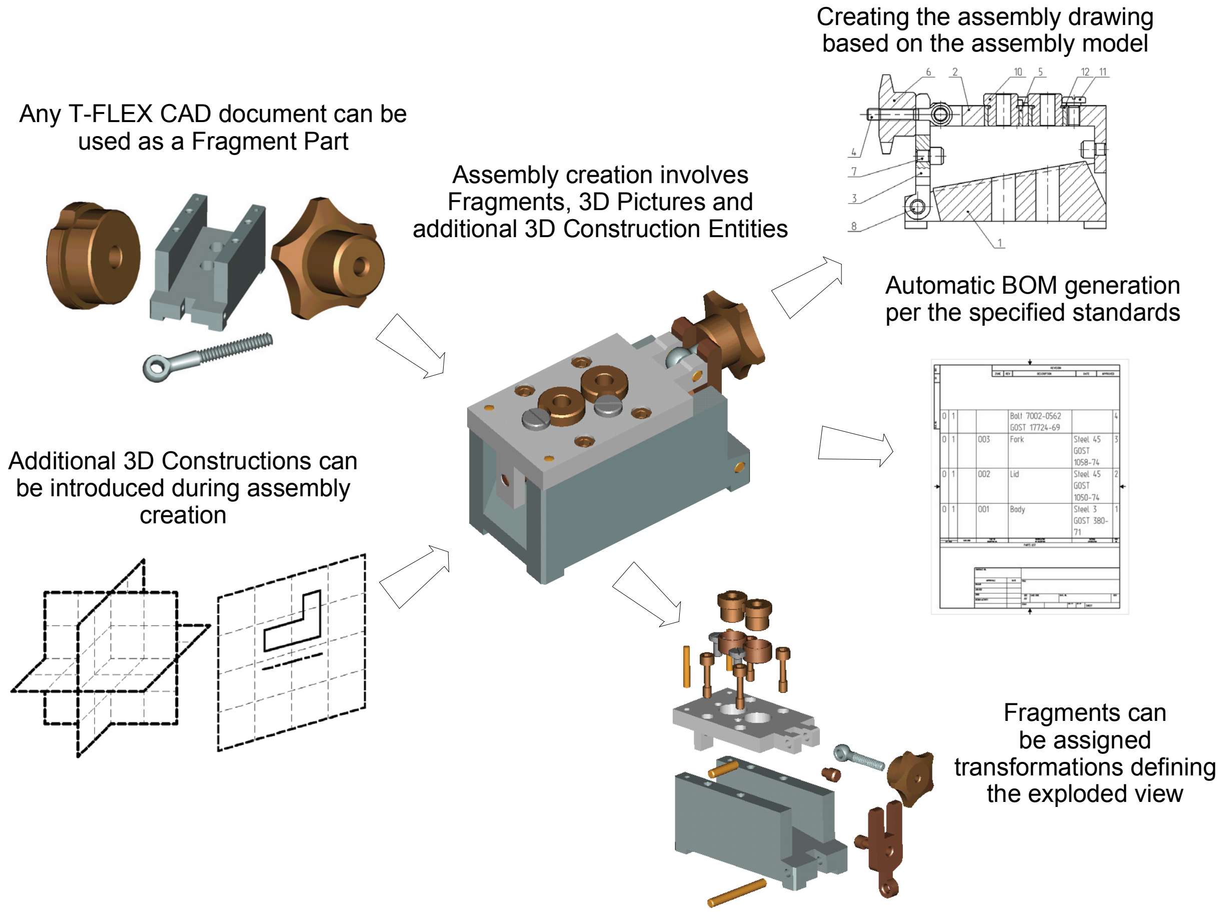
Any T-FLEX CAD 3D document that contains a three-dimensional model can be inserted into another 3D model as a fragment. Thus composed model is called **assembly**. External models, imported from other systems in an appropriate format, can also be used in assemblies.

The component architecture of T-FLEX CAD assemblies has certain advantages. Thus, one can create libraries of parametric elements, to use them later in assembly creation.

An assembly document keeps the references to the fragment files. Once a fragment file is modified, it automatically becomes updated, causing modification of the respective assembly component. Any fragment may have external variables that drive the parametric relationships of the part. At any time, one can either modify the original part file of the fragment, or specify different values of the external parameters of the fragment. In the latter case, the part file is unchanged, while the assembly component – the fragment – is recalculated per the new external parameter values. A special functionality helps keeping various sets of the assembly parameter values thus facilitating quick loading of a desired variation.

Each fragment file provides storage for BOM data. If this data is entered in each fragment file then the assembly BOM can be output automatically.

The assembly model can be used for creating drawings by creating the necessary views, cuts, sections and applying dimensions and other layout elements.

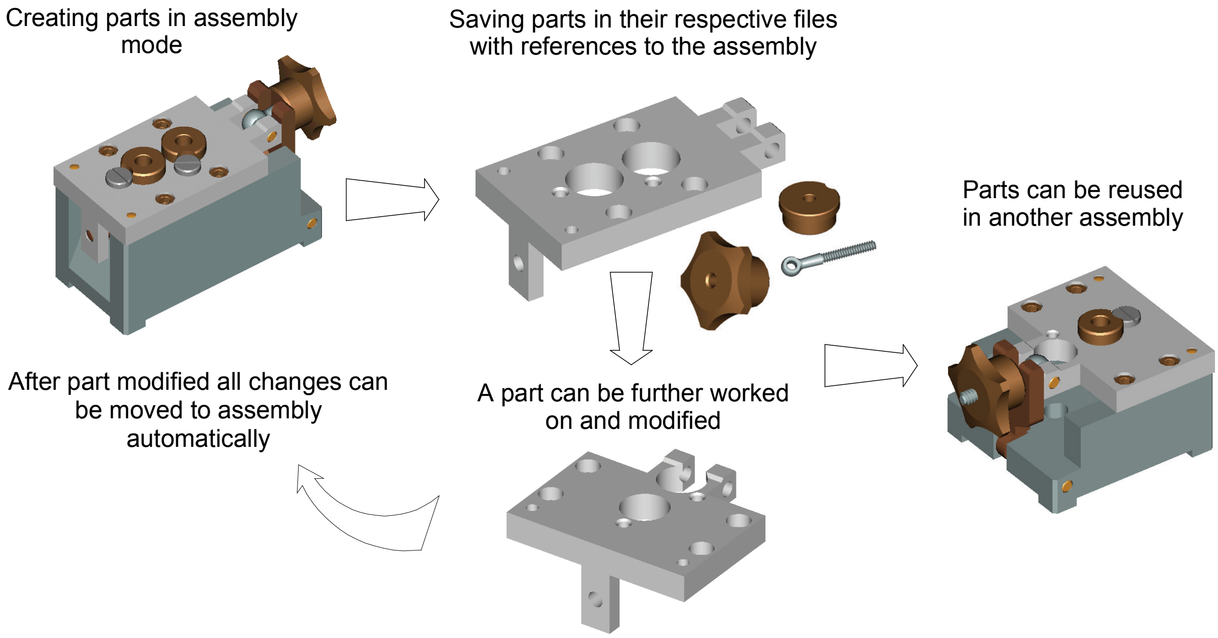


“Top-down” Assembly Modeling

There is an alternative approach to assembly creation, as opposed to the typical one briefly described above. T-FLEX CAD supports new part creation based on any geometrical and topological elements of the parent assembly. In this way, it is no longer necessary to specify placement conditions and the part mutual configuration. A part is automatically placed according to the references to elements used in the part definition. The parametric relationships among the assembly elements are saved with the model. Once a dimension or a part placement is modified, all related elements of the assembly will automatically adjust.

When working with a part in assembly mode, all elements that are not referenced by the part are shown transparent. Snapping works with all assembly elements. Any assembly element may be referenced at any moment.

The part is saved in a separate file. The file may be opened apart from the assembly and worked on separately. The assembly references are meanwhile preserved.



3D Model Rollback Mode

If you need to make a correction to an already created model, you can use the 3D model rollback mode at a certain operation level. This function is helpful in the cases when you need to do additional work in the middle of the model tree.

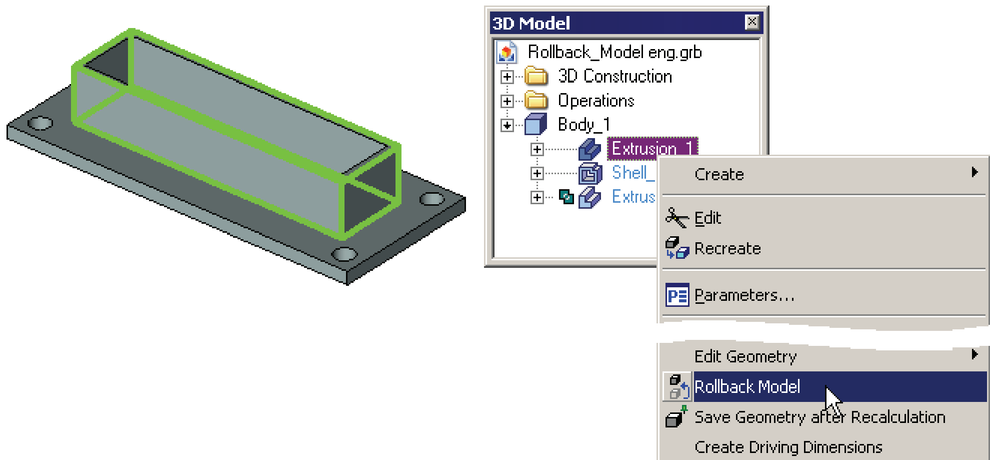
In the rollback mode, the 3D model returns to an earlier stage of its creation. Elements and operations that follow after the rollback point in the model's history become hidden in the rollback mode and are inaccessible for selection. In this state, those are marked with semitransparent icons.

All elements and operations created in the rollback mode are automatically inserted in the 3D model tree between the operation, upto which the rollback was made, with the successive operations unloaded from the scene. It is possible to do a series of rollbacks.

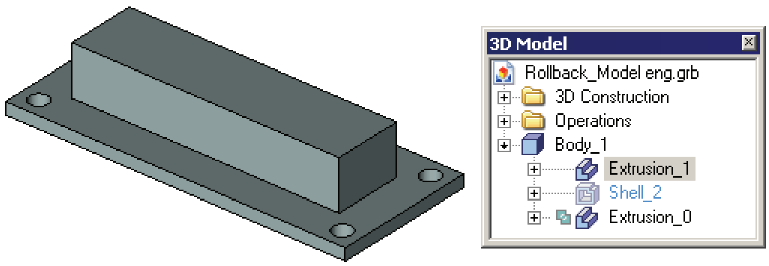
Upon finishing the rollback mode, the system automatically rebuilds internal model relations to restore the later operations. If for some reason the introduced changes result in errors in the model, then the system will offer restoring the model state before the rollback. This protects the user from introducing invalid modifications to the model.

The rollback mode is invoked and finished using context menu commands of an operation in the model tree or in the 3D view window. The rollback will be up to the level of the operation, from whose context menu the command was called.

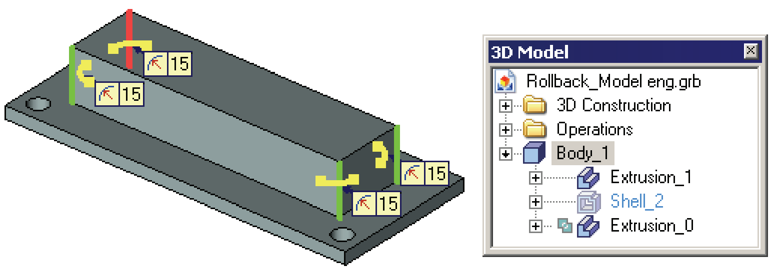
Let's review the use of a rollback on a simple example. In this model, it is necessary to add a blending on corner edges.

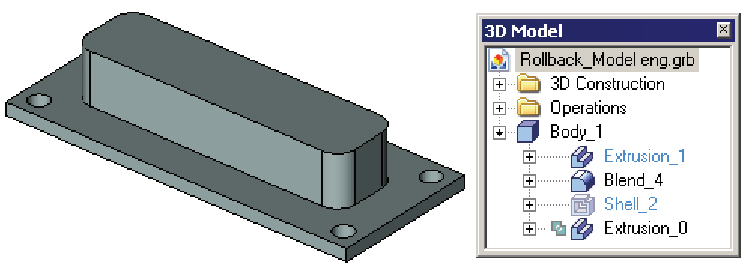


We perform the model rollback up to the base extrusion operation. In this case, the operations Shell_2 and Boolean_3 are unloaded from the scene.

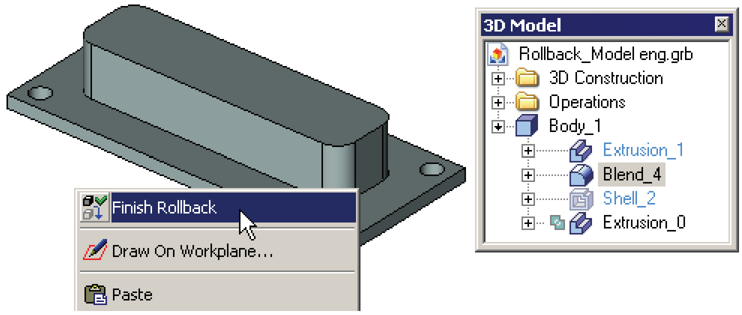


Then we make an edge blending (the Blend_4 operation is added in the model tree).

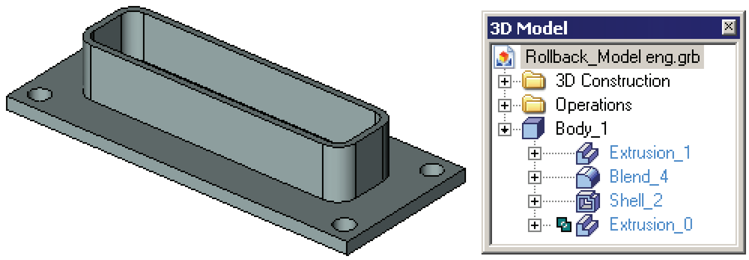




We then finish the rollback.



As a result, we get the model presented at the following figure.



T-FLEX CAD 3D System Operation Tips

Getting Help

There are several ways of getting answers to the questions arising during the work process, as follows:

- Help on the current command can be accessed by pressing ***<F1>*** key or by selecting the menu command **"Help|Current"**. When no command is active, pressing ***<F1>*** or selecting **"Help|Contents"** invokes the help contents.
- While within a command, limited information is provided as prompts and hints in the status bar.
- Pop-up help appears by the icon buttons on a toolbar and by the graphic elements when the cursor is briefly held pointing at an element or item. The pop-ups report the command or element name, the type or parent operation of the element. Pop-ups messages are duplicated in the status bar at the bottom of the system window.

Creating a New Document. Using Prototype Templates

A new project begins with setting up the new document. Depending on the design intent one can select appropriate initial options for the new model. To begin with a 2D drawing creation or other 2D construction in 2D window use the command **"FN: Create New Model"**:

Keyboard	Textual Menu	Icon
<F><N> or ***<Ctrl><N>***	**"File|New 2D Model"**	

This will bring up the 2D window.

When beginning directly with a 3D model creation, use the command for creating a new file,

Keyboard	Textual Menu	Icon
<F><3>	**"File|New 3D Model"**	

This brings up the 3D window with the standard set of workplanes.

A new document is created using one of the prototype files whose names are defined via he command **"Customize|Options…"**. These files may contain some useful elements and settings that will be created or set in the new document. In the case the new document settings need to be changed, edit the prototype file (such as, for instance, the ANSI drawing prototype file, named `ANSI.GRB`, or a 3D model prototype `ANSI 3D Front Right Bottom Workplanes.GRB`). These prototype files must be present in the folder …`T-FLEX CAD\Program\Template`.

It is possible to create any number of prototype files and use them as desired. For creating a new file from a prototype, use the dialog "New From…" invoked by the command **"FP: Create New Document Based on Prototype"** (**"File|New from prototype…"**). In addition to that, this dialog is available in the window "Start Page".

Each folder in the directory …`T-FLEX CAD\Program\Template` becomes a tab in the dialog box. All the prototype files that exist in a respective folder are displayed on the selected tab. Thus, the prototype files should be placed in the directory …`T-FLEX CAD\Program\Template\<The respective folder of the prototype file>`.

To save a file as a prototype use the menu command **"File|Save as Prototype"**. Use the appearing on screen dialog box to assign a name to the would-be prototype file. One can create new folders (turning into tabs on the dialog box) for prototype files as desired. One can also delete any prototype files and folders, except the folder "Common".

Mouse Interface. Context Menu

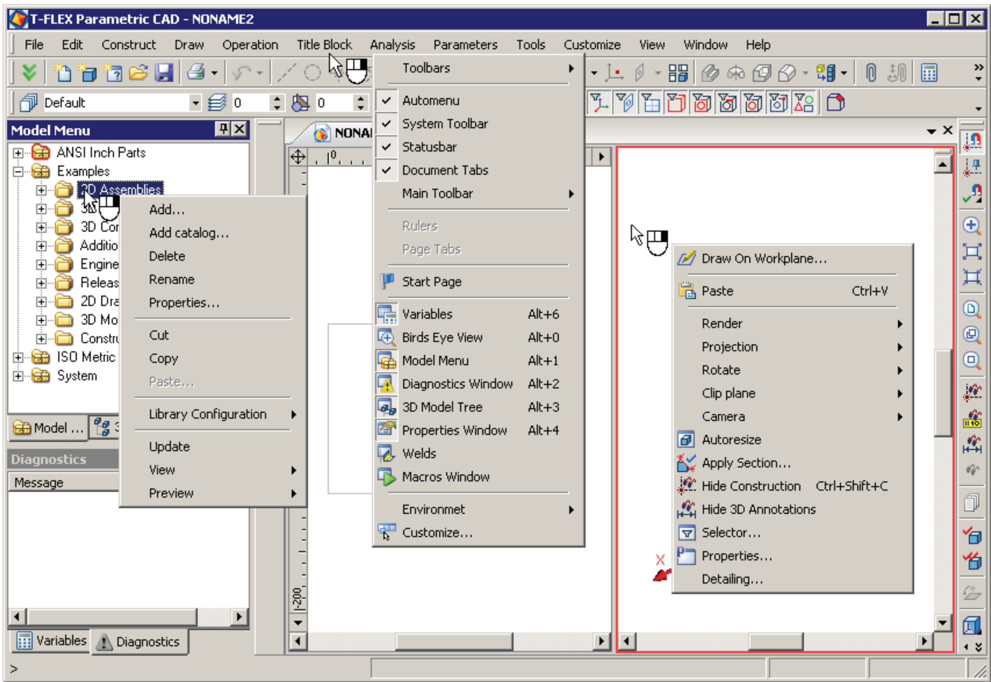
Part modeling in T-FLEX CAD 3D is performed mostly by mouse. The keyboard is only used for inputting numerical values, names and command accelerators (see below).

Using left mouse button

- Pointing cursor at an icon and pressing launches the command represented by the icon.
- A command can also be launched by pointing cursor at a textual menu item and pressing .
- Pointing cursor at an object on the 3D scene and pressing selects the object.
- Pointing cursor at a 3D construction element or operation and double clicking invokes the dialog box "Element/Operation Parameters".
- Pressing and drugging spins the 3D scene. The cursor must be within 3D window bounds. The system should not be in drawing mode.
- Drag&Drop can be used for managing libraries and customizing toolbars and dialog boxes. Point the cursor at an element, press and hold , and drag the element to a desired location while holding the button. For more information, refer to the appropriate sections of documentation.

Using right mouse button

- While within a command, pressing cancels the last active command.
- Pressing while selecting elements on the 3D scene unselects the last selected element.
- If no command is active, pressing right button invokes the **context menu**. The menu is composed of the commands currently available for the particular element. The content of the context menu will be different depending on where the cursor is pointing. These could be the area of the 3D scene, a model element, the command area of T-FLEX CAD, such as a dialog box, a menu, or a toolbar. To launch a command point the cursor at the corresponding line of the context menu and press .



- Context menu can also be invoked while working with dialog boxes. See the topic “Context menu for the dialog fields”, the chapter “Customizing Drawing” in the volume “Two-dimensional design”.

The described right mouse button functions are set by default, but can be customized. To do so, call the command **“Customize|Options”** (**“Preferences”** tab). For more information, refer to the chapter “System Customization” in the volume “Two-dimensional design”.

In “3D drawing” mode mouse manipulations are the same as in 2D window.

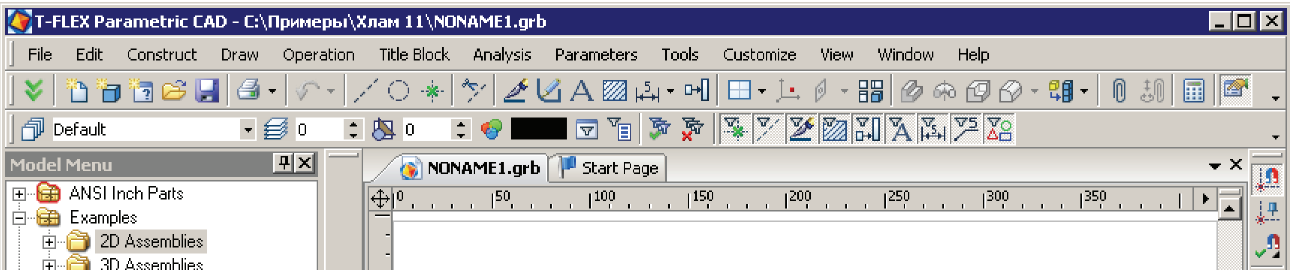
Using mouse with wheel (IntelliMouse)

- Spinning the mouse wheel zooms in and out.
- Dragging the mouse with the wheel depressed pans the 3D scene.
- The mouse wheel can also be used for alternative selection from a set of objects under the cursor. Alternative selection mode activates after two-second delay after pointing cursor at the object. Scrolling through the list of objects is done by spinning the wheel.
- Spinning the wheel performs the standard functions of scrolling in the appropriate field of dialog boxes.

Calling Commands from Keyboard, Using an Icon, from Textual Menu

There are various ways of calling T-FLEX CAD commands. Firstly, a command can be selected via an icon on a toolbar using the mouse.

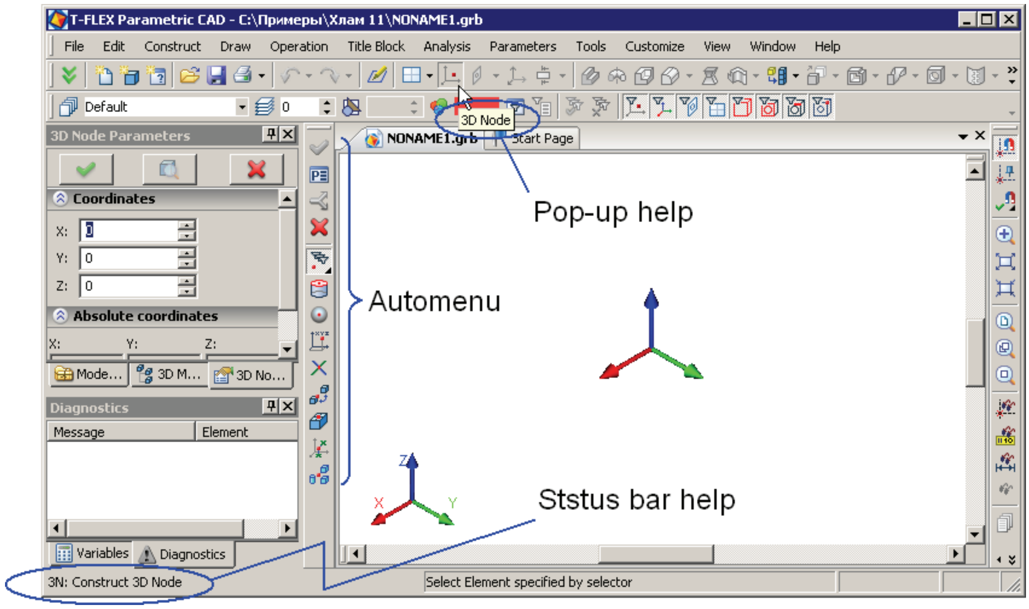
A command can also be launched from the textual menu. All T-FLEX CAD commands are grouped in a certain way. Each group has an entry item in the menu.



Thus, for instance, the commands that create 3D operations are grouped under the entry “Operation”. The drawing commands are combined under the entry “Draw”. The 2D and 3D construction commands are united in the “Construct” group. The file managing commands are united in the menu “File”. Other groups include the editing commands (“Edit”), the variable managing commands (“Parameters”), utility commands (“Tools”), the system and the model customization commands (“Customize”), the visualization controls (“View”), the window management (“Window”). The system reference and information group of commands is under the “Help” entry.

Most T-FLEX CAD commands are bound to function key combinations (simultaneously pressed), or keyboard accelerator sequences (pressed in order). The textual menu item buttons contain keyboard accelerator sequences for the commands next to the command name, whenever defined. The keyboard accelerator can be changed for any command. For detailed description, refer to the “System Customization” topic of the chapter “Customizing Toolbars and Keyboard”, “Keyboard” tab, in the volume “2D design”.

As said before, some commands can be launched by typing the keyboard accelerator sequence. Particularly, this is supported for 3D model and drawing creation and editing commands. The keyboard sequence and the command name is shown on the pop-up help and in the message field of the status bar.



Just like the 2D subsystem of the T-FLEX CAD, each 3D command has an additional set of options and subcommands accessible via the automenu – a toolbar containing subcommand icons, and function key combinations. The function key combinations are displayed in the pop-up help widgets.

Certain commands are best accessed via the context menu. This menu is available after selecting one or several elements on the 3D scene, such as faces (see diagram below). The context menu contains the list of commands available with the selected set.

Confirming actions of 3D element creation

Unlike 2D elements, a three-dimensional element creation takes several steps. Not all of these steps are required. To complete an element definition, use the provided option “Finish input” by clicking on the icon  or pressing ***<Y>*** key. This option becomes accessible only after the required minimum sequence of actions has been completed.

> When calling a command, make sure about the command exiting option. Some commands remember the current state while the others revert to the initial settings.

Canceling and exiting a command

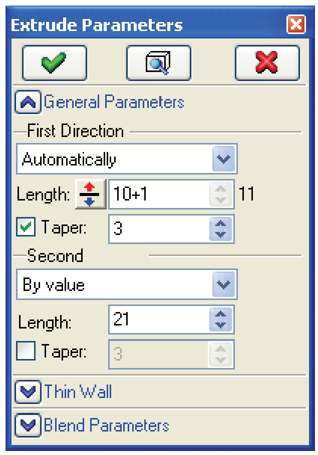
Exiting a command is done by pressing ***<Esc>*** key or . Alternatively, use the  automenu icon.

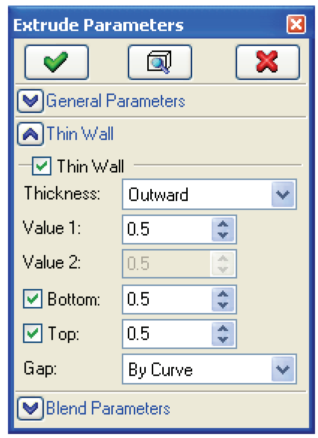
If the system has entered a 3D command without doing anything (nothing was selected), then calling another 3D command makes the former command quit. However, if something has been selected within the former command then the newly launched command does not close the former one and becomes nested. Once done with the nested command, the system returns into the former command. This is different in 3D mode from the 2D mode operation. The above does not affect the 3D drawing mode. To return to the command-waiting mode, close subsequently all active commands. Instant exiting from all nested commands is done by simultaneously pressing ***<Shift><Esc>***. A 3D command quits automatically upon calling any 2D command.

Setting up Parameters of a New Element

The parameters can be assigned to an element being created or edited at any stage of the creation or editing. To specify an operation parameters use the **Property window** that supports transparent mode, or the **Parameters dialog box** that always requires confirmation before the input takes effect.

The parameters dialog is necessary for specifying general properties of an element or operation, such as color, level, material, etc., as well as the transformation parameters. These properties are uniformly defined for all construction elements and operations in T-FLEX CAD. The ways of handling these properties are described in details in a separate chapter of this volume. This dialog box can be used for editing operation properties without calling the operation editing command. This is especially helpful when the operation being edited is deep in the model history. Otherwise, the model rollback would be required up to the edited operation, consuming considerable time and system resources. Now, it is sufficient to select the operation of interest in the model tree and call the parameters dialog using, for instance, the context menu. The parameters dialog box may contain several tabs. The common parameters and the transformation parameters are always present and are located on their own separate tabs. These parameters are common for most of the system elements; therefore, their handling is isolated into a separate chapter of this volume. All other tabs in the dialog box repeat those in the property window. Due to this, the related chapters will describe main parameter management based on the example of the property window.



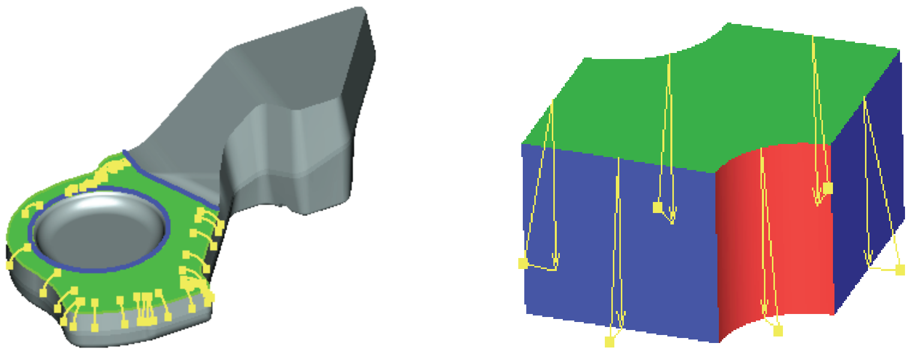


The property window is typically divided into several sections. The number of sections depends on a particular operation. Each section of the window can be expanded or collapsed using the buttons and . Once expanded within a particular command, this section state will be remembered for this particular command, and automatically accounted for in the subsequent command calls.

There is a field to the right of a parameter input box that displays the current parameter value. This is helpful in the cases when the parameter is defined via a variable or an expression, or else when defined “from status” (that is, uniformly for the whole document).

Some automenu buttons are duplicated in the property window, such as the “Accept”, “Exit” and “Preview” buttons. The property window works together with the automenu. Pointing cursor at some areas of the dialog box causes activation of the respective options in the automenu, such as, say, selection options.

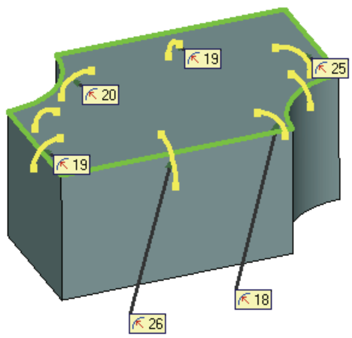
Special auxiliary graphic objects called draggers are introduced for dynamically controlling parameters of a 3D operation being defined. These appear automatically while within a command for creating or editing a 3D operation or a construction entity. Dragger manipulation is instantly reflected in the property window. The mouse-controlled dragger input defines the numeric parameter values of the operation. There can be several draggers on the scene simultaneously, providing control over various operation parameters. As an example, the “Bend” operation provides draggers for the bend angle, radius, offset and the two shifts. The “Blend” provides control over the rounding radius, etc. The values of the driven parameters are simultaneously updated in the property window of the operation being created or edited.

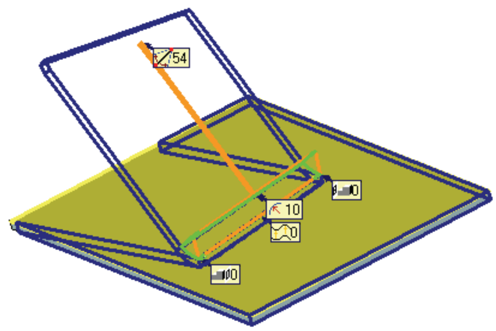


Examples of draggers in the commands “Blend Edge” and “Taper”

Marks with the current parameter values are displayed on the screen for each respective dragger. Besides the current value, a mark has an icon indicating the type of the displayed parameter. As the parameter value is modified via the property window or a dragger, the value on the mark changes dynamically.

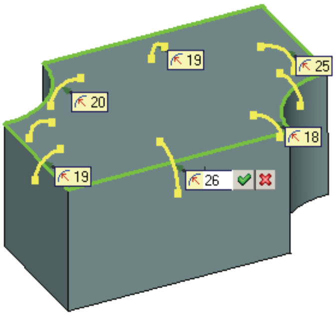
If it's too many marks and those interfere with working in the 3D scene, you can hide them or move sideways. Hiding marks is done in the current 3D command parameters. To move a mark sideways, point the cursor to the mark's icon. Once the cursor turns to , depress and, while holding the mouse button, drag the mark image to the desired location. The moved mark will be connected to the dragger with a leader line.

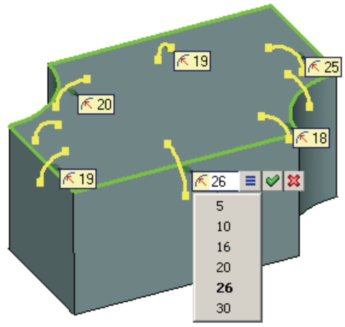




Marks serve not only to track the current parameter values, but also to modify them. To do that, click on the parameter value displayed in the mark. The mark will go into the editing mode of the parameter it controls. Just like in other system fields for editing values, the user can create a list of frequently used parameters 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 mark. **“Font”** command can modify font that will be used for displaying the dragger 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.

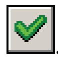




Preview

The following option is provided with each operation to see the result without completing the operation:

	***<F5>***	Preview Operation Result

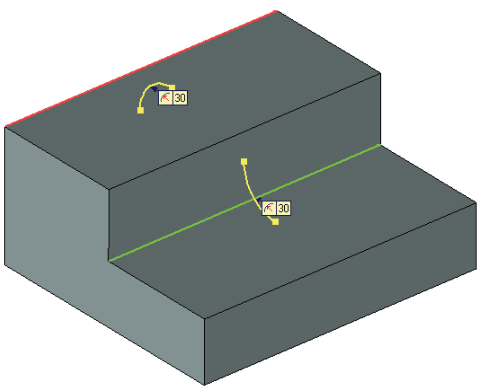
This option becomes accessible once the required parameters of the operation have been defined. Once this option is set, the displayed body starts to reflect the operation per its current parameters. In the case of satisfactory results, complete the input by pressing the icon . Otherwise, turn off the preview option by pressing on the icon once more, and adjust parameters as desired.

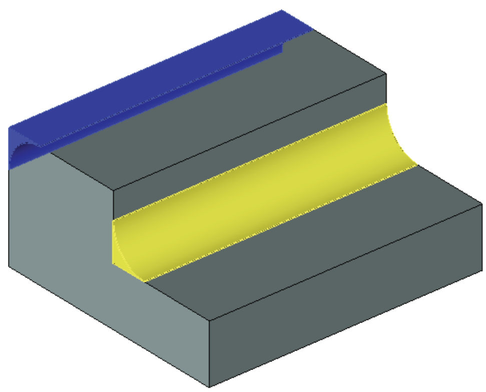
Preview of Operation Result

In the commands for creating and editing the modifier operations, there is an additional mode allowing a user to see the changes produced in the model by the current operation:

	***<Ctrl+F5>***	Preview Solid Changes

In this mode, the parts of the volume which will be added to the source body are shown with a yellow color, and the parts of the volume which will be removed from the source model are shown with a blue color.





Similar to the option for previewing, the option  is available only when all necessary elements for creating operation have been specified.

The Groups of T-FLEX CAD 3D Commands

Workplane and work surface management commands

	<3W>	Construct Workplane
	<3EW>	Edit Workplane
	<3SU>	Construct Work Surface
	<3ESU>	Edit Work Surface

Commands for auxiliary 3D element creation

	<3N>	Construct 3D Node
	<3PR>	Construct 3D Profile
	<3PA>	Construct 3D Path
	<3SE>	Construct Section
	<3O>	Create Local Coordinate System
	<3CA>	Construct Camera
	<3H>	Create Light Source
	<3PP>	Construct Pipe Path

Commands for auxiliary 3D element editing

	<3EN>	Edit 3D Node
	<3EPR>	Edit 3D Profile
	<3EPA>	Edit 3D Path
	<3ES>	Edit Section
	<3EO>	Edit Local Coordinate System
	<3ECA>	Edit Camera
	<3EH>	Edit Light Source
	<3EPP>	Edit Pipe Path

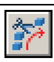
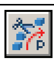
3D model creation commands

	<3X>	Create Extrusion
	<3RO>	Create Rotation

	<3B>	Create Boolean
	<3DE>	Create Blend
	<3DF>	Create Face Blend
	<3DT>	Create Three-Face Blend
	<3SL>	Create Loft
	<3SW>	Create Sweep
	<3SA>	Create Parametric Sweep
	<3PI>	Create Pipe Sweep
	<3F>	Insert 3D Fragment
	<3U>	Set Adaptive Fragment Parameters
	<3CT>	Create 3D Mate
	<3CP>	Create Copy
	<3SY>	Create Symmetrical Body
	<3MO>	Insert External Model
	<3I>	Insert 3D Picture
	<3AL>	Create Linear Array
	<3AR>	Create Circular Array
	<3AN>	Create Array By Nodes
	<3AP>	Create Array By Path
	<3AA>	Create Parametric Array
	<3SD>	Divide Solid
	<3SR>	Create Spiral
	<3SP>	Create Spring
	<3CU>	Cut By Section
	<3SH>	Create Shell/Offset Body
	<3TA>	Create Face Taper

	<3TB>	Create Body Taper
	<3AM>	Apply Material
	<3AT>	Create Thread
	<3H>	Create Hole
	<SMC>	Create new Base Part
	<SMB>	Create Bend
	<SMU>	Unbend Bend
	<SMR>	Re-bend
	<SMF>	Create Sheet Metal Forming Feature
	<SMP>	Sheet Metal Operation Defaults
	<3SS>	Sew Surfaces
	<3SZ>	Imprint Elements
	<3ZD>	Delete Faces
	<3ZX>	Separate Faces
	<3ZR>	Replace Faces
	<3ZC>	Change Faces
	<3ZT>	Transform Faces
	<3SX>	Extend Faces
	<3ZF>	Fill Hole
	<3DRT>	Scale/Twist Deformation
	<3DRB>	Bend Deformation
	<3DRS>	Skew Deformation
	<3DRC>	Sculpt Deformation
	<3DRV>	Deformation by Curve
	<3DRF>	Deformation by Surface
	<3SI>	Simplify

3D model editing commands

	***<3EE>***	Edit Operations
	***<3EX>***	Edit Extrusion
	***<3ER>***	Edit Rotation
	***<3EDE>***	Edit Blend
	***<3EDF>***	Edit Face Blend
	***<3EDT>***	Edit Three-Face Blend
	***<3EB>***	Edit Boolean
	***<3ESL>***	Edit Loft
	***<3ESW>***	Edit Sweep
	***<3ESA>***	Edit Parametric Sweep
	***<3EPI>***	Edit Pipe Sweep
	***<3EF>***	Edit 3D Fragment
	***<3ECP>***	Edit Copy
	***<3ESY>***	Edit Symmetrical Body
	***<3EM>***	Edit External Model
	***<3EAL>***	Edit linear array
	***<3EAR>***	Edit circular array
	***<3EAN>***	Edit array by points
	***<3EAP>***	Edit array by path
	***<3EAA>***	Edit parametric array
	***<3EI>***	Edit 3D Picture
	***<3ESR>***	Edit Spiral
	***<3ESP>***	Edit Spring
	***<3ECU>***	Edit Cut By Section
	***<3ESD>***	Edit Divided Solid

	<3ESH>	Edit Shell/Offset Body
	<3ETA>	Edit Face Taper
	<3ETB>	Edit Body Taper
	<3ESS>	Edit Sew Faces Operation
	<3EAM>	Edit Material Application
	<3EAT>	Edit Thread
	<3EH>	Edit Hole
	<SMP>	Edit Base Part
	<ESM>	Edit Bend
	<ESU>	Edit Unbend Bend
	<ESR>	Edit Re-bend
	<ESF>	Edit Sheet Metal Forming Feature
	<3ESZ>	Edit Imprint Elements
	<3EZD>	Edit Delete Faces
	<3EZX>	Edit Separate Faces
	<3EZR>	Edit Replace Faces
	<3EZC>	Edit Change Faces
	<3EZT>	Edit Transform Faces
	<3ESX>	Edit Extend Faces
	<3EZF>	Edit Fill Hole

3D model visualization commands

	<3AS>	Apply Section in 3D View
	<3CD>	Show Clip Plane Position
	<3CL>	Enable Clip Plane
	<3CS>	Set Clip Plane Position
	<3CW>	Paint Clip Plane Section

	<3RC>	Select Center Of Rotation
	<3RF>	Full Rotation
	<3RX>	Rotate about X axis
	<3RY>	Rotate about Y axis
	<3RZ>	Rotate about Z axis
	<3RS>	Rotate about global/local axis
	<3RA>	Automatic Rotation
	<3VA>	Auto-Resize 3D Scene
	<3VB>	Back View
	<3VF>	Front View
	<3VL>	Left View
	<3VR>	Right View
	<3VT>	Top View
	<3VU>	Bottom View
	<3VI>	Axonometric (front)
	<3VK>	Axonometric (back)
	<3VW>	View Wireframe
	<3VS>	View Shading
	<3VD>	View Render
	<3VH>	View Hidden Line Removal
	<3VZ>	Precise Hidden Line Removal
	<3VE>	Perspective Projection
	<3VO>	Orthographic Projection
	<3VP>	Set View Parameters
	<3VV>	Show Surface Curvature
	<3VX>	Exploded View

	<3VC>	Select Camera
	<3VG>	Move Camera
	<3VY>	View Ray Tracing

Geometry analysis commands

	<PM>	Measure Element or relation between two Elements
	<3MP>	Mass-Inertial Properties
	<QM>	Check Model
	<QI>	Check Model Intersections
	<QC>	Show Curve Curvature
	<3VV>	Show Surface Curvature
	<QD>	Show Face Deviation
	<QH>	Show Gap Between Faces
	<QN>	Show Normal Deviation
	<QZ>	Show Surface Smoothness (Zebra)
	<QS>	Check Model Separation

3D model management commands

	<3G>	Regenerate 3D Model
	<3RG>	Full 3D Model Regeneration

Commands for 2D drawing creation and editing based on three-dimensional model

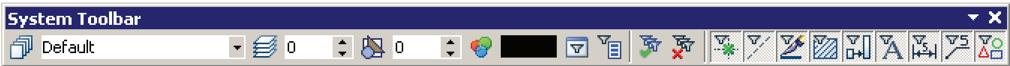
	<3J>	Create 2D Projection
	<3EJ>	Edit 2D Projection

Material creation and editing command

	<3MT>	Edit Materials

Customizing List of Element Types for Selection

The user can control the element types that are subject to selection by mouse. The selection filter icons are located in the right part of the system toolbar. For detailed description of selection filter management refer to the “Two-Dimensional Design”, the chapter “Main Concepts of System Operation”.



To manage the filters, make sure the 3D window is active. To activate the window, click anywhere within.

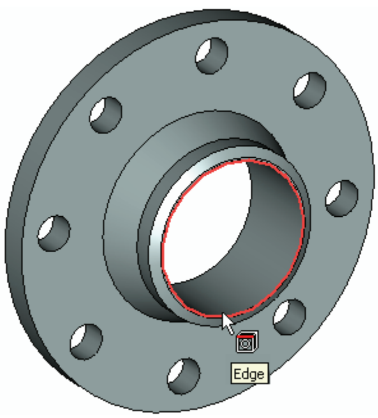
Element Selection

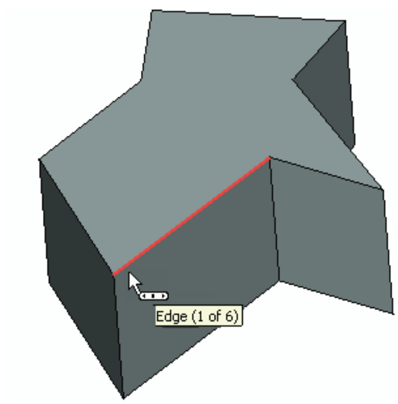
3D element selection can be done either in the 2D or in 3D window. A 3D element selection in the 2D window is possible only if this 3D element creation was based on 2D elements. For example, consider a 3D node defined by two 2D nodes. It can be selected either directly in the 3D window or via any of the two 2D nodes in the 2D window. Either way results in the 3D node selection.

The elements of a 3D sketch can be selected while in “3D Drawing” mode just the same way as in the 2D window.

Selecting a 3D element

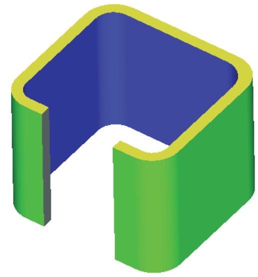
Pre-highlighting is turned on in the 3D window. When the cursor approaches an element on the 3D scene, the element is getting highlighted, and the cursor gets an attached glyph that indicates the type of the highlighted element. If the cursor is briefly held over the element, a pop-up help appears displaying the type of the element. To select the element, press. To select a group of elements, hold the ***<Shift>*** key while selecting.





In complicated models, several elements of the same or different types can be under the cursor simultaneously. In this case, use the “Other…” button of the context menu that provides selection from list. The list is composed of the elements nearest to the cursor, whose types are admitted by the selection filter. An alternative way is to point the cursor at the desired element and briefly hold still. The cursor will be appended a “scroll list” glyph, and a pop-up widget will appear indicating the total number of elements available for selection at this point, and a current element type. The widget allows scrolling through the elements with the mouse middle wheel button (if equipped). Pressing selects the currently highlighted object.

To select a 3D profile, point the cursor at the contour line. To select bodies, set up the selector for operation selection and turn off edge and face selection. If in “Shading” or “Render” display mode, point anywhere on the body. In “Wireframe” mode, point at an edge of the body. Since the bodies may overlap on the scene, re-orient the model to bring the desired body up front.



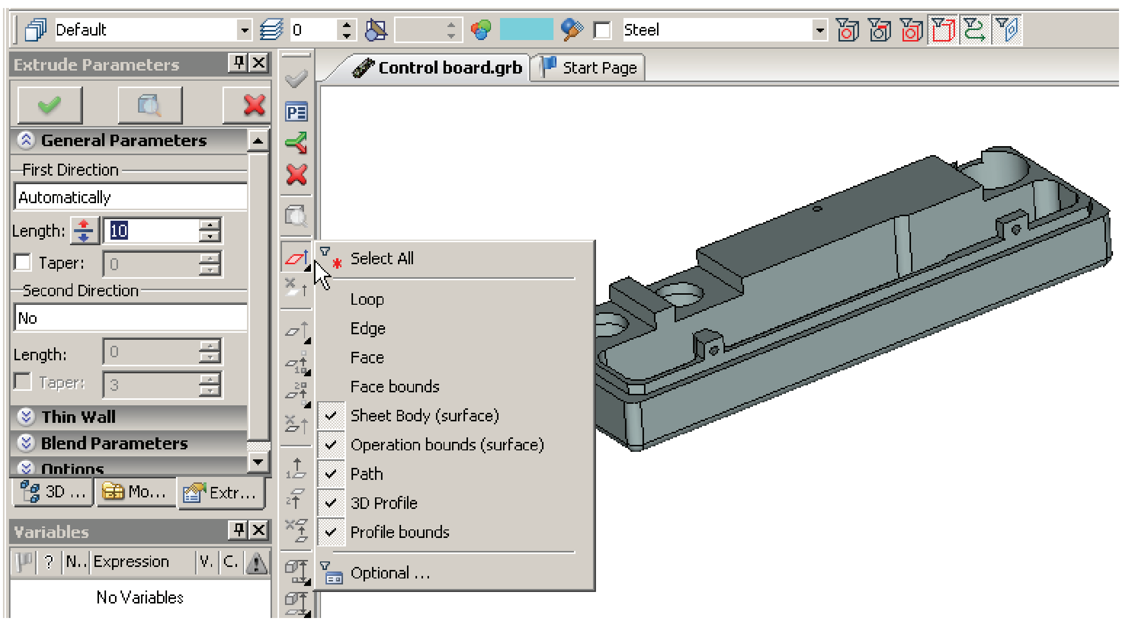
The selected elements are marked with a color, depending on the object type and the purpose of selection. All color settings are defined in the system customization functionality.

In some 3D commands, different elements of the same type are marked with different colors. For example, in the three-face blending operation, the user specifies the left, right and middle sets of faces. In this case, the faces of each set are marked with different colors. These colors also mark the respective tabs of the property window dialog box, that contain the corresponding lists of faces.

Selecting elements within a command

The user often needs to select some existing elements on the 3D scene when defining a command input data. The system automatically adjusts the selector for the element type required by the current step in the command automenu. Suppose, for instance, while defining a blend we pushed the option “Select edge”. The selector then adjusts to edge selection only. Next, reorient the scene for easy selection, point the cursor at the desired element, and press . In this case, selecting a group of elements is done by picking subsequently, without holding the ***<Shift>*** key.

Often, a required geometrical input in a command may be defined by a number of different-type 3D objects. Thus, for instance, a direction can be defined by wire objects, such as edges, paths, profiles, etc., or by a pair of 3D nodes, or by a normal to a planar surface, etc. In this case, the automenu option provides an extended list of filter settings for the selector. The availability of the list is indicated by a black triangle in the right-bottom corner of the option button. A simplified representation of the list is also displayed in the system bar.



Should a wrong element get selected (say, a neighboring one), use the automenu option,

	<I>	Select Other Element

A subsequent pick on the icon switches the selection to the next nearest element, on and on until all elements are traversed. This option is available with all 3D commands that require object selection.

Canceling selection

While within a command, element selection can be cancelled by pressing or ***<Esc>*** key. The elements are getting unselected one by one in the reversed order of the selection. Many commands allow unselecting all elements at once by using an appropriate automenu icon. (Refer to the respective topics in the documentation.)

To cancel selection of an element or a group of elements while outside any command, simply select another element or click at a blank on the 3D scene.

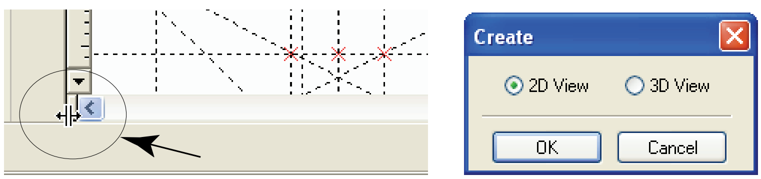
Element Search

Search functionality can be invoked in transparent mode, and is available with all commands. For quick access to the search command, the button found in the set "Edit" of the main toolbar can be used.

Advanced search can be performed by the command **"Edit|Find… "**.

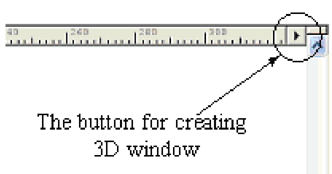
Opening New Windows

When creating a new document from a 3D model prototype, a 3D window opens first. Later in design, the user may need to open additional 2D or 3D windows. The window can be split vertically or horizontally by dragging the separator bars that originally appear as notches at the top of the vertical scrollbar and at the left of the horizontal one. Point the cursor at a notch so that it changes to juxtaposed arrows, and drag with to the desired location of the split. Then release the mouse button, select the desired type of the new window in the appearing dialog box, and press **[OK]**.



Once the window is divided in two, the notch turns into the separator that can be further dragged to adjust the windows division.

If the original file was created from a drawing prototype, the 2D window was opened first. A simple way to create another 3D window is to click on the button with a triangular arrow in the right-top corner of the graphic area of the current window.



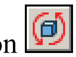
As a result, the graphic area of the current window will split in two equal parts, a 2D window on the left, and a 3D one on the right.

Finally, one can call the command for opening a new window, **"WO:Open New Window"** by typing on the keyboard or via the textual menu **"Window|New Window"**.

> Depending on what kind of window is currently active, some or other commands will not be accessible. Thus, if a 2D window is active then the 3D Visualization commands won't be accessible, and neither will be 3D view controls. Vise versa, while in a 3D window, the most of 2D commands could only be accessed in a special mode - when a workplane is activated.

Manipulating Model in 3D Window

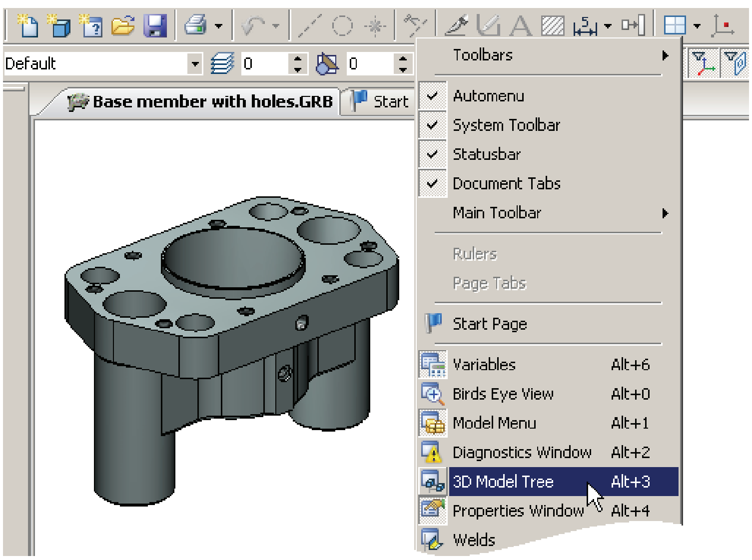
While working in 3D window a model can be spun in any way, zoomed at in and out, and panned. These actions can be done at any moment using mouse or keyboard input. When using the mouse, sometimes it is necessary to set certain options on the "View" toolbar (located at the right side of the screen).

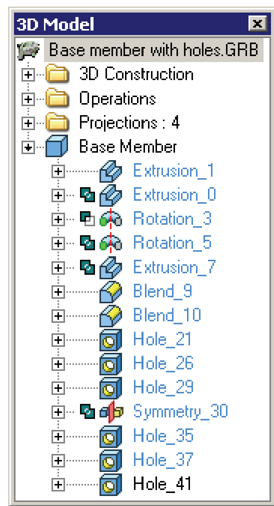
To spin the 3D scene while working on an active workplane, use the button  found on the main toolbar in the mode "Workplane" or "Workplane (Sketch)". It is also possible to spin the model with the help of the mouse simultaneously pressing the ***<Alt>*** key.

For more details see the chapter "Working with the 3D View Window".

"Model" Window

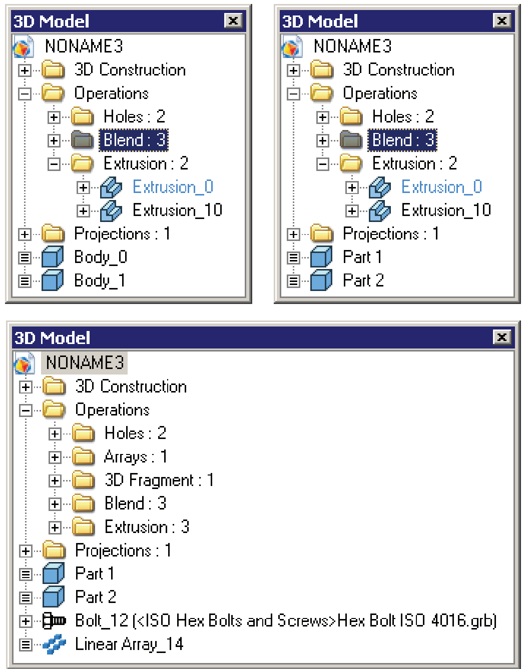
To open the 3D model tree window use the command **"Customize|Tool Windows|3D Model Tree Window"**. Alternatively, press the right mouse button while in the command area of the application window, as, for instance, over a toolbar. This will bring up the list of tool windows. Select "3D Model Tree".





The window titled "3D Model" will appear at the left side of the screen. This tool window reflects the structure of the 3D model as a tree.

Bodies are put at the root of the model tree. Depending on the Body geometry type (solid or sheet object), the respective icon will be used for it in the model tree.



Each model Body is assigned a unique name, by default composed of the word “Body” and a number, for example, “Body_0”. If desired, any Body can be assigned an arbitrary individual name.

The 3D Fragment, Part and 3D Array creation operations can be displayed at the top level of the model structure, along with Bodies.

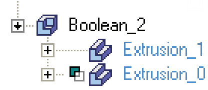
The Body knots are marked by the glyph ⊟. It allows watching the history of the given body’s creation.

Point the mouse at the glyph ⊟ and click . As the model tree branch with the Body history expands, the glyph changes to ⊞. The Body history consists of a sequence of their defining operations, displayed as a list. The list is formed from top down in the order of creating or using operations. To hide the Body history, point to the glyph ⊞ and click .

Each model element entered in the Body history has a unique name. By default, the name consists of the name of the element type and a number, for example, “Extrusion_6”. If desired, the element, just like a Body, can be assigned an arbitrary individual name. The “plus” glyph before the element means this element is based on other elements. To expand this model tree branch and view the element’s parents, point the mouse to the glyph ⊞ and click . After that, the glyph will change to ⊟. If there is no glyph before the element, this means the element is the last one on this branch of the 3D model tree.

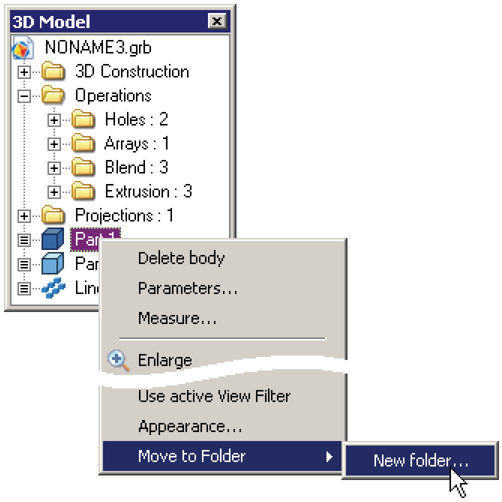
If the creation history can be built for an operation, then the history will be displayed instead of the parents. In this way, the ⊟ glyph is put before the operation name instead of the ⊞ glyph. An operation's history is displayed in the same way as a Body’s history.

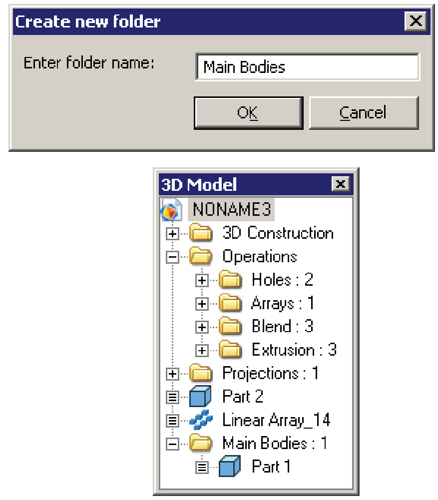
The Boolean operations are displayed in a special way in the history. The type of Boolean (addition, subtraction or intersection) is drawn before the icon of the operation that comes as the second operand of the Boolean. The diagram shows the history of the operation “Boolean_2”: the body “Extrusion_1” is subtracted from the body “Extrusion_0”.



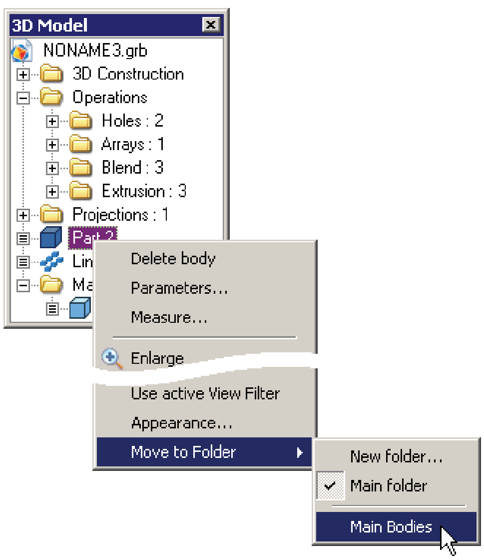
In addition to the list of Bodies, in the root of the model tree there are also special branches of the tree in which all 3D elements created in the current model are enumerated. Auxiliary 3D elements are put on the branch “3D Construction”. All operations are on the second branch, “Operations”. 3D annotation (3D dimensions, 3D leader notes, etc.) are put into the branch “3D service elements”. The “Constraints” branch lists all constraints created in the given model. All elements are sorted into the folders by types: 3D Nodes, Workplanes, 3D Profiles, etc.; within the folders, the elements are sorted alphabetically. The number next to the folder name and the colon means the number of elements of this type contained in the model. By expanding branches of the model tree, you can access any element of the 3D model.

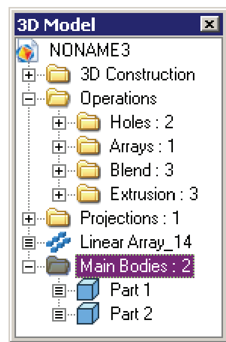
Separate branches-folders can be also created for grouping Bodies (and also 3D fragments and 3D arrays). To do it, in the context menu for the Body which you want to put into a separate folder, invoke the command **“Move to Folder|New folder…”**. A window for specifying the name of the folder being created will appear on the screen. After specifying the name and pressing **[OK]**, there will be a folder having the specified name in the tree of the 3D model, and the selected Body will be put into this folder.



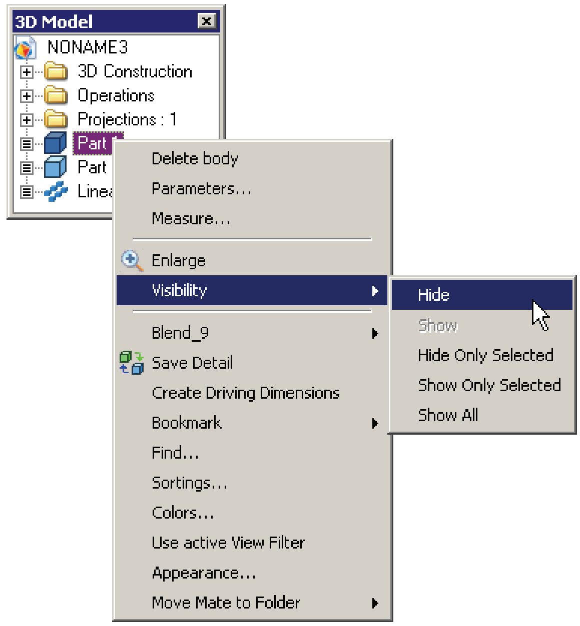


If in the tree of the 3D model at least one Body folder has been already created, in the context menu for each body in the submenu **"Move to Folder"**, a list of existing Body folders will be also present. The folder that contains the current body will be marked. For moving Body into one of the already existing folders, it is enough to select the folder in the list.





It is also possible to move Bodies from the root of the model tree to the folder and backwards just by dragging Bodies while pressing .



Selecting objects with the help of the model tree is sometimes very convenient in many commands, for example, when selecting this element in the 3D window is difficult for some reasons.

By pointing to an element in the model tree, you can access the context menu (by pressing ). Besides the common commands for managing the selected element, the context menu contains some additional commands for working with particular Bodies, operations and the model tree as a whole:

The **"Visibility"** group of commands unites the commands to control the visibility of Bodies and 3D construction elements:

Hide. This command makes the selected 3D objects invisible in the 3D scene;

Show. This command cancels the previous command, making the selected 3D objects visible in the 3D scene;

Hide Only Selected. This command makes all selected 3D objects invisible, and shows all the rest, if those were previously hidden.

Show Only Selected. This command makes all the selected 3D objects visible, and hides (makes invisible) the rest;

Show All. This command makes all objects in the 3D scene visible.

The accessibility of the commands in the group depends on which 3D objects are selected. For example, if all selected 3D objects are currently hidden, then the "Hide" command will be inaccessible.

When different-type 3D objects are selected (for example, 3D profiles, 3D paths and Bodies), then the commands "Hide Only Selected" and "Show Only Selected" bring up an additional dialog to choose one of the two ways to proceed with those commands:

- Apply this command for all model Elements – if this option is selected, then all 3D objects of the 3D model will be hidden/shown, except those selected, regardless of their type;
- Apply this command only for Elements of selected types - if this option is selected, then only those 3D objects of the 3D model will be hidden/shown that belong to the same types as selected (except the selected ones themselves).

> The hidden/shown Body state can be defined using the "Hide" option in the Body's parameters dialog without the use of the described commands. This option's value can be defined via a variable. If a variable is used, then the commands of the "Visibility" group will not affect the visibility of such Body. This will be stated in the provided warning message.

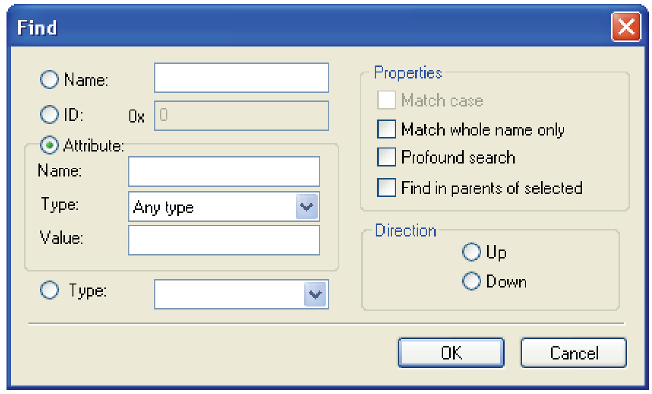
Delete Body (for Bodies). This command serves to delete all the operations which are part of this Body's creation history. In addition, all parent 3D construction elements of this Body can be deleted, or just those parent elements that are not referenced by other created Bodies.

Delete (for operations and auxiliary 3D elements). The command deletes the selected operation or 3D construction element.

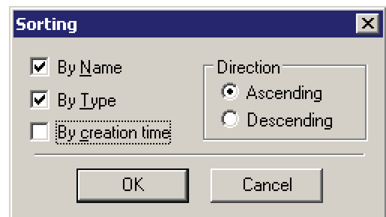
Suppress (for operations). This command excludes the operation from regeneration. This allows user to temporarily remove 3D model elements. To select a suppressed operation, use the “3D Model” window or do the element search.

Bookmark. A bookmark can be assigned to any element for quick navigation through the model tree. The bookmarked element gets a blue triangle mark next to the element type glyph. To set or unset a bookmark, use the context menu items **“Bookmark|Insert”** and **“Bookmark|Remove”**. Once set, one can navigate from one to another bookmark, using the context menu **“Bookmark|Go Next”** and **“Bookmark|Go Prev”**.

Find. An element can be searched within the model tree. The search is performed only through the branches that were expanded at least once in the current session. The search parameters are input in the dialog box invoked by the command.



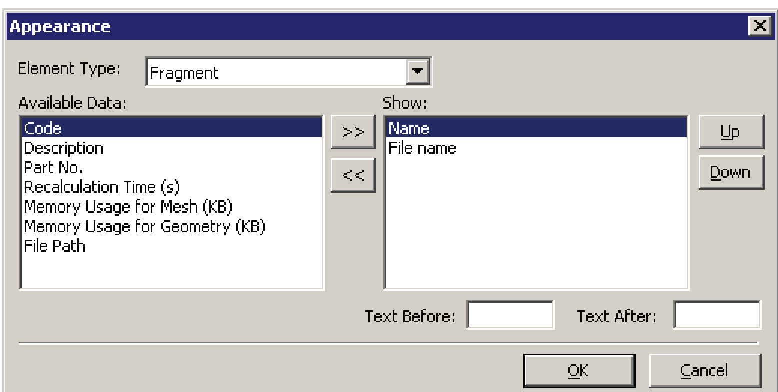
Sortings. This command is used for setting up the sorting parameters. Sorting is performed on the portions of the model tree where it makes sense, as, for instance, at the top level among the model operations if several of them are listed. Sorting can be done by element type, name, or creation time.



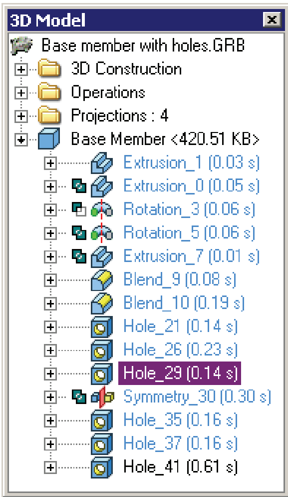
Colors. The element names are displayed in one of three colors. The elements differ by their visibility in 3D window. An element name is highlighted in the tree when pointed at by the cursor. Simultaneously, the corresponding element is highlighted in the 3D window. This color is also used for highlighting the selected object.

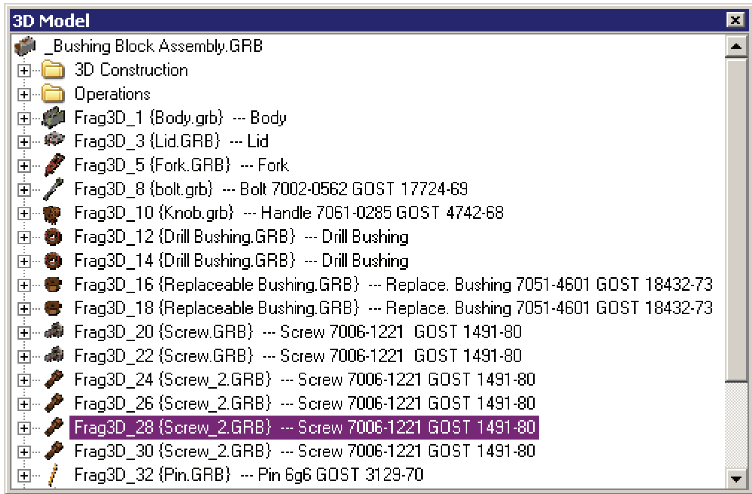
Use active View Filter. The 3D window selector can be adjusted for selecting elements in the model tree. To activate the filter, flag this item in the context menu.

Appearance… This command allows a user to customize information which is displayed in the tree of the 3D model. Upon calling this command, the settings dialog window appears.



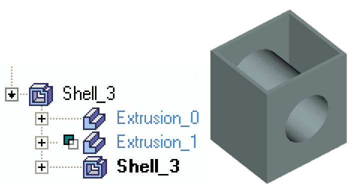
In the dialog “Appearance”, it is possible to select items which will be displayed in the tree for objects of various types: time spent by a computer on object recalculation, the memory size of the geometric data for the object and the size of the data required for its displaying (mesh), etc. This customization allows a user not only to see the objects in the tree in the convenient for the user form, but also carry out the recalculation time and the memory usage optimization for the model. The information about the name and description of the assembly components can be also convenient when working with the assembly models. This information can be displayed together or instead of the filename and the path of the file for the assembly components (fragments).

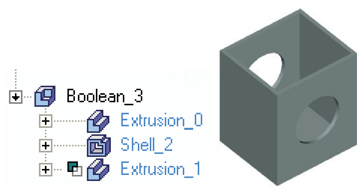




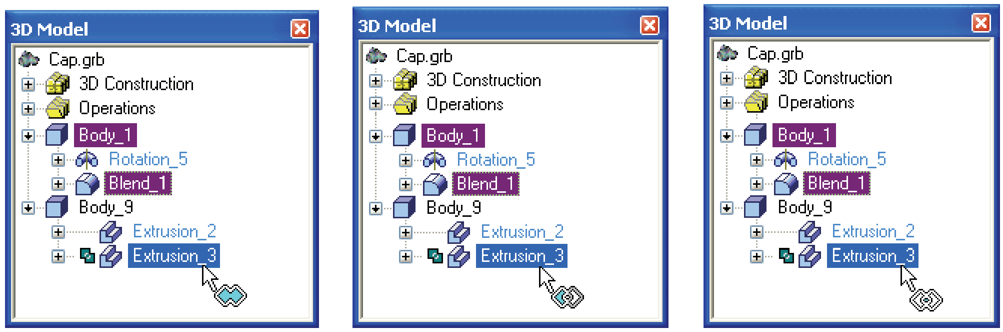
Settings specified in the dialog “Appearance” are saved in the document file and used upon its subsequent opening.

In certain cases, operations can be reordered by moving their labels within the model tree. This effectively changes the model structure and geometry. Press on the element label and drag it to a new location in the tree. The following diagrams demonstrate the result of reordering an element in the model tree. In the beginning, the operation Shell_3 followed the subtraction operation Boolean_2. Then, the Boolean, together with its tool body operand Extrusion_1 (a cylinder), was reordered to be after the shell operation.



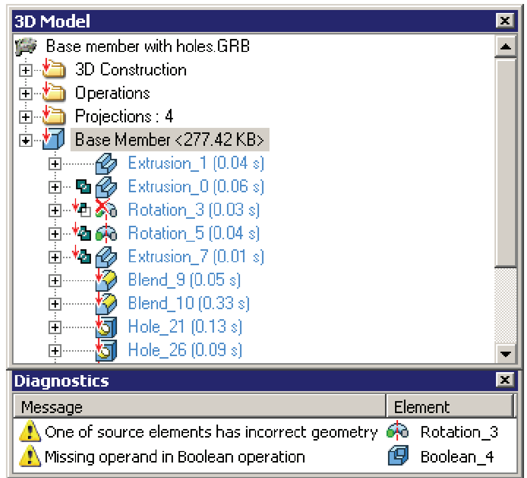


The model tree helps quickly create Boolean operations of all types. If you select a Body and drag it over another Body, the indicator of the Boolean “addition” will appear at next to the pointer. If you additionally press the key ***<Ctrl>*** or ***<Shift>***, the type of the Boolean changes to subtraction or intersection, respectively. Instead of Bodies, you can select operations that are last in their histories.



"Diagnostics" Window

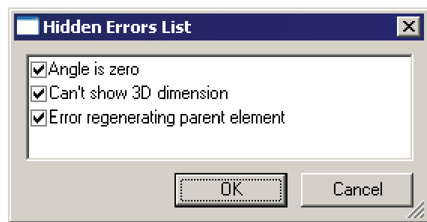
This tool window is used for displaying warning and error messages. The errors may occur due to invalid operation parameters input, an out-of-range variable value, regeneration with incompatible parameters, etc. The system attempts to identify the reason of a failure and outputs a message in this window. Besides the failure reasons, the system outputs the information about the failing element. The failing element and its dependents are marked with a red cross in the model tree.



The context menu in the diagnostics window provides, along with the standard set, some special commands, as:

Hide this message. This command serves to hide messages of the selected type. When this command is applied, the messages of the selected type will no longer be displayed in the diagnostics window up until this restriction is canceled. The command also affects the future sessions of working with the system.

Hidden errors list. This command calls the dialog window with the list of hidden messages. To cancel the restriction on the messages of the given type, simply clear the check before a message.



Show window automatically. This option makes the window appear automatically whenever new messages are generated. It is useful in the case when several tool windows are grouped together on one console.

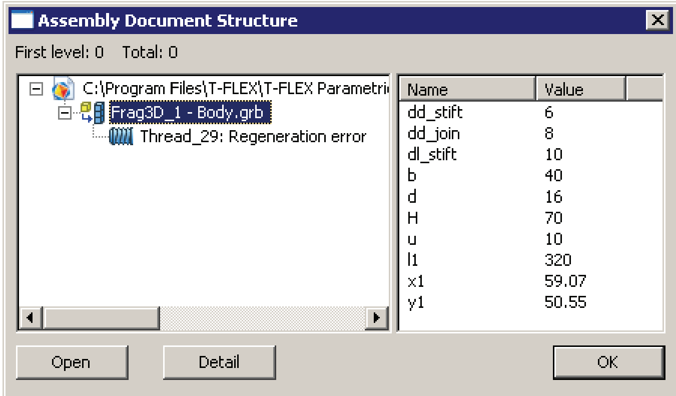
Clear window. This command clears the diagnostics window from accumulated messages.

Hide window. This command removes the diagnostics window. To bring it back, use the main menu command **"Customize|Tool Windows|Diagnostics Window"**.

Edit. This command is accessible only when a specific message is selected. It invokes editing of the failing element or operation.

If the errors occurred within a fragment file then the diagnostics window references the operations from the fragment model that caused the error, while the context menu gains another command,

Show Fragment Structure. This command invokes the "Model structure" dialog box.



The fragment in which the error was identified is automatically highlighted. The right half of the window contains the information about the fragment variables: their names, values, and comments. To handle the error, one can open the fragment file by pressing the **[Open]** button.

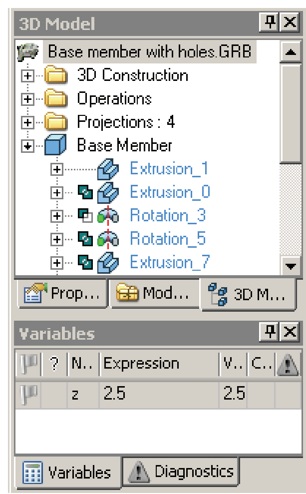
An error in the fragment model may be caused by an invalid value of a fragment variable. To open the fragment with the current values of the variables use the button **[Detail]**.

You can select multiple messages in the diagnostics window by using the combination ***<Ctrl>***+. When several messages are selected, the following command will be available in the context menu:

Delete Related Elements. This command serves to delete all 2D and 3D elements related to the selected messages.

Arranging Tool Windows

All tool windows in the T-FLEX CAD application can be combined in a common console with tabs. This saves up the working area on screen and eases the user from constantly adjusting the windows borders while working with the application. To combine two windows, pick one of them with at the title area and drag into the title area of the other window. The two windows collapse into the common console. To switch between the windows, use the tabs that appear in the bottom of the console. The system may automatically switch between the windows during operation.



Toolbars

Toolbars provide additional convenience to T-FLEX CAD 3D operation. Working with toolbars, their customization and access to toolbars are done in the same way as in the 2D drawing mode. Further refer to the volume “Two-dimensional design”, the chapter “Main Concepts of System Operation”.

The necessary options of the command are set using the **automenu** together with the **property window**, similar to the 2D mode. The set of icons in the automenu is specific to each command. However, some options in the automenu are common for all commands:

	<Y>	Finish input. Confirms element creation
	<P>	Set entity parameters
	<I>	Select other element
	<Esc>	Cancel selection
	<F4>	Execute Edit Command
	<X>	Exit command
	<F5>	Preview Operation Result

Customization

Certain customization is necessary for working with T-FLEX CAD. Customization settings include both the system-wide ones, and those specific to a particular document. The system settings are automatically saved in the Windows registry and are applied at subsequent starts of T-FLEX CAD. The settings specific to a particular document are saved with the document. The system customization is done via the "**Options**" dialog box, invoked by the command:

Keyboard	Textual Menu	Icon
<SO>	**"Customize|Options…"**	

All system settings related to 3D modeling are located on the "**3D**" tab. For more details on working with "Options" dialog refer to the volume "Two-dimensional design", the chapter "Customizing System".

Customization of document parameters is done via the **"Model Status"** dialog box invoked by the command:

Keyboard	Textual Menu	Icon
<ST>	**"Customize|Status…"**	

All drawing settings related to 3D modeling are located on the "**3D**" tab. For more details on working with "Model Status" dialog refer to the volume "Two-dimensional design", the chapter "Customizing Drawing".

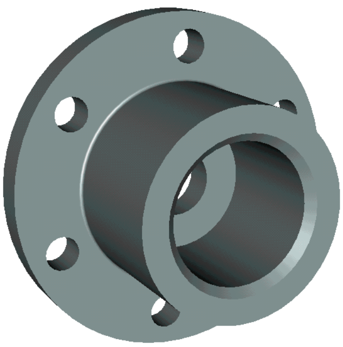
BRIEF INTRODUCTORY COURSE IN 3D MODELING

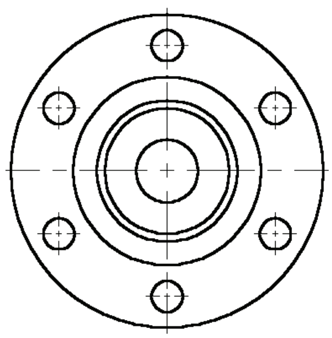
In this chapter, you will learn how to create a three-dimensional model using simple examples. The manual describes all necessary steps in the modeling process. Further, you will learn the basic commands and principles of creating a 3D model.

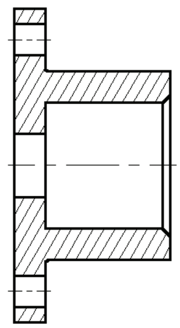
T-FLEX CAD system allows various approaches to creating a 3D model. The main method is creating most of constructions directly in the 3D window. In another approach, a 3D model is created based on already prepared 2D drawings or auxiliary 2D constructions. This construction may further be used for generating drawings by projecting the model in all necessary views and making cuts and sections. These 2D objects can further be furnished with necessary dimensions and other drawing attributes.

Main Approach to 3D Model Creation

As mentioned above, this approach makes possible creating a three-dimensional model without using the 2D window. The diagram below shows a sample part that we will be modeling. We will create the three-dimensional model first, and then automatically generate views and a section.







> The model file is located in the library "`Documentation samples`", the folder "`3D Modeling\Brief introductory course\Detail 1.grb`".

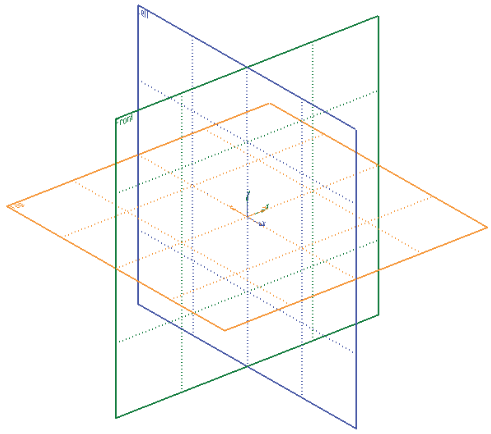
The model creation will be done in several steps. First, one has to create initial auxiliary elements. Based on those, you will be able to create a first draft of the considered part – without the holes and chamfer. To do this, use the "Rotation" operation. At the next step, add six holes in the part's body. Holes can be created by different means. We will discuss several methods here to give you a more complete idea about 3D model creation methods. Then, to get the final version of the part, you will just need to apply a chamfer via the "Blend" command.

Creating Auxiliary Elements

Let's start from scratch. Create new document.

> To begin modeling, one can use an appropriate prototype file among available in the T-FLEX CAD system (**"File|New 3D Model"**).

You have just created a new document that already has three standard workplanes – Front, Right and Bottom. You can see that the 3D window with these elements opens automatically within the system window.



> For convenience of modeling in 3D window, spinning, panning and zooming of the scene view are available at all times, even when commands are active. Note: the view manipulations while in certain commands require a specific option to be set. These commands will be described below.
>
> To spin the scene press and hold and move the cursor as desired. One can also use the two arrow key pairs and the pair ***<Page Up>*** and ***<Page Down>*** for spinning around the three respective axes.
>
> Zooming can be done at any moment by scrolling the mouse wheel button (as in IntelliMouse design) or via the provided commands on the "View" toolbar located along the right border of the window. Panning and zooming of the view can also be controlled via ***<Ctrl>***+ and ***<Shift>***+ combinations respectively.

As we move the cursor close to a workplane, the latter gets highlighted. T-FLEX CAD provides pre-highlighting on cursor over for all elements in the 3D window depending on the selection filter settings. The selection filter icons are located on the system toolbar. To pick an element among the icons, simply press .

Let's select the "Right" workplane. One can see now that 2D drawing commands become accessible. We will further use these commands for creating auxiliary elements in the 3D window.

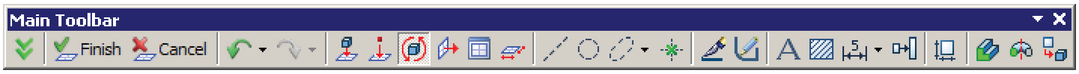
What auxiliary elements do we need? The first body to be created is a body of revolution. To create it, we will need the contour and the axis of rotation for this contour.

> Drawing in 3D window can be performed using any of the tools available for 2D drawing. Thus, one can use the sketching environment for rapid creation of non-parametric models. Accordingly, parametric modeling approach requires creation of construction lines first, then application of graphic lines. The system automatically creates 3D profiles based on drawn graphic lines, to be later used in 3D operations.

To start drawing, call the command,

Keyboard	Textual Menu	Icon
<L>	**"Construct|Line"**	

After activating the workplane by any 2D drawing command, the main toolbar is switched to the mode "Workplane". In this mode a set of icons for working on the active workplane is displayed on the toolbar.



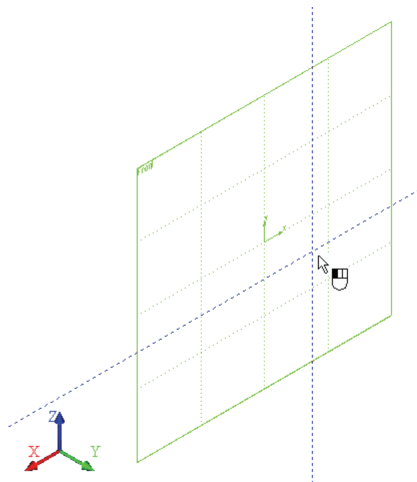
For convenience, the active workplane is reoriented parallel to the screen. However, it can be spun in any desired way, after setting the option .

It is also possible to open the 2D window and continue drawing in 2D mode. Once the 2D window is closed, all changes will be reflected on the 3D scene. The 2D window can be opened or closed using the icon.

Let's construct two base lines, a vertical and a horizontal one. To do so, push the following button in the automenu:

	<X>	Create two crossing Lines and Node

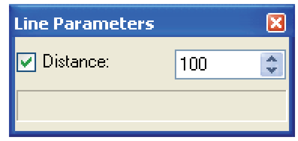
Point cursor at the bottom-right portion of the workplane and click .



Thus, we have created two orthogonal lines and a node. These will be used as a reference for all further construction. Press to quit the last active command. We are now in the parallel line creation mode. (This mode is set by default in the command **"L: Construct Line"**).

> Just like in 2D drawing, we first need to create a framework of thin lines, and then apply graphic lines along the intended segments.

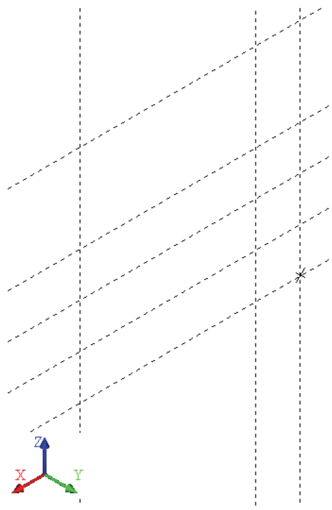
To construct a parallel line, select a reference line first, that will be used as the "parallel" reference for the new one. Similarly to 2D operation, the 3D drawing mode also supports object snapping. Thus, to select a line, move the cursor to the vertical line. The cursor will gain a glyph like this, . Click and move the cursor leftwards. One can see the cursor changing appearance again – it is now rubberbanding a line parallel to the selected.



To fix the line at an arbitrary location, just click . However, we would like to specify the distance of 100 mm, by typing it in the property window. (In future, this parameter value can be changed at any time as desired.) Press ***<Enter>*** to complete the line creation.

We have just created a line parallel to the selected, at the distance of 100 mm from it. As one can see, rubberbanding resumes at this point. The system is still in the mode of parallel line creation with respect to the original selection. Type a new value of 20 mm right away, followed by ***<Enter>***, thus creating another line. That's all with the parallel line creation for now. Click or ***<Esc>*** key for quitting the parallel line creation mode.

Next, create four more lines in the similar way, parallel to the horizontal base line. Use the distances equal to 20, 40, 60 and 100 mm respectively. The result should look like shown on the following diagram.

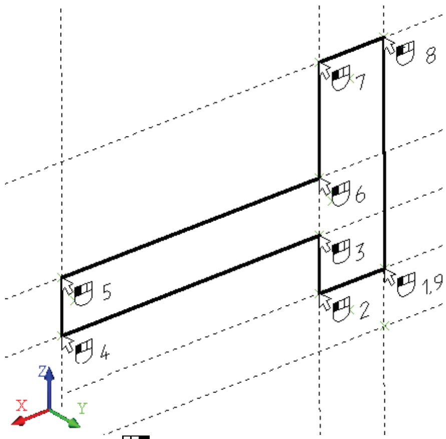


Now, apply graphic lines along the construction lines appropriately. Call the graphic line creation command:

Keyboard	Textual Menu	Icon
<G>	**"Draw|Graphic Line"**	

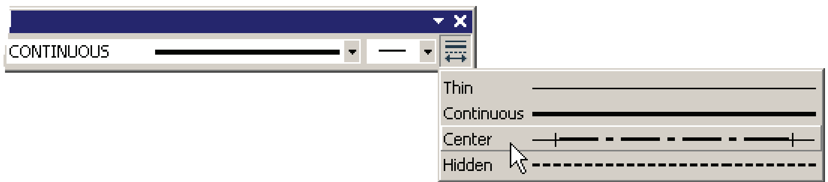
Graphic lines snap to the construction entities – lines, circles, nodes, etc., as well as to construction line intersections. In the latter case, a 2D node is automatically created at intersections snapped to by graphic lines. Straight graphic line segments maintain snapping at their end points. To have an end of a graphic line snapped to some entity, move the cursor to the intended location. Let the cursor snap as indicated by an appropriate glyph, and click .

In this example, we use snapping to construction line intersections and to 2D nodes. Draw the profile as shown on the following diagram. To do so, click with at the appropriate points in the order marked on the diagram.

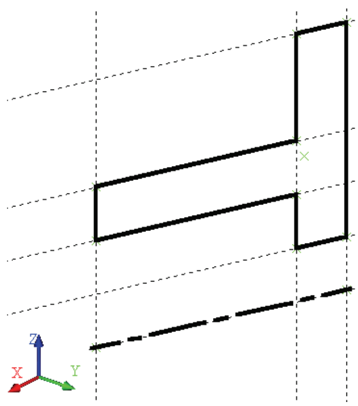


To quit continuous line creation mode, click . Note that the graphic line creation command is still active.

Change the line type in order to draw the axis. To do so, pick the icon  on the system toolbar and select CENTER type from the pull-down list.

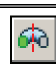


Draw the centerline as shown on the following diagram.

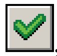


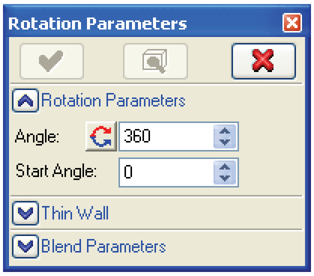
Creating Rotation Operation

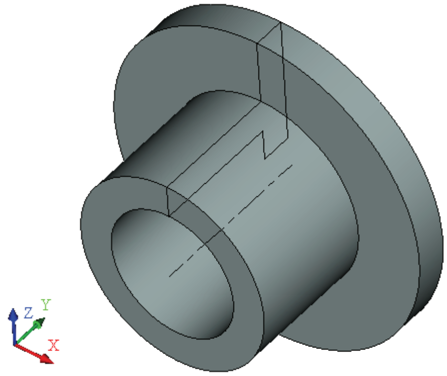
One need not quit drawing explicitly in order to create a body of revolution. Simply call the "Rotation" command as follows:

Keyboard	Textual Menu	Icon
<3RO>	**"Operation|Rotation"**	

The system automatically determines the rotation axis and the contour among the created lines to be used as a 3D profile. You can preview the result as a wireframe model. In our case, the rotation angle is 360 degrees.

Note that the value of 360 degrees for the rotation angle is set by default in the property window. Just confirm the operation by pushing the check button , and the body of revolution is created.





Creating Holes

Next, we need to cut six holes in the part.

Holes can be created by different methods. The fastest and simplest one is using the dedicated “Hole” operation. The latter allows creating holes of standard shapes in solids, using templates provided in the T-FLEX CAD utility library. In this way, the user only needs to specify the position of the would-be hole within the solid and specify its type and dimensions.

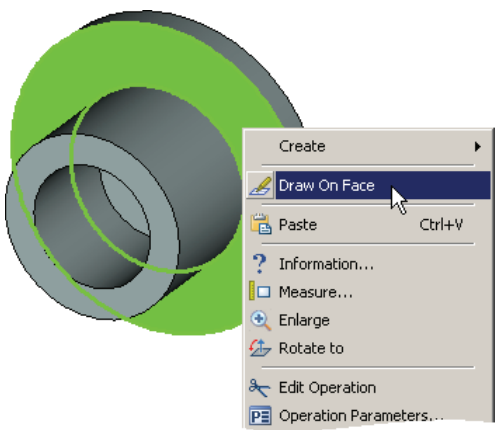
Non-standard holes and slots can also be created without using the dedicated operation. To do this, the user should create an additional solid representing the internal volume of the hole, and then “subtract” it from the main body’s volume by a Boolean operation.

Let us consider both techniques.

Creating holes using a dedicated command

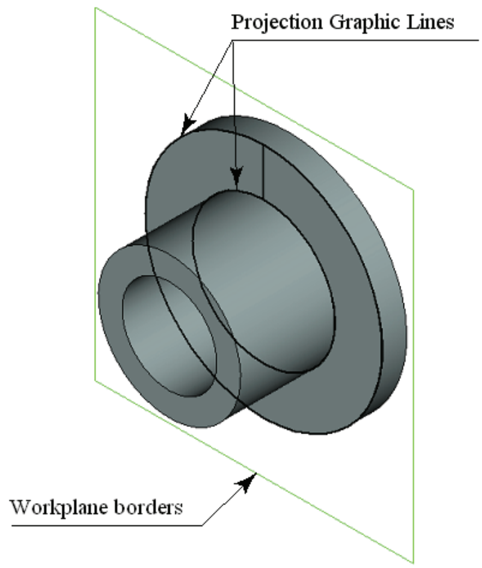
First of all, we will create 3D nodes corresponding to the centers of the would-be holes, on one of the part’s faces. To create them, we will again use the 2D drawing mode for creating auxiliary 2D nodes on the respective face of the part.

To select a face, move the cursor to the desired element of the model so that it pre-highlights. At this point, press  to invoke the context menu and select “Draw On Face” item in it (see the diagram).



> If the desired element is not getting pre-highlighted, make sure the selector is set up for this element type. To change selector's settings, you can use selector filter icons on the system panel or select a combination of types from the list.

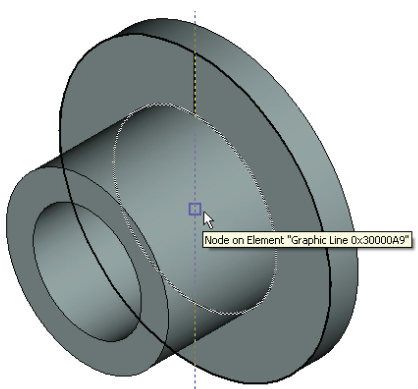
The "Draw On Face" command creates a new workplane based on the selected flat face. The selected face is automatically projected on this plane, and the mode of drawing in the 3D window activates. Further construction can use snapping to elements of the face projection.

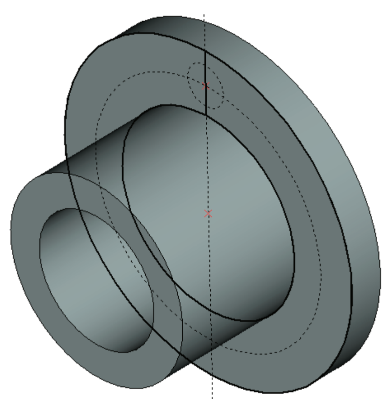


Call the command **"L: Construct Line"** again. Select the following option in the automenu,

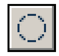
	<V>	Create vertical Line

Move the pointer to the center of the circle - the circle center will be pre-highlighted. It is available for snapping to the vertical line. Click . The constructed line will be snapped to the 2D node which was automatically created at the circle's center.



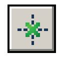


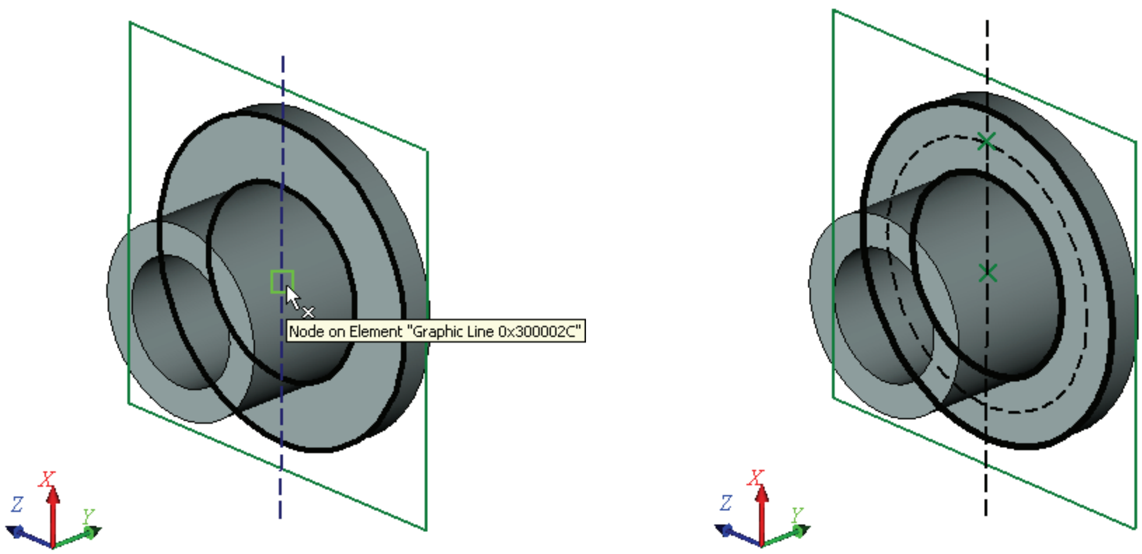
Call the command:

Keyboard	Textual Menu	Icon
<C>	**"Construct|Circle"**	

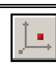
Point the cursor at the center node to select it as the center of the new circle. Specify the radius of the circle equal to 80 mm by typing in the property window.

Create a 2D node at the intersection of the vertical line and the new circle using the command:

Keyboard	Textual Menu	Icon
<N>	**"Construct|Node"**	



Now, one can construct a 3D node based on the created 2D node. To do this, without leaving the drawing on face mode, call the command:

Keyboard	Textual Menu	Icon
<3N>	**"Construct|3D Node"**	

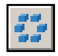
When the 3D node creation command starts, move the pointer to the created 2D node and click . The 2D node will be highlighted, and the option  will become accessible in the automenu. Click , and the 3D node will be created. It will be located in the plane of the selected face, while the 2D node will be its projection on that face.

The created 3D node will define the center of one of the six holes. There are two ways to define centers of other holes:

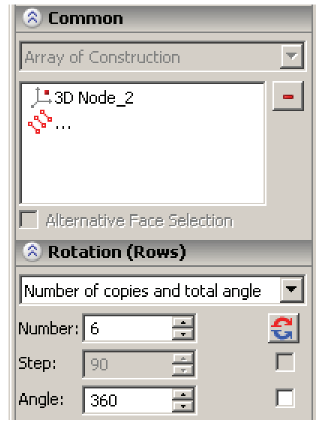
1. Construct five more 2D nodes on the same workplane and create the required 3D nodes based on the 2D ones (similar to the first 3D node creation);

2. Create the required 3D nodes by means of a 3D array based on the first 3D node.

The second method is faster, so we will use it in this case. Call the command **"3AR: Create Circular Array"**:

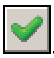
Keyboard	Textual Menu	Icon
<3AR>	**"Operation|Array|Circular"**	

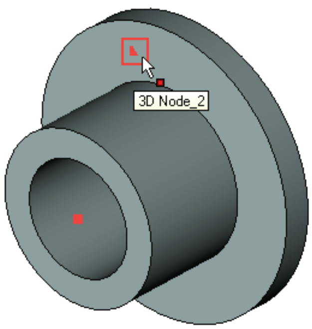
> You do not have to explicitly finish the draw on face moden - it will terminate automatically upon launching the 3D array creation command.



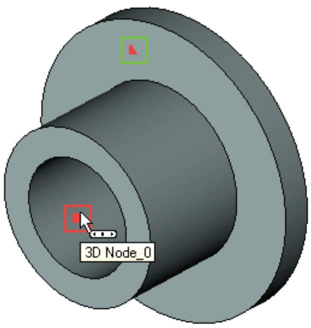
In the command's properties window, set the array type as "Array of Construction". Then, in the 3D window, move the pointer to the created 3D node (the latter shall highlight) and click  to select it. If all was done right, the name of the 3D node selected for copying shall appear in the properties window.

After that, you need to specify the rotation axis of this circular array. To do this, you can use a pair of 3D nodes automatically created when defining the axis for the rotation operation. Select these two 3D nodes one by one.

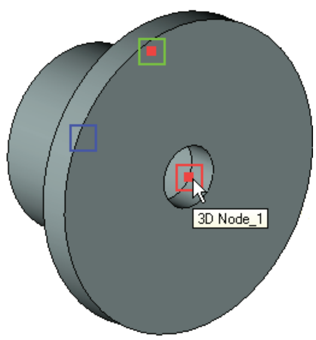
Next, let's specify what array parameters will be used ("Number of copies and total angle"), in the "Rotation (Rows)" section of the properties window, and enter the required values of the parameters (the number of copies – 6, the total angle – 360^0). To complete creation of this circular array of 3D nodes, simply click .



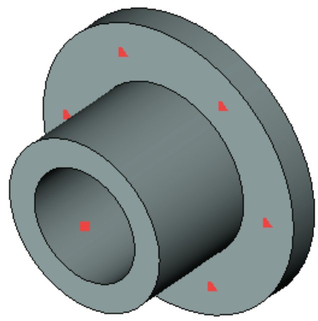
Selecting 3D node to copy



Selecting first 3D node of the array's axis of rotation



Selecting second 3D node of the array's axis of rotation



Result of creating an array of 3D nodes

After creating the array of 3D nodes, you can call the command **"3H: Create Hole"**:

Keyboard	Textual Menu	Icon
<3H>	**"Operation|Hole"**	

Upon entering the command, you need to activate the option:

	<M>	Add Hole Array

After that, move the pointer to one of the 3D nodes of the circular array and click . In the command's properties window, select the hole type – "Hole for fasteners". The preview image of the holes being created will appear in the 3D window.

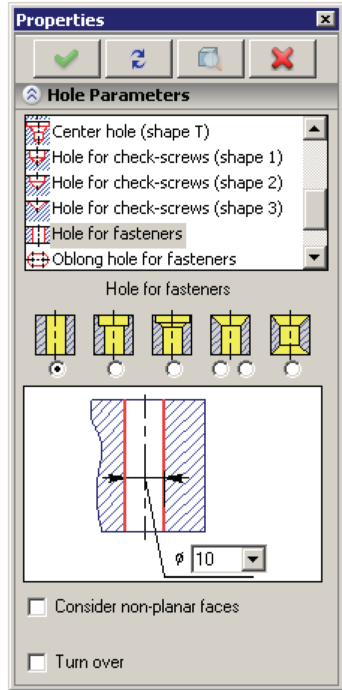
At the bottom of the properties window there is a pane with a schematic image of the hole of the selected type and fields for entering the hole parameters. Set the hole diameter to be 20 mm.

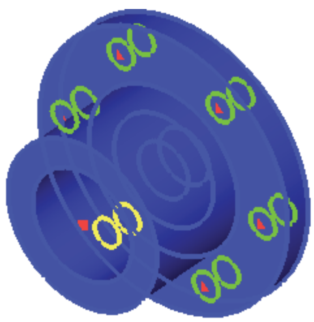
The following option will be automatically activated in the command's automenu for the holes of this type:

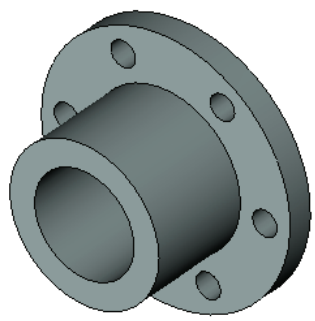
	<F>	Through all

When this option is set, the depth of the holes will be defined automatically by the part's thickness.

To complete creation of holes, simply click in the properties window or in the command's automenu.







Creating holes without using a special command

In this approach, initial constructions of the holes creation will be same as in the previous method.

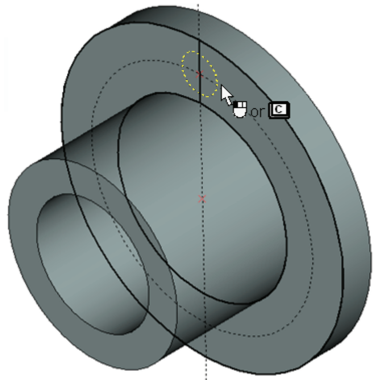
You select a part's face and call the context menu command «Draw On Face». Call the command **"L: Construct Line"**. Select the automenu option . Construct a vertical line passing through the center of the circle (of the face's projection). Then, call the command "C: Construct Circle". Create the circle with the radius equal to 80 and the center at the central node. Similarly, create another circle with the radius equal to 10 mm at the intersection of the previous circle's circumferential and the vertical line.

Then draw graphic line over the newly created circle. To do so, call the graphic line creation command:

Keyboard	Textual Menu	Icon
<G>	**"Draw|Graphic Line"**	

Pick the icon on the system toolbar, and select CONTINUOUS type from the pull-down list.

Move the cursor to the just drawn circle. The circle will get highlighted, and the cursor gain a circle glyph. Once so, click, that applies graphics over the whole circle. On a crowded drawing with multiple elements near the cursor, snapping may occur to an unintended element. In such cases, the circle can be explicitly selected by typing ***<C>*** key. This will select the circle nearest to the cursor.

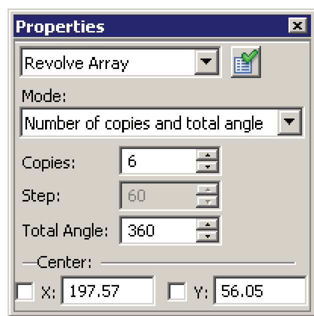


Two possible approaches are available for further construction. In one of them, you could draw five more circles, to extrude and subtract them all from the body of the part. Alternatively, extrude just a single circle, and then "replicate" the resulting hole by means of the circular 3D array.

The first approach by steps:

A handy tool for creating the required number of copies of the circle is the "Create Circular Array" command:

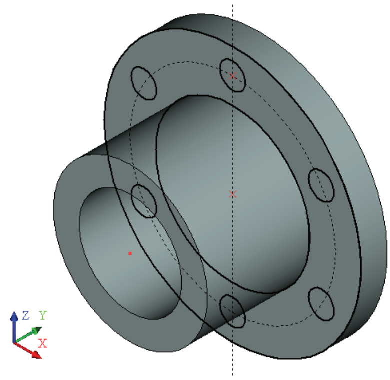
Keyboard	Textual Menu	Icon
<XR>	**"Draw|Array|Circular Array"**	



Once in the command, the system waits for selection of a graphic entity. Move the cursor to the image of the hole and click to select the graphic line. That's all to be selected, therefore, press in the automenu.

Now the system expects selection of the array center, which must be a 2D node. Note that the default setting for the number of copies of the circular array is 4, and we need six. Therefore, specify the correct parameters in the property window, as shown on the diagram at right.

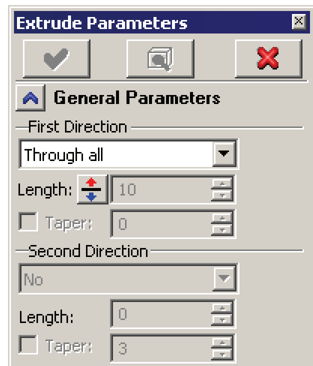
Now you can select the node. Move the cursor to the center node and click. The result will look as on the following diagram.



Next, we will need to call the profile extruding command,

Keyboard	Textual Menu	Icon
<3X>	**"Operation|Extrusion"**	

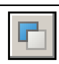
The system automatically makes a 3D profile from the drawn lines and sets the extrusion vector normal to the profile.



The amount of extruding, that is, the depth of the hole, can be defined in various ways (by a numerical value, by the length of the direction vector, etc.). In this particular case, we need to get a through hole, that is the one penetrating the entire width of the part.

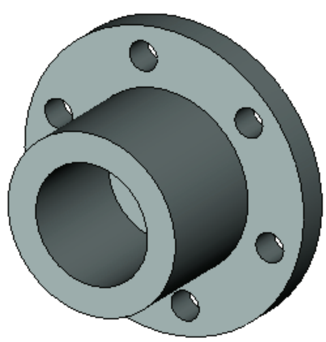
In the properties window, set the "First direction" parameter to the value "Through all", by selecting from the drop-down list. In this definition, the extrusion amount is determined by the thickness of the auxiliary bounding body. Additionally, the created extrusion is automatically subtracted from that body. This method of specifying the extrusion distance was devised specifically for quick creation of through holes in various bodies.

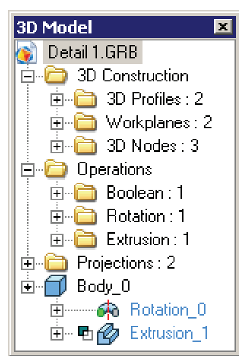
The option of creating a Boolean operation will automatically activate in the automenu:

	***<Ctrl><B>***	Subtraction

Since only one body is present in the scene at the time of creating the extrusion, it will be automatically selected as the bounding body and the first operand of the Boolean subtraction.

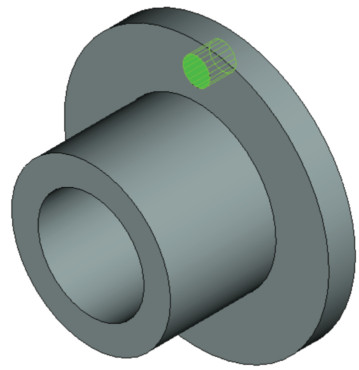
Upon clicking , two operations will be created at once – an extrusion and a Boolean operation.





The second approach by steps:

Call the profile extruding command . Specify extrusion length parameter equal to 20 in the reverse direction, and confirm the extrusion operation.



Next, let's call the circular array creation command,

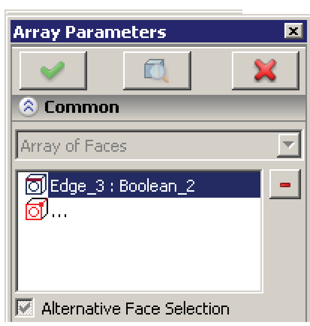
Keyboard	Textual Menu	Icon
<3AR>	**"Operation|Array|Circular"**	

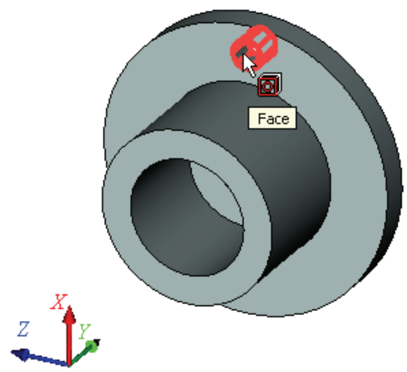
In the command's properties window, select the array type – "Array of Faces".

The following option will be automatically activated in the automenu:

	<N>	Select Faces to Copy or their boundary Edges or Loops

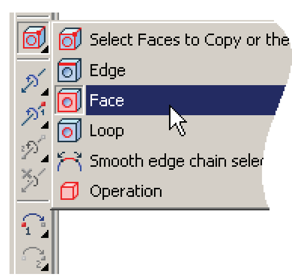
Aim the pointer at the cylindrical face of the hole and click .



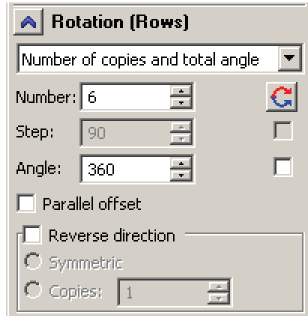


If edges or loops highlight instead of the face, then you can do any of the following:

- rest the pointer over the face until the multiple selection widget appears next to it. Then scroll through the list of elements offered for selection by rotating the mouse wheel until you get the desired face;
- before selecting the face, set up the list of 3D objects available for selection

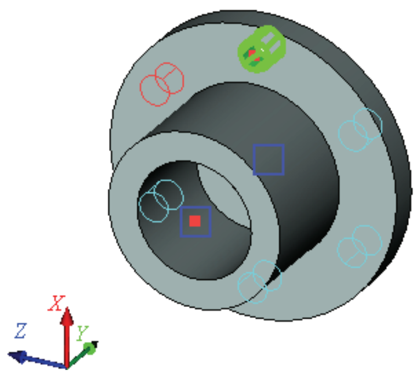


(banning from selection anything but faces). To do this, double-click the icon of the faces filter . The list of selectable objects can be also set up by the drop-down menu of the option : point at the option, depress and hold the mouse button for a second. In the appearing menu, disable all flags except "Faces".



Once the intended operation is selected, the system expects input of the axis of revolution for the array. The axis can be assign by the same two 3D nodes as those used in the first method of creating holes. Select these two 3D nodes one by one.

You can define the array parameters in the properties window – number of copies and total angle.



Press the button in the automenu to complete the operation. As a result, five new faces will be "embedded" in the original body, making up the additional holes.

Creating a Blend

The next step is finalizing 3D model creation. This includes creation of a chamfer and a rounding.

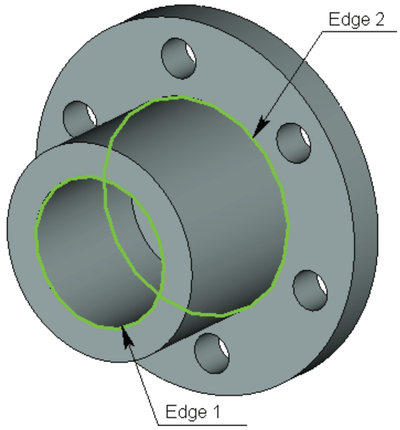
Call the command:

Keyboard	Textual Menu	Icon
<3D>	**"Operation|Blend/Edge"**	

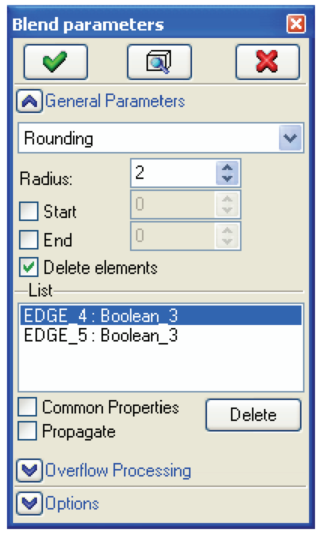
Pick the following option in the automenu,

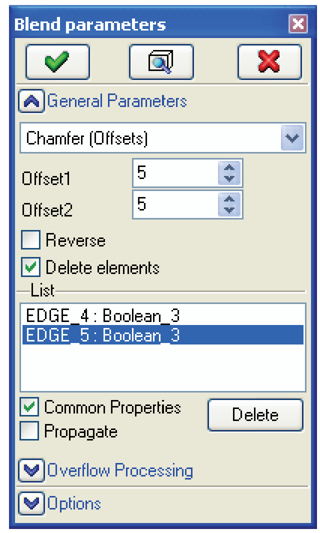
	<E>	Select Edge

The system is waiting for an edge selection. Select two edges in the order shown on the following diagram.

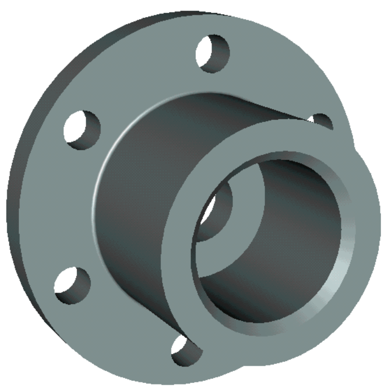


Next, specify the operation parameters. Each edge can be assigned its specific set of parameters in the property window by unchecking the “Common Properties” option.



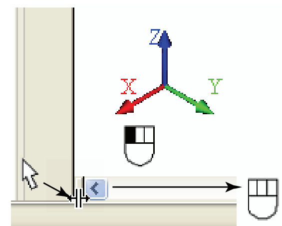


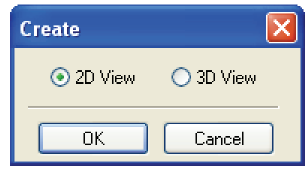
Select the first edge in the list and specify the blending type as Rounding with Radius equal to 2 mm. For the second edge, define the blending type as Chamfer (Offsets), with Offset1 and Offset2 both equal to 5 mm. Confirm the operation by pressing . The result appears as shown on the following diagram.



This completes creation of the given 3D model.

Creating a Drawing





Open a 2D window. Do this as follows. Move the cursor to the bottom-left corner of the 3D window and locate the split box before the scroll bar. As the cursor approaches the split box, it changes to juxtaposed arrows. Press and drag rightwards up to the middle of the window, then release the mouse button as shown on the diagram.

The system will ask what kind of window should be opened. Mark the "2D View" and press **[OK]**.

The 2D and 3D windows will be located side by side of the vertical split bar. The same result could be obtained by calling the command:

Keyboard	Textual Menu	Icon
<WSR>	**"Window|Split Vertically"**	

A new window can also be opened with the command:

Keyboard	Textual Menu	Icon
<WO>	**"Window|New Window "**	

To activate the newly opened 2D window, place the cursor within and click .

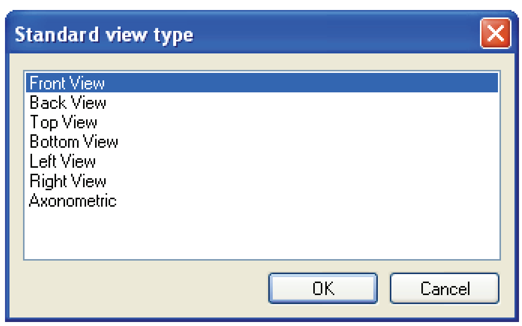
Now we can generate projections and sections. Call the projection creation command:

Keyboard	Textual Menu	Icon
<3J>	**"Draw|2D Projection"**	

Press the automenu button:

	<6>	Create standard projection

In the coming dialog box, select "Back View" and press **[OK]**.



A green box will be displayed on screen, marking the size of the projection to be created. Next, use the option:

	<M>	Change-Projection Placement

Define the attachment point of the projection in absolute coordinates by clicking at the desired location in the 2D window. Press in the automenu to confirm projection creation.

Next, let's generate a section of the part.

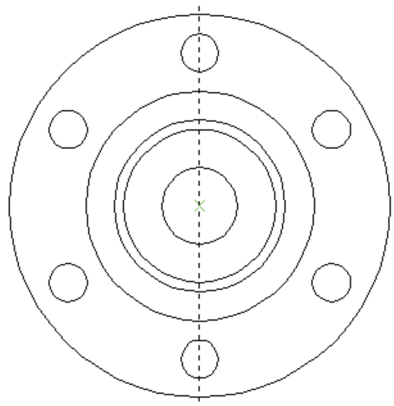
First, we need to do some auxiliary construction. Call the line creation command,

Keyboard	Textual Menu	Icon
<L>	**"Construct|Line"**	

Pick the automenu option for creating vertical lines,

	<V>	Create vertical Line

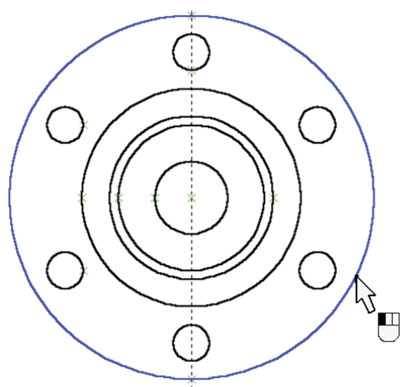
Point at the center of the circle and click , creating a line along the symmetry axis of the projection.



Then, call the circle creation command:

Keyboard	Textual Menu	Icon
<C>	**"Construct|Circle"**	

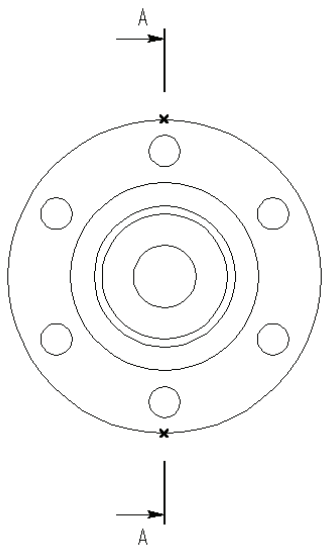
Next, we need to reconstruct a construction circle from a graphic circle on the projection. Move the cursor to the circle graphic line and click .



Now, we can create the section points snapped to construction line intersections. Call the section creation command,

Keyboard	Textual Menu	Icon
<SE>	**“Draw|Section”**	

Select two intersection points between the line and the circle one by one. Press automenu icon to confirm the section creation.



The next step will be creating a projection of the newly created 2D section. Call the 2D projection creation command,

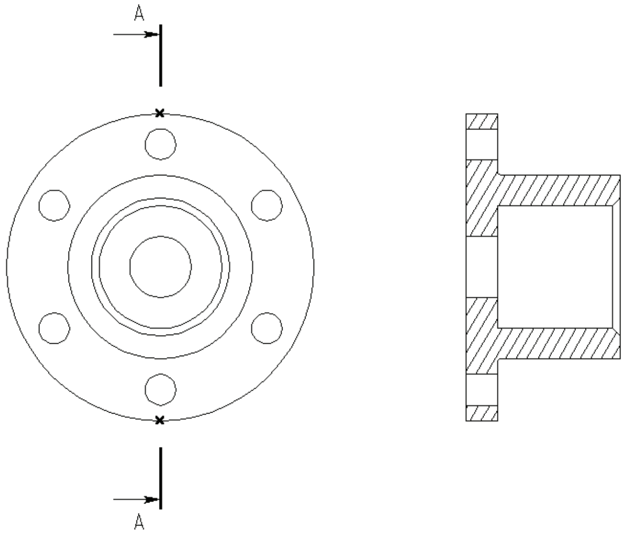
Keyboard	Textual Menu	Icon
<3J>	**“Draw|2D Projection”**	

In the automenu, select subsequently the following options:

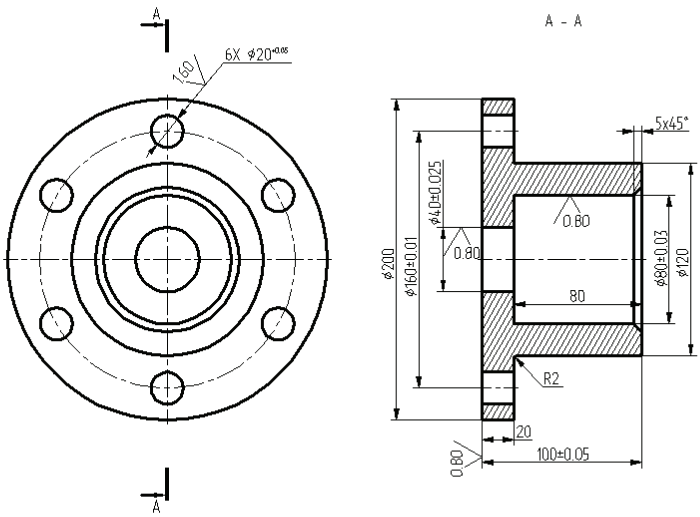
	<2>	Create section view
	<L>	Select Section defining Projection

Point the cursor at the section symbol and click . The outline of the section appears rubberbanding with the cursor. The projection outline will slide along the projection direction adjusting to the cursor movement. To fix the placement, click .

To complete projection creation, press in the automenu.

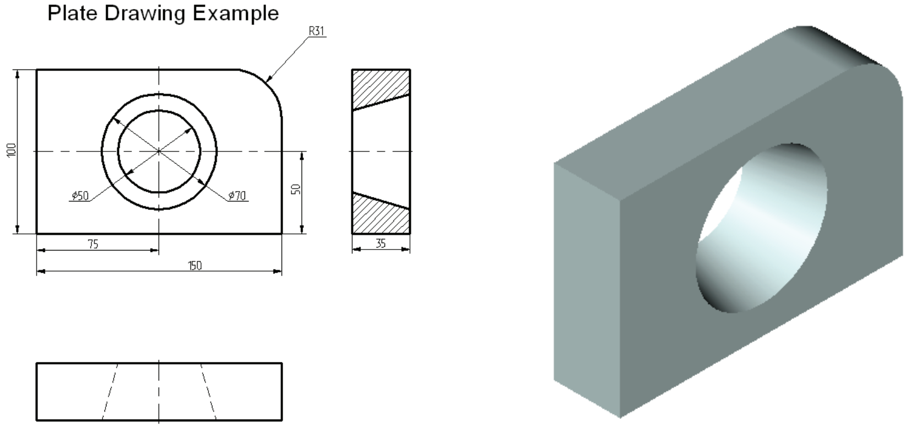


If necessary, the projection entities (graphic lines, arcs and circles) can be used as references for dimensions, auxiliary graphic lines such as centerlines, and drawing layout attributes.



“From Drawing to 3D Model” Approach

Let’s create a 3D model from a 2D drawing. The model will be based on the familiar drawing of a plate with a conical hole. This drawing creation was reviewed in details within the brief introductory course of 2D design. The following diagram presents the 2D drawing and the 3D model of the plate to be created.



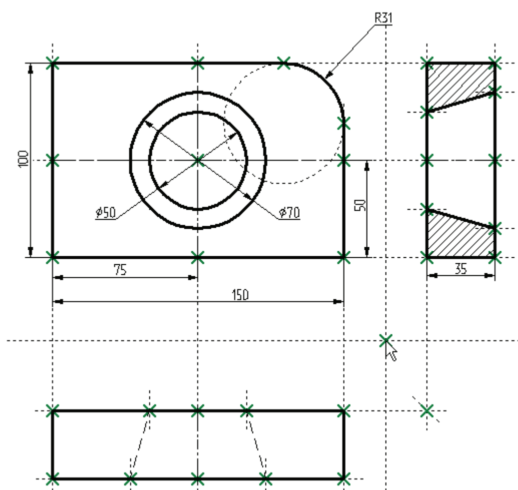
> The drawing document is located in the library "`Documentation samples`", the folder "`3D Modeling\Brief introductory course\Plate 1.grb`".

We will pass through several design steps in order to create the 3D model of the plate. The first design step includes creation of the workplanes that will further be used for all the rest of 3D element construction. In the second step, we will perform extrusion of the defining contour of the plate at the specified depth in order to obtain the three-dimensional body of the plate. This will be done by extrusion operation. The next design step is creation of the tool body for the conical hole. We will create it using rotation operation. Finally, we will use Boolean operation in order to create the three-dimensional model of the plate in its final shape. We will subtract the second, tool body, created by rotation operation, from the original body created by extrusion.

The first step in 3D model creation will be creation of the workplanes, as was mentioned earlier. Note that the workplanes should be constructed in such a way to maintain projective relationship between the views.

> If not explicitly defined within the drawing, the projective relationship between the views can be established by fitting the drawing with associative auxiliary construction.

The 3D model creation process requires two workplanes. Let's begin with creating a 2D node defining the separation point of the views. To make the node, first create two orthogonal lines. Place these lines as shown on the diagram below.



Then create the workplanes.

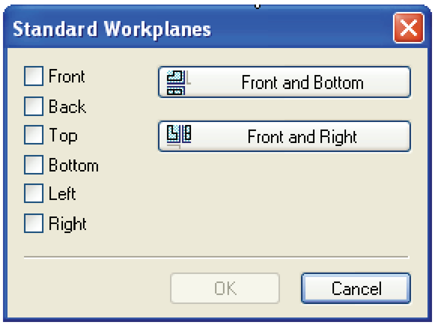
Call the command:

Keyboard	Textual Menu	Icon
<3W>	**“Construct|Workplane”**	

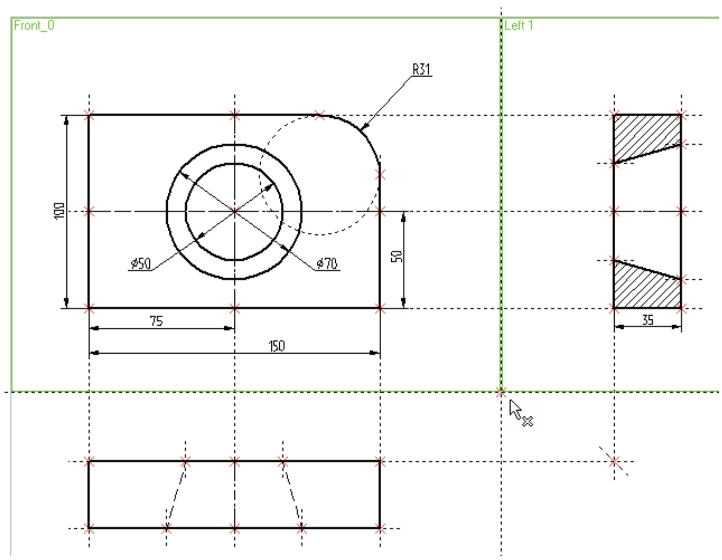
Set the automenu option:

	<S>	Construct standard Workplane

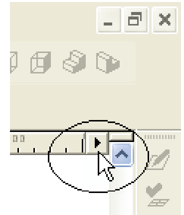
In the coming dialog box, press the button [**Front and Right]**



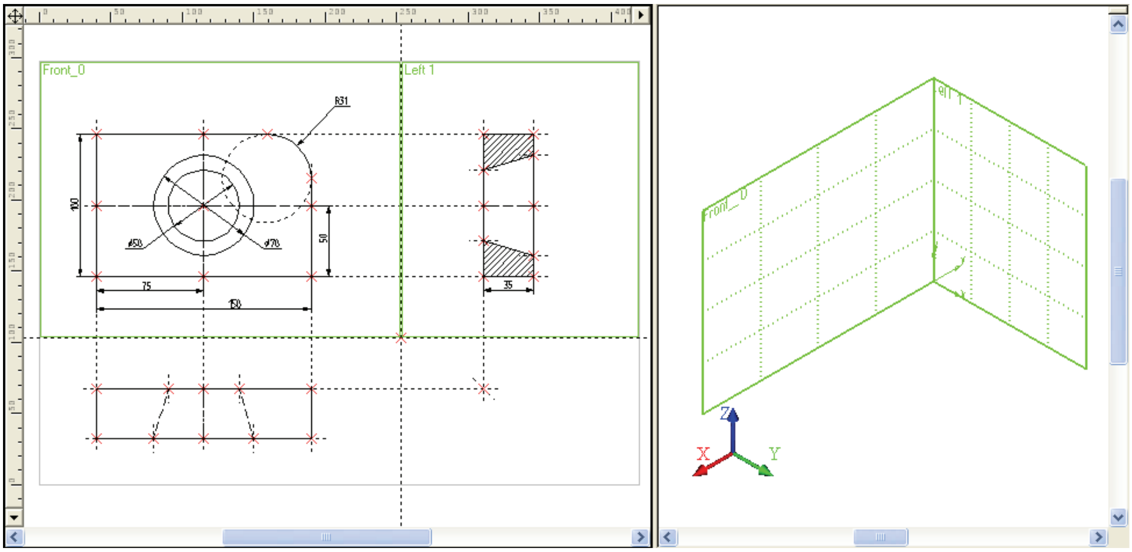
The cursor on screen gains the node mark . This mark means the system is now in selection mode. Move the cursor to the newly created node to be used for workplane snapping, and click or type ***<N>*** key. Two horizontal workplanes will be created on screen for the main (elevation) and the left side view.



Then quit the command by clicking or pressing icon in the automenu. One can also type the ***<Esc>*** key.



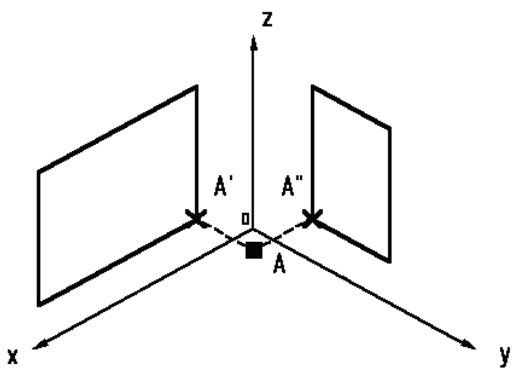
Now we can proceed with creation of 3D auxiliary elements. First, open the 3D view window of the T-FLEX CAD 3D system. The 3D view can be opened by clicking on a button with a triangular arrow in either the top-right or the bottom-left corner of the current drawing window next to the scroll bars. Place the cursor over such a button in the top-right corner, and click on it with . The current drawing window will be split vertically into two windows. One of the windows will be displaying the two-dimensional drawing, while the other, 3D view one, reflect the three-dimensional elements and bodies as they appear in the process of the 3D model creation.



> Set the workplanes property "**Show in 3D View**" in order to have them displayed in the 3D window. To do it, it is necessary to select these planes in the tree of the 3D model, call the context menu with the help of  and invoke the command "Parameters" in it.

To create the main body of the 3D model of the plate, extrude the perimeter contour at the length equal to the plate depth. To define the extrusion based on the existing 2D drawing, we need to create auxiliary 3D elements: a 3D profile and 3D nodes in a special way.

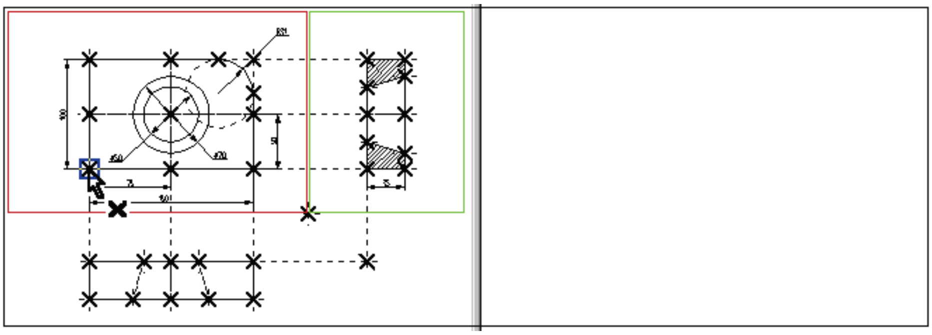
Let's create 3D nodes using existing 2D nodes and the workplane mechanism. To create a 3D node, one can specify either one node on one workplane, or two nodes on two different workplanes. In the latter case, the two nodes must be relatively located as two projections on their respective workplanes of one would-be 3D node.



Call the command:

Keyboard	Textual Menu	Icon
<3N>	**"Construct|3D Node"**	

Next, proceed with selecting 2D nodes. Move the cursor to a node of the first projection of the would-be 3D node and click as shown on the diagram below. The 2D node and its workplane will get highlighted.



> Normally, upon entering a 3D command, the system expects further user actions. Depending on these actions, the system assumes one or another specific mode. This mode usually depends on what kind of object was selected by user. The necessary options may automatically be set in the automenu depending on the user actions and the specific mode. The user can simply proceed with a predefined sequence of actions following the prompts in the status bar.

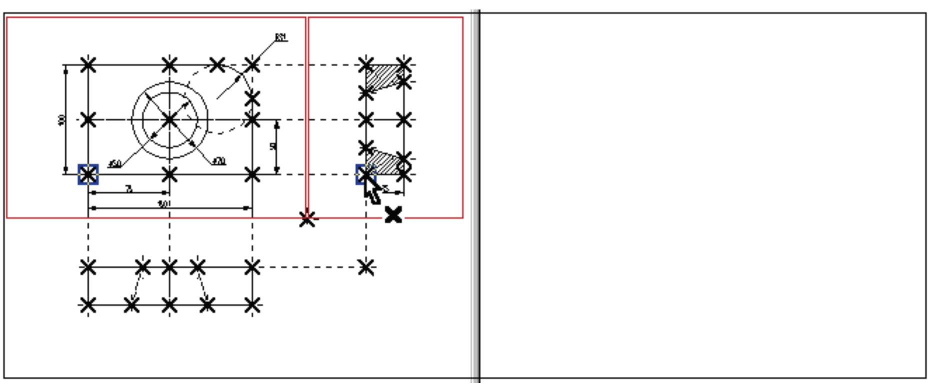
Once the first 2D node is selected, the system automatically turns on node creation by two projections:

	<J>	Create 3D Node by two projections

Once this mode is on, the user will be subsequently prompted to select the first, then the second projection 2D node, on and on, as he keeps selecting:

	<F>	Set 3D Node 1st projection
	<G>	Set 3D Node 2nd projection

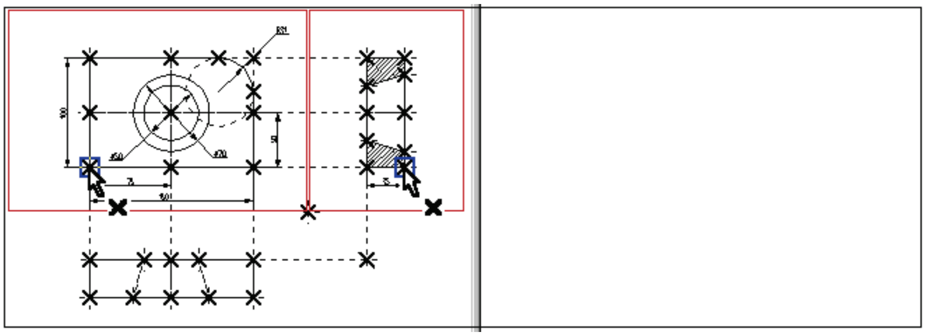
Move the cursor to the second node, representing the second projection of the 3D node, and click . The 2D node and the respective workplane, where the selection occurs, will get highlighted.



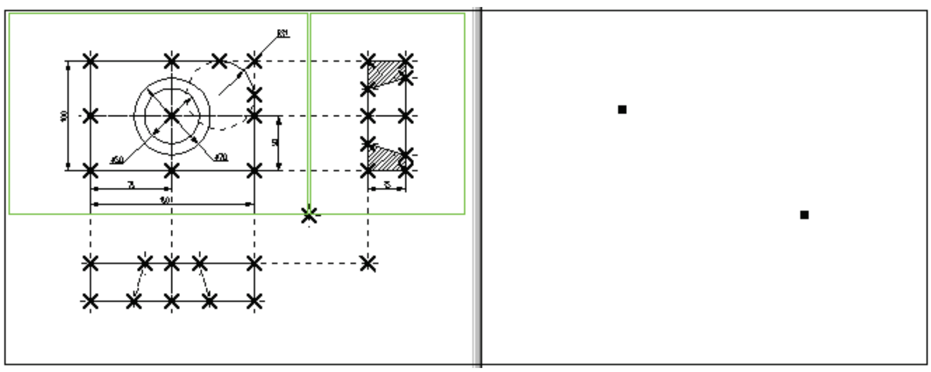
Confirm the 3D node creation by pressing  icon in the top portion of the automenu. The 2D drawing elements get de-highlighted, and the newly constructed 3D node appears on the 3D scene.

> Remember that some 3D commands are inaccessible while the 2D view of the current drawing is active. To make these commands accessible, activate the 3D view window. To do so, click  anywhere within the 3D view window. Switching from 3D view back to 2D view is done in the same way.

Let's create the second 3D node. We are still within the command **"3N: Construct 3D Node".** Construct the second 3D node. To do so, we need to select two nodes as shown on the diagram below.

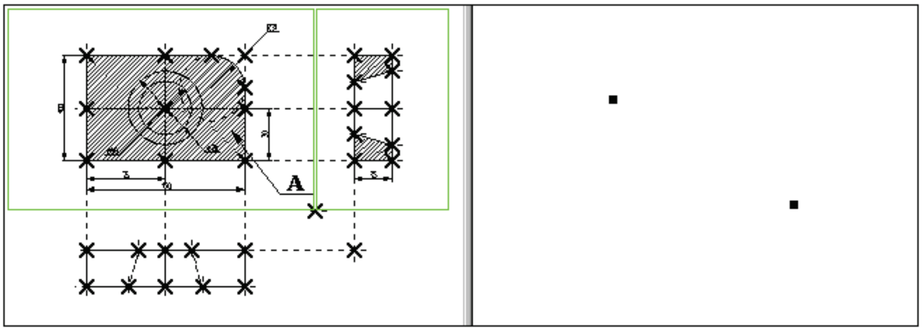


To confirm the second 3D node creation, press the  icon in the automenu and exit the command. Two 3D nodes will be displayed in the 3D window.



The next step is creation of a 3D profile. We need to establish a relationship between the profile and the drawing. Drawing on the active workplane is not available to us since there is more than one workplane on one drawing page. Therefore, we will proceed with the 3D profile construction based on a hatch and a workplane. First, let's create the hatch.

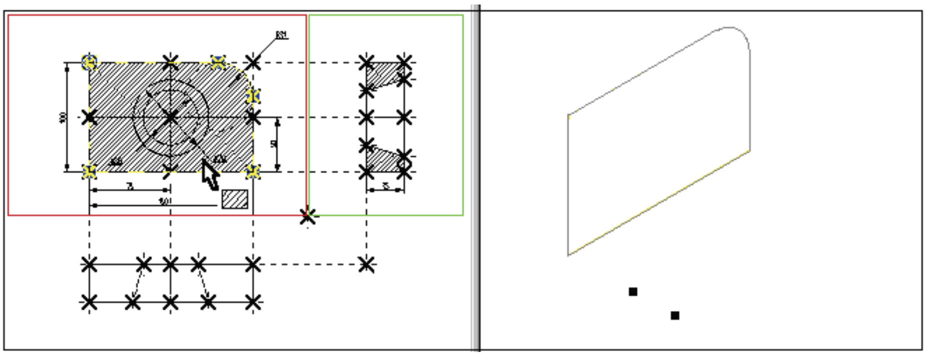
Enter the command **"H: Create Hatch"**. Create a hatch **A** on the elevation view. The hatch can be made invisible by setting the "**Fill method**" attribute of the hatch parameters to "**Not Visible**" mode. This is necessary for preserving the original look of the part drawing.



Now call the command:

Keyboard	Textual Menu	Icon
<3PR>	**"Construct|3D Profile"**	

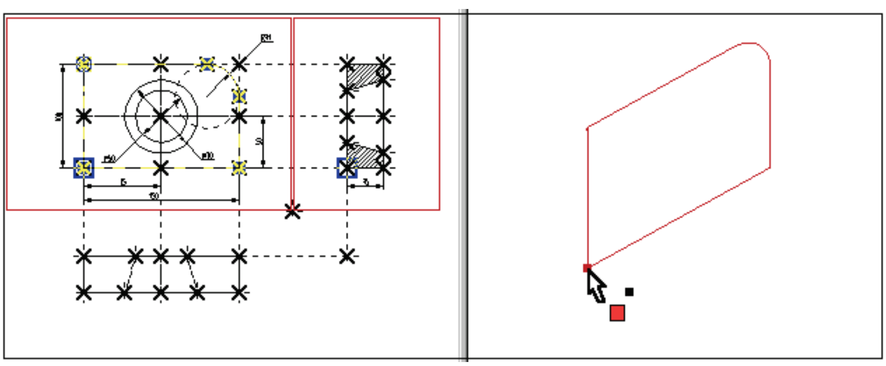
The system waits for further user action. Select the hatch **A** in the 2D window. The contour of the selected hatch and the workplane will get highlighted, and the respective profile will appear in the 3D window.



Once the hatch is selected, the system enters the mode for constructing a profile based on a hatch, and the following option activates in the automenu:

	<M>	Select 3D point for fixing Profile plane

The 3D node mark appears next to the cursor. Use the cursor to select one of the 3D nodes in the 3D view window. The profile will relocate to pass through the selected 3D node. The node itself and all its 2D and 3D construction references will be highlighted.



Confirm the 3D profile creation by pressing the icon in the automenu. All elements in either window will then be de-highlighted. We have just created a 3D profile. Quit the command to continue.

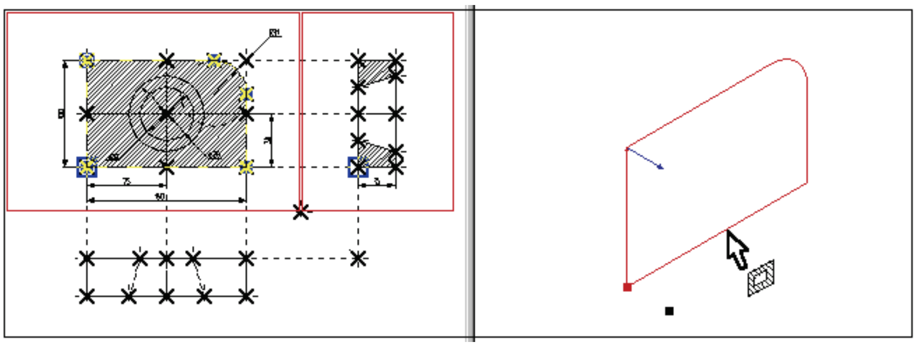
All auxiliary 3D elements have been constructed that are required for a three-dimensional body creation by extrusion.

Let's create an extrusion operation . The following option will be set in the automenu by default,

	<R>	Select Contour

As we are to use the just created profile as the contour, let's set up the filters for 3D profile selection. This can be done on the system toolbar by on the desired filter icon as well by clicking while holding the ***<Ctrl>*** key. All the rest of the filters will get turned off.

Move the cursor to the 3D window, place it by the profile and point at an edge of the profile. The cursor will gain a mark . Click to select the profile. The profile will be highlighted in the 3D window.



Keep in mind that selection of 3D elements required by 3D commands can be performed either in the 3D, or in the 2D view window. This is especially relevant to selection of 3D nodes and 3D profiles (contours) that can be selected in the 2D view as nodes and hatches respectively.

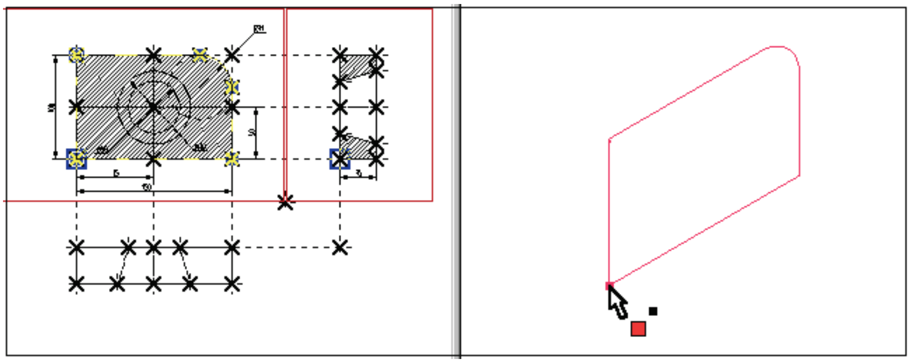
Next, we need to specify direction and the depth of the extrusion. In this example, the depth of the extrusion is equal to the thickness of the plate. This dependency can be maintained if the extrusion vector is defined based on the existing 3D nodes.

Once the 3D profile is selected, turn on the following option in the automenu,

	<F>	Select starting Point of Extrusion

Several views of the object can be used for specifying a 3D point. Make sure of the correct selection filter settings for 3D node selection on the system toolbar or in the pull-down list of active options.

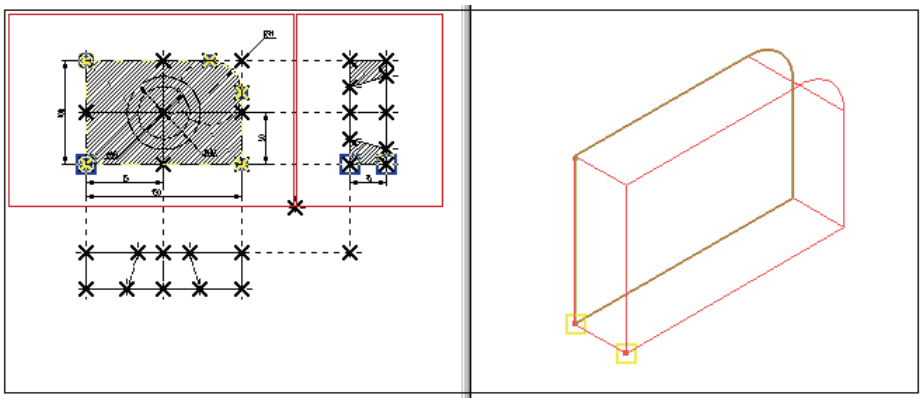
As the cursor is pointed at a 3D node, it gets the node mark . Select the 3D node, which the 3D profile plane is passing through.



After selecting the first 3D node, another option will be automatically turned on in the automenu,

	<S>	Select ending Point of Extrusion

Select the second 3D node with the cursor (point and click as usual). The preview of the current extrusion operation will appear on screen in wireframe mode.

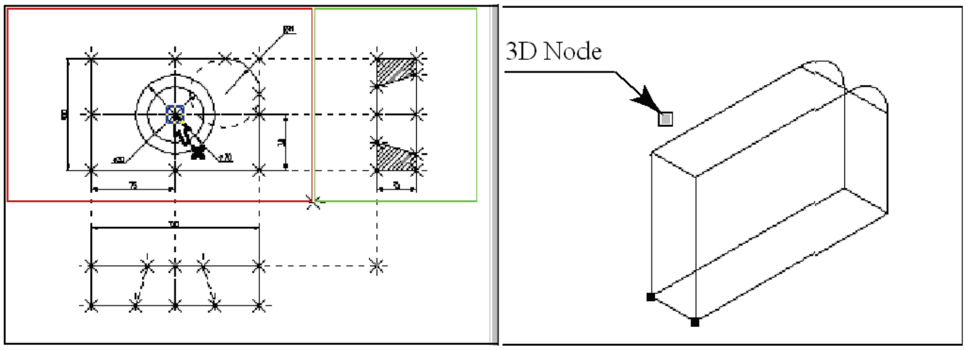


Press the icon, thus completing the extrusion operation. Quit the command.

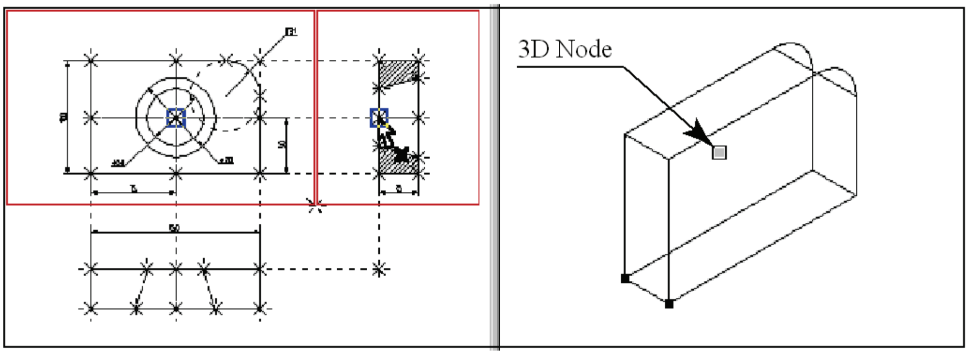
To create a hole in the three-dimensional model of the plate, we need to create the second 3D body. The latter will be used in the next step as the tool body for Boolean subtraction from the original extrusion. The 3D model of the hole is easiest to create by rotation operation.

To create a three-dimensional model by rotation, we need to define auxiliary 3D elements: the 3D profile and the axis of revolution for rotating the profile. To create the axis, we will need to construct 3D nodes.

Begin construction by calling the command **"3N: Construct 3D Node".** Enter the command, and select a 2D node as shown on the diagram below, by pointing and clicking . A highlighted 3D node will appear in the 3D window.

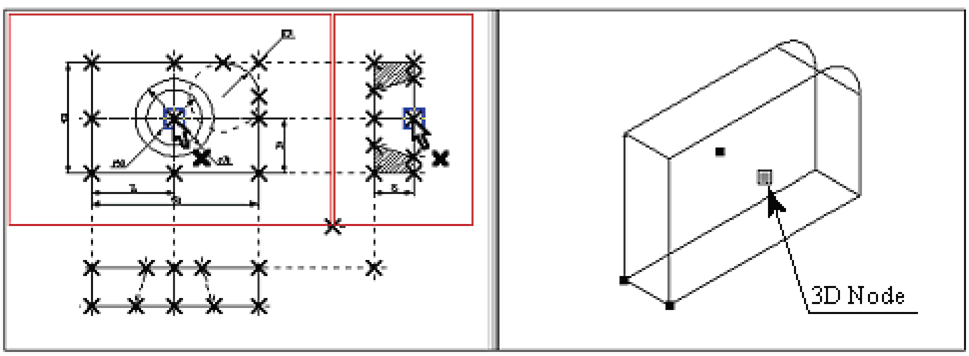


Next, move the cursor to the second 2D node, defining the second projection of the 3D node, and click . The highlighted 3D node in the 3D window will move along the Y-axis.



Confirm 3D node creation by pressing icon in the automenu. Highlighting turns off in both windows.

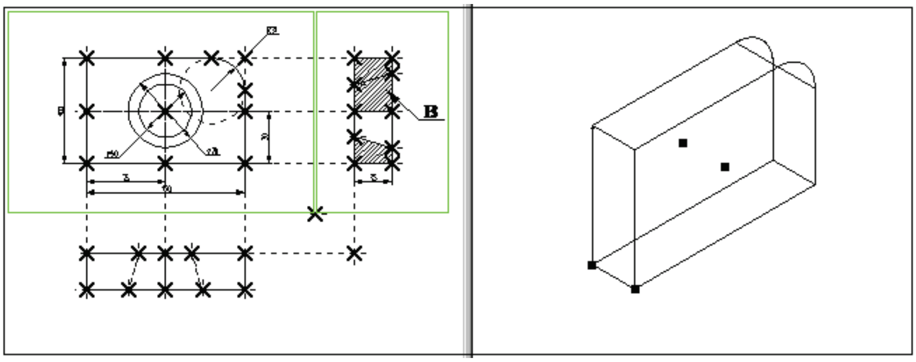
Create the second 3D node. It is necessary for defining the axis of revolution. The system is still within the command **"3N: Construct 3D Node".** Go ahead with the second 3D node creation. Do this by selecting two 2D nodes as shown on the following diagram.



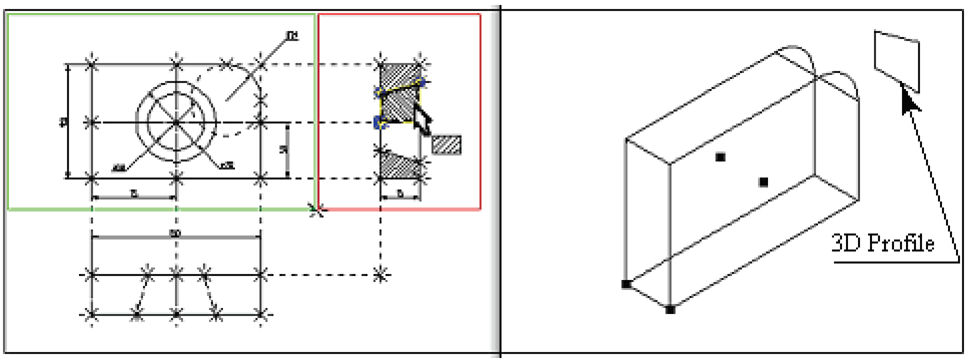
After the final selection press the icon in the automenu, and quit the command.

The next step is creation of the 3D profile.

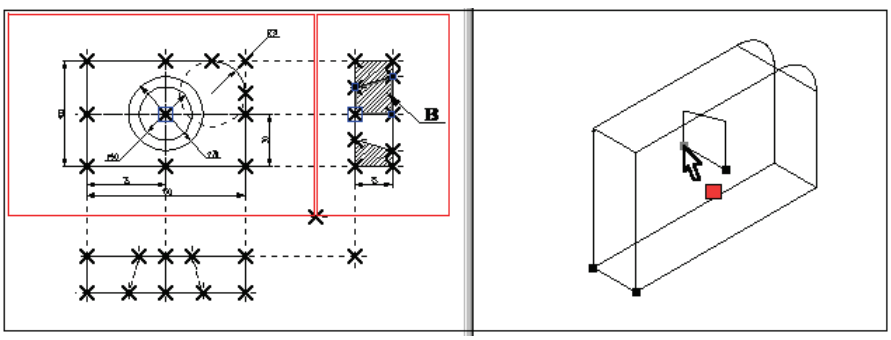
Create a hatch B on the side view of the 2D drawing using the **"H: Create Hatch"** command. Make the hatch invisible to keep the actual drawing legible.



Call the command **"3PR: Construct 3D Profile"**. Use the cursor to select the hatch B. The selected hatch contour and the workplane will get highlighted, and a 3D profile appear in the 3D window.



Once the hatch is selected, the cursor will gain the 3D node mark prompting the user for selecting a 3D node for locating the contour plane in space. Use cursor to select one of the 3D nodes in the 3D window. The profile will snap to the selected 3D node.

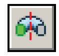


Confirm 3D profile construction by pressing the icon in the automenu.

The profile is now created. To continue with the three-dimensional model creation, quit the command.

We have created all required auxiliary 3D elements for creating the three-dimensional body by rotation operation.

Let's create the rotation operation. Call the command:

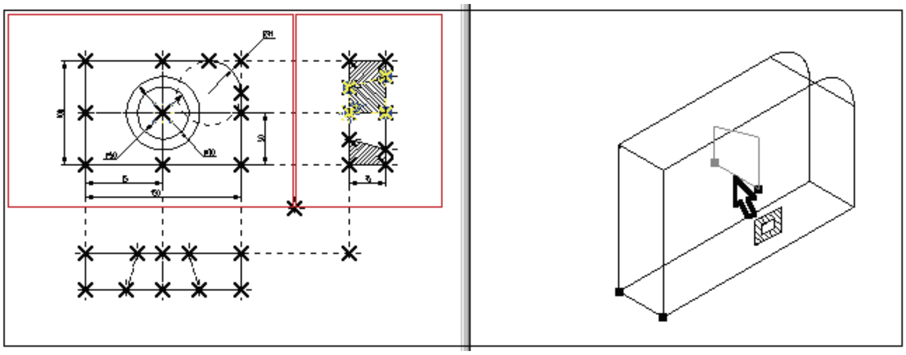
Keyboard	Textual Menu	Icon
<3RO>	**"Operation|Rotation"**	

Once within the command, the following default option is set in the automenu:

	***<R>***	Select Contour

Various elements can be used as a contour for rotation operation. We will use a 3D profile created specifically for this purpose. Before selecting a profile, check on the system toolbar or in the pull-down list that the selector filters are set correctly.

As the cursor approaches the 3D profile, it gains the profile mark . To select the profile, click . The profile will be highlighted in the 3D window.

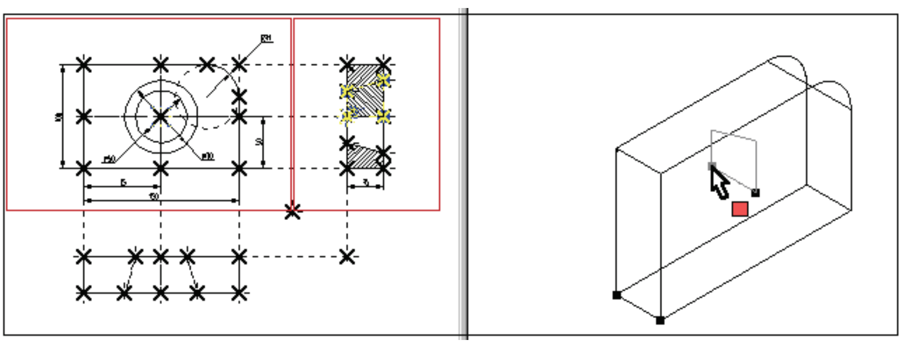


Next, define the axis of revolution by selecting two 3D nodes.

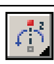
Set the following option in the automenu,

	***<F>***	Select 1st Point of axis

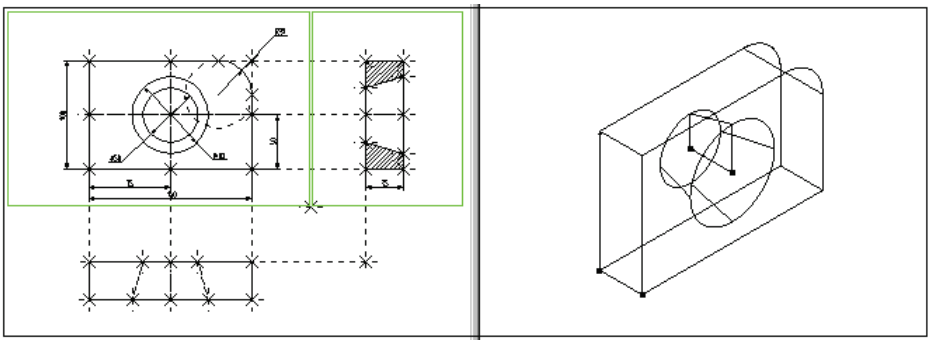
Move the cursor to a 3D node as shown below, and click .



The option will activate in the automenu thereafter:

	***<S>***	Select 2nd Point of axis

Move the cursor to the second 3D node defining the axis of revolution, and click . The body to be created will appear as a wireframe preview in the 3D window. The icon becomes accessible in the automenu, meaning that the rotation operation can now be completed. Press the icon. The following three-dimensional image will appear in the 3D view window.



We have just created two simplest 3D bodies, an extrusion and a body of revolution. Further, let's use the Boolean operation to get the intended shape. Subtract the second, tool body of revolution from the original extruded plate.

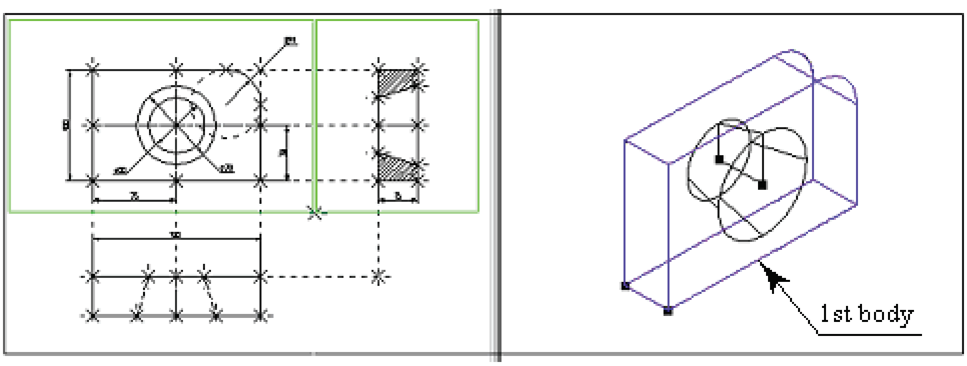
Use the following command to apply the Boolean operation to the 3D body:

Keyboard	Textual Menu	Icon
<3B>	**"Operation|Boolean"**	

Upon entering the command, the following option will be turned on by default in the automenu:

	***<F>***	Select 1st body

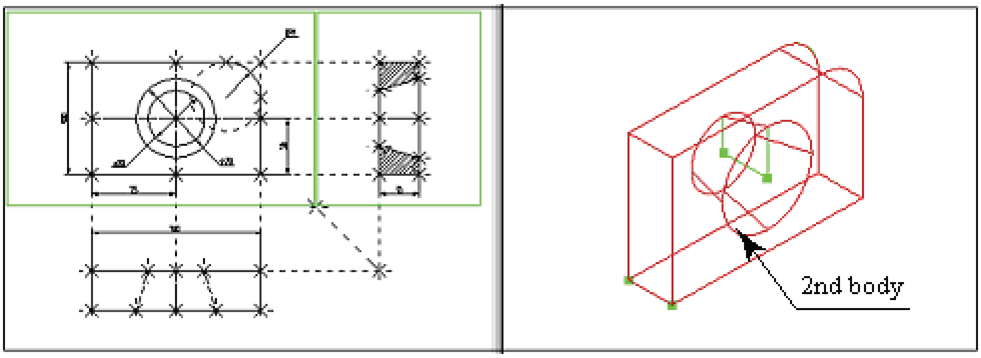
The cursor on screen will gain a mark . Select the first body for the Boolean operation in the 3D view window. This should be the extrusion, the target of the subtraction. Point at the body with the cursor, and click . The selected body will be highlighted in the 3D window.



Now we need to select the second, tool body for the Boolean operation. The following option will automatically be turned on in the automenu,

	***<S>***	Select 2nd body

Select the second body in the 3D window - the tool body to be subtracted. Point the cursor at the body of revolution, and click . The mark next to the cursor will disappear, and the second selected body will get highlighted in the 3D view window.



Next, select the Boolean operation type, which is subtraction in this case. Pick the following option in the automenu,

	<->	Subtraction

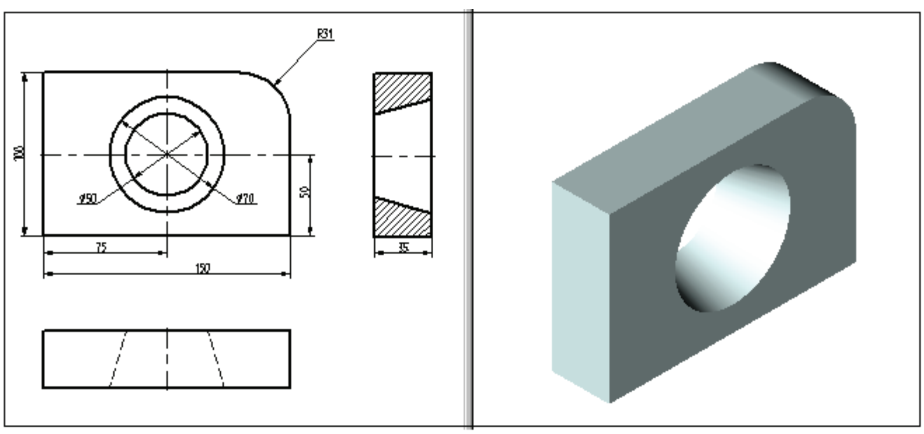
and press . The icon  becomes accessible in the automenu. This means, all operands have been set up for the Boolean. Press the icon. Highlighting turns off in the 3D window, and the operation completes.

To display shaded model in a specified color, call the command

Keyboard	Textual Menu	Icon
<3VS>	**"View|Render|Shading"**	

In the three-dimensional model creation, we were using two-dimensional parametric elements of the T-FLEX CAD system as references. Therefore, parametric modifications of the two-dimensional drawing will drive parametric modifications of the three-dimensional model.

Modify one or more dimensions on the 2D drawing using the command **"EC: Edit Construction"**, or **"V: Edit Variables"**.



Then call the command:

Keyboard	Textual Menu	Icon
<3G>	**“Tools|Regenerate”**	

This will cause recalculation of the 3D model geometry, and the model will adjust to the latest modifications made on the 2D drawing.