VariCAD

VariCAD

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Chapter 1. Foreword

This reference manual contains technical descriptions of the entire VariCAD interface, and consists mainly of detailed descriptions of modules and functions. If you need only brief basic information, read *Getting Started (page 10)* or *Tips and Tricks (page 322)*. We recommend that you read this manual while simultaneously using VariCAD.

To learn how to work with VariCAD, we recommend to read Basic Tips, available from pull-down menu Help. You can watch videos at youtube.com/varicadsystem. It is also possible to run Quick Demonstration examples.

Chapter 2. Introduction to VariCAD

Designed for use in mechanical engineering, VariCAD contains the following modules:

- 2D drawing
- 3D solid modeling
- Libraries of mechanical parts
- Libraries of 2D symbols
- Bill of material, automatic filling of title blocks
- Support of assemblies, including multi-level assemblies
- 3D parameters
- Geometric constraints
- Mechanical part calculations
- Sheet metal unbending (surface development)
- Crash tests (interferences)
- Pipes
- Thick shells (offset of patches)
- Import from and export to other CAD systems
- Support of 3D threads

All of these modules are included with the standard VariCAD package; there is no additional cost for any module.

Chapter 3. VariCAD Installation

VariCAD is distributed as downloadable files from our website, or optionally on DVD. After downloading the file - installation package, simply click the file icon. Installation runs automatically.

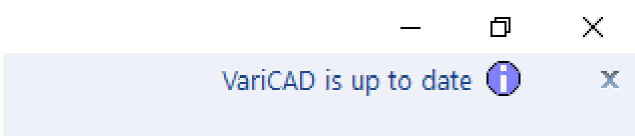
When installing on Windows, VariCAD uses NSIS routines, and for Linux the installation is created according to Linux distribution. To uninstall VariCAD, use the functions of your operating system. See also *Trial Versions, Online Purchasing (page 152)*.

Installing Upgrades

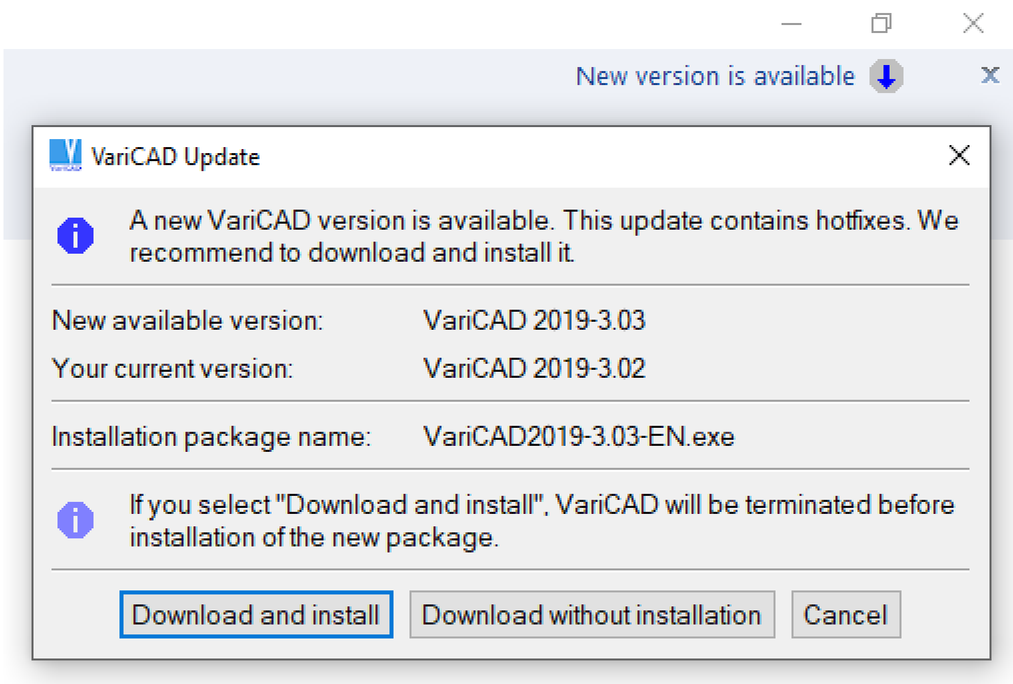
One-year upgrades are included in purchased installation package. After the upgrade period expires, you can purchase another one-year upgrade within maintenance package. During the upgrade period, you can install the new version over the old one and all system files will be upgraded properly. Your own 2D/3D files (VariCAD native files) will remain unchanged. Higher versions of VariCAD can always open files from lower versions. If you attempt to upgrade VariCAD after the upgrade period has expired, the installation routine will not work.

During start, VariCAD checks our server for available updates. If the system detects that you run older version, a message is displayed at the upper right corner of the VariCAD window. Click it, and you can install a new version.

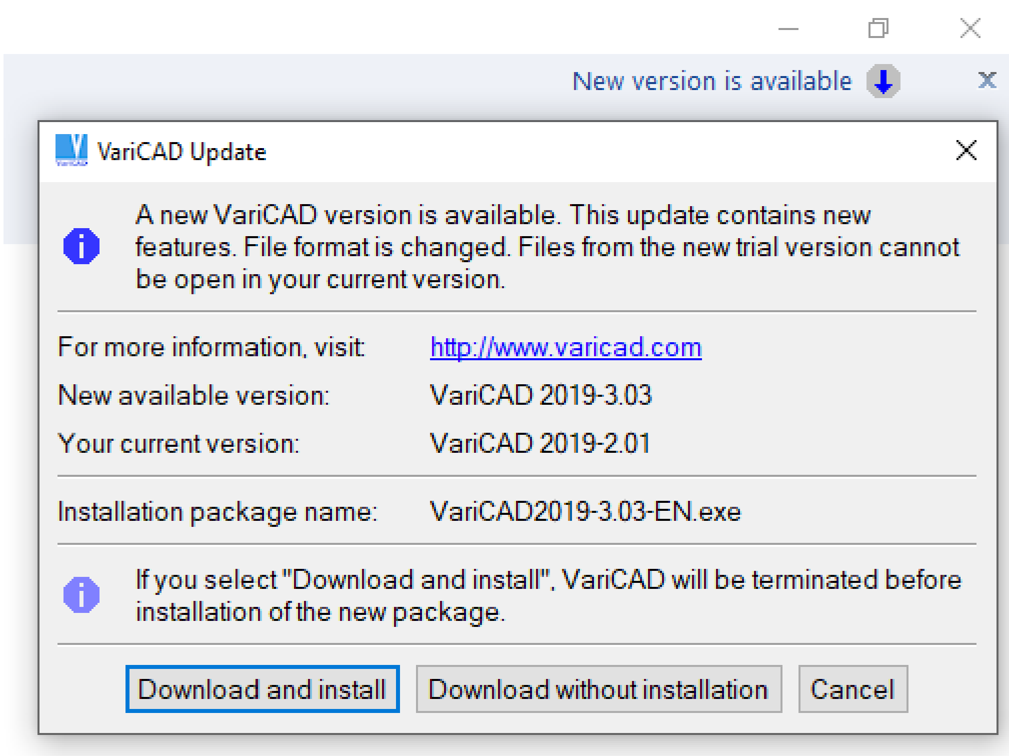
The version is downloaded and installed automatically. We strongly recommend to install each update containing hot-fixes. For update containing new features, you have an option to test a trial version parallel to your working environment, before you update the full version.



Upper right VariCAD window corner, VariCAD is up-to-date.



Upper right VariCAD window corner, a new version is available. The upgrade contains hot-fixes.



Upper right VariCAD window corner, a new version is available. The upgrade contains new features.

Hardware and Software Requirements

For Windows users, we recommend Windows 10, 8.1 or 7. For Linux users, we recommend SUSE, Kubuntu, Ubuntu, Debian or Red-Hat distributions. Preferred distributions are named as “LTS” (long-time supported) or professional distributions.

The recommended amount of RAM is at least 4GB, large 3D assemblies need more. Working with insufficient memory can cause swapping problems and significant decrease of operation speed. For most of tasks, 8 – 16GB of RAM may be sufficient. For extremely large data, use rather 32GB or more.

It is necessary to use a graphic card supporting OpenGL. Preferably, VariCAD works with OpenGL 4.0 or 4.3. If not available, we use old (legacy) OpenGL 1.1. In such case, some features may not be fully available and rendering of large 3D assemblies is slow. For most of tasks, graphic card with 2GB of RAM is sufficient. For large data is good to increase the amount of GPU RAM to 4 or 8GB.

VariCAD can efficiently exploit a powerful hardware. If available, VariCAD can use up to 64 threads (multi-threading), in various tasks. On the other side, if operating system and graphics run, VariCAD can run, too.

VariCAD can run at display resolution 800x600, although some functionalities are compromised. Commonly used resolution 1920x1080 (Full HD) is the best option for 24” monitors. Two monitors can be also used, but in such case, one monitor displays 3D or 2D drawing window. The second monitor displays windows containing data (like BOM, parameter tables, assembly scheme …). VariCAD works well under 4k resolution (3840x2160) – all graphic elements are adjusted automatically for high dpi.

A mouse is used for graphic input. VariCAD works with all configurations, uses up to 5 buttons (if they are available). For 3D input, 3D mouse working with 3DConnexion drivers is supported.

There are no special requirements for printers or plotters. VariCAD uses the device drivers. VariCAD supports output to all standard output devices. You can print at a physical device, or into a file. You can create output to *.pdf or post-script formats.

32-bit and 64-bit Versions

VariCAD is available as a 32-bit or 64-bit version. The 64-bit version requires, of course, a processor working in 64-bit mode and an operating system supporting 64-bit mode (like the 64-bit version of Windows 10, 8.1 or Windows 7).

If there is an option, we do not recommend to work under 32-bit system.

For Linux operating systems, you should choose the correct version of VariCAD before installation. For Windows, the correct version is selected automatically during the installation process.

Chapter 4. Files and Directories Used by VariCAD

Running VariCAD the First Time

On Windows, VariCAD is installed by default to “C:\Program Files\VariCAD”. On Linux, the default installation directory is “/opt/VariCAD”. The installation directories contain all system files. When running VariCAD the first time, the configuration directory is created under your working directory. Configuration files are copied to this directory, in which system settings are stored. A second directory is also created automatically after VariCAD startup; this directory will contain your data such 2D/3D drawings, additional data structures, bills of materials, etc. You are always informed when new directories are created. The only file in the work directory is one that contains definitions of directories used or created by VariCAD. To change the working directory, right-click on the VariCAD desktop icon, select Properties, and edit the line “Start in.”

Default VariCAD Files

The files you create are stored as the following file types:

- *.dwb - 2D/3D files containing 2D drawings and 3D solids
- *.bkb - block files containing 2D objects (obsolete, loaded only from older versions)
- *.sym - symbol libraries containing 2D symbols
- bom_mask.bmask – BOM, attributes and title block configuration
- *.mdata – list of materials
- *.dwb_fpv – file preview of folders containing VariCAD files

If you need to transfer VariCAD projects to another VariCAD user, be sure to include all files of these types. If you have to provide data created in a BOM import or export for another system, see *Bill of Materials (page 298)*. For Linux, VariCAD file extensions must be lowercase.

There are other files containing VariCAD configuration. These files can be saved into backup or restored by corresponding command (Menu Tools, then Backup or Restore All Settings).

Converting 2D/3D Objects to and from Other Formats

VariCAD supports the following formats:

- *.STP - STEP 3D, import or export.
- *.STL – Stereo lithography 3D (rapid prototyping), export.
- *.DWG - AutoCAD 2D, import or export. The DWG translator supports files from AutoCAD 9 through the latest AutoCAD. For AutoCAD 12 through the latest AutoCAD, you can load DWG files created as a result of VariCAD export.

- *.DXF - Same as DWG. The DXF format should be used for all systems other than AutoCAD.
- *.IGS - IGES 3D, export.

To import a specific file type, use the Files of Type field in the File-Open window. To export the active file to another format, use File / Save As. You can also export only the selected objects to any supported file format or import objects from any supported file format to the active 3D space or 2D drawing area.

Batch File Conversion - FCO

Use File / Batch File Conversion to convert multiple files of a selected format in a specified directory. The converted files are written to another defined directory. You can also customize settings for data translation. For DWG/DXF conversion, you can set units to inches or millimeters, or set the units to be detected automatically. For STEP format, you can select an application protocol and usage of NURBS vs. analytic types of curves or surfaces. Use the options in the Tools / System Settings menu to specify conversion settings.

How 3D Objects Are Converted to STEP or IGES

In general, 3D Objects are described as NURBS patches trimmed by NURBS curves. For some objects, curves and patches can be described analytically. These objects are:

- Line
- Circle
- Plane
- Cylinder
- Cone
- Torus (created by rotating a circle around an axis) - spheres are included but not lemon-shaped surfaces.

In most cases, the NURBS description is sufficient. However, analytic description can be faster and more accurate, and may require less memory and less storage space when saved. Run “CFG” command to set parameters for converting 3D objects to STEP or STL.

Creation of STL format

Run “CFG” command to set parameters of creation of STL files. STL format contains triangles covering surfaces of all exported objects. You can select accuracy of such approximation, a format of coordinates of triangle vertices, whether the objects are transformed to ensure all coordinates are positive and whether the coordinates are expressed in millimeters or inches. VariCAD exports all objects from the current file, or all selected objects, if the “Save Selected Objects” function is used. Some systems can require only one object in one file. In such case, use “Save Selected Objects” and select only one desired object.

STL output settings allow you to select triangles density (and consequently, accuracy of surfaces interpolation). It is possible to select from predefined accuracy, or to define accuracy by moving sliders. STL output can be visually checked. To do so, run command:

Test STL Output to Display – TESTSTL

Sections in Exported 3D Objects

You can choose export to STEP, STL or IGES formats with sections. Normally, export of sections is meaningless because sections are only used for display and do not describe geometry of the actual parts. However, exporting with sections can be useful if you want to render 3D objects in a visualization application. If the export with sections is not selected and the exported file contains any objects in an active section, warning message is displayed and export is cancelled. You can export the sections only when converting a single file, not in the batch files conversion.

Objects Imported from STEP

Objects imported from STEP are described as closed solids or open shells. In case of detected problems, some patches may be deleted during STEP file import. For more information about 3D objects shape representation, see *3D Objects Shape Representation (page 164) (page 154)*. In case some solids are not imported correctly, you can use tools for reparation, see *Solving Problems in 3D (page 167) (page 154)*.

Directories

Run “CFG” command to set directories of VariCAD backup files. We recommend not using network directories for backups. This directory is used also for work and temporary files created by VariCAD.

You can also define whether a directory of a particular file type is the same for both files loaded or saved or is different for load and save operations. You can also define a different or same directory for loading/saving complete files vs. loading file into the current file or saving only selected objects.

User Data Directories (2D/3D Files)

Directories used for user data saving or loading are by default the last used. Changing the directory when the file is saved or loaded, you change the default directory for the next usage.

All Directories (Folders) used by VariCAD

You can list all VariCAD system directories. In the list, you can right-click any line containing a directory and open the directory in File explorer (or Dolphin under Linux). The list displays these directories:

- Current working directory.
- Safety backups and temporary files
- Configuration directory
- Directory used last time for file open or save
- BOM mask directory
- Directory of list of materials
- Title block directory
- Bitmap images and print to files output

List of directories (folders) can be called from pull-down menu Tools or by command:

 **Information – INFO**

Information about Current File Changes

You can display information about currently open VariCAD file. Listed information is:

- VariCAD version the file was saved at
- Time and date of last save
- Time and date of last file modification
- User name and computer name used when the file was saved
- User name and computer name used when the file was modified

By other words, you can learn who and when saved or modified a VariCAD file the last time.

This feature is available from pull-down menu Tools, or can be called as command:

Information about Current File Changes - INFF

Chapter 5. Getting Started

VariCAD User Interface

The VariCAD window is divided into three parts. The largest part is the 2D drawing / 3D modeling area. The part above the modeling area contains the Menu Bar. Below the modeling area is the Status Bar. Toolbars are docked in various places on the screen, and toolbars can be undocked to become floating windows.

Working with Multiple Monitors

If VariCAD detects two monitors first time, a dialog panel is displayed and you may select from following options:

- Use only one (primary) monitor. In such case, VariCAD window and all dialog panels are open at primary monitor.
- Use two monitors, drawing at right (secondary) monitor.
- Use two monitors, drawing at left (primary) monitor.

If two monitors are used, some dialog panels (like BOM, list of assembly structure, panel containing icons for solid transformations …) are displayed at monitor opposite to main VariCAD window. Run command “CFG” to modify which panels are displayed at second monitor.

Two monitors can be configured if both desktops have the same dimensions and in virtual desktop, they touch each other from left to right and have the same Y coordinate origin.

If your computer (if it is a notebook) contains an external monitor and external graphic card, VariCAD selects the external graphics by default.

2D/3D Area

This area contains the 2D or 3D objects you create. You can switch between 2D and 3D at any time, and menus and toolbars will change accordingly.

Status Bar

For functions that do not require a window for input, all messages and prompts are displayed in the Status Bar. The following items are displayed on the right side of the Status Bar:

- In 2D and 3D, current units in millimeters or inches
- In 3D, dragging distance or dynamic rotation.
- In 2D, Ortho mode and Increment mode
- In 2D, cursor coordinates. Coordinates can be measured relative to a user-defined origin, as DX, DY from the last point, or as an angle and radius from the last point.

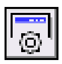
Cursor coordinates in 2D or dragging distance in 3D is also displayed near the cursor by default. If Ortho mode or cursor Increment mode is active, corresponding icons in tool-bar are checked permanently.

Toolbar Icons

Toolbars can be docked to sit above, below, or to the side of the drawing area, or they can be used as floating windows. Toolbars typically contain groups of icons for related functions, such as drawing functions, basic solids, dimensioning, etc. We recommend not removing the following toolbars:

- Switch to 2D/3D
- 2D layer selection box
- Command box

You can right-click on any toolbar to invoke a menu enabling you to add, delete, or reconfigure toolbars. You can also manage toolbars by using the following function:

 **Toolbar Settings - TLBS**

Configuration of toolbars for small icons is different than configuration of toolbars for large icons.

Dialog Windows - Cancel and Back Buttons

Optionally, VariCAD windows may contain buttons for “Cancel” and “Back.” Then the “Cancel” button cancels the current function completely. By default, only “Cancel” button without “Back” button is used. Then the “Cancel” button performs a step back.

Right-click while the cursor is inside the window is equivalent to click a highlighted button, usually the OK button.

Mouse Buttons

For both 2D and 3D, the default mouse settings are as follows:

- Left button - used for selecting objects or defining position. If the left button is pressed and held and if you move the cursor, selection window starts
- Middle button - proceeds one step back within a function. If the middle button is pressed and held and if you move the cursor, pan starts (display content is moved). Pressing a mouse wheel has the same effect as pressing the middle button – separate middle button is not a usual mouse configuration.
- Right button - completes a selection, equivalent to pressing Enter or OK. If the right button is pressed and held and if you move the cursor, 3D display rotation starts. Right-click object opens a pop-up menu with related features. Right-click an empty area opens a menu containing features related to current 2D/3D state.
- Mouse wheel – rotation enlarges or shrinks the display content (zoom). You can configure the rotation direction – whether the rotation toward you enlarge or shrinks the zoom.
- For 5-button mouse, VariCAD supports these additional buttons. They perform Undo or Redo actions.

Within a selection or location input, you can simultaneously press right and left mouse button, or press Ctrl + Right button to obtain a pop-up menu with currently available options.

See *Tips and Tricks (page 322)* for more information.

3D Mouse

For 3D view rotation, and 3D or 2D zoom or pan, VariCAD can optionally work with 3D mouse using drivers issued by 3DConnexion. According to 3D mouse model, you can work with additional buttons, too.

You can configure 3D mouse sensitivity and direction of movement or rotation, separately from driver settings. Run “CFG” command and in general section, select “Space Mouse Settings”.

If you are using a 3D mouse and if you right-click standard mouse button during display change, a pop-up menu with following options appears:

- 3D mouse view rotation on/off (on/off according to current state). If the rotation is turned off, only pan or zoom can be controlled
- 3D mouse pan/zoom on/off (on/off according to current state). If the pan/zoom is turned off, only view rotation can be controlled.
- View rotation center to defined point. This option is very useful especially for large models and greater zooms – you can redefine view rotation behavior whenever it is necessary.

In 2D mode, only pan and zoom can be controlled. If pan and zoom is turned off in VariCAD, it still works in 2D.

Invoking and Running VariCAD Functions

To invoke a function, you can:

- Right-click an object (see below).
- Select objects, then finish selection with right-click (see below).
- Click a toolbar icon. Tool-tip appears when you hold the cursor over an icon – it helps you remember the icon meaning.
- Use the Pull-down Menu. Some functions are embedded in several menu layers.
- Enter the command in the command box. A command history list is created; in which you can access previously used functions.
- Use hotkeys. Ctrl, Shift, Alt, and F-keys are used, sometimes with other keys, to invoke functions. When Ctrl is used, the current command ends and is replaced by the new one. The F-keys only interrupt the current function temporarily. You can configure usage of hot-keys.

Some functions always behave as the embedded ones – current function is interrupted temporarily regardless the method of embedded command calling. For instance, you can change a view or measure distances without the necessity of finishing the current command. Measuring distances is especially convenient. You can “Cut and Paste” results – use them as input for any values.

For a list of all embedded functions, see *Embedded Functions (page 350)*

You can configure the usage of hot-keys. It is possible to re-assign commands to selected hot-keys. Run command “CFG” to maintain the settings.

Finishing VariCAD Functions

Many VariCAD functions are “continuous.” For example, when drawing a line, you define two endpoints. After the line is finished, you can begin creating a new line. To end and exit a function, you can:

- Call a new function
- Press ESC if there is not a dialog window
- Press ESC if there is a dialog window - if necessary, multiple times to perform all steps back

If you exit a function, the Status Bar displays “Ready.” If the cursor is still within the drawing area, you can right-click an empty space to invoke the previous function from pop-up menu containing a list of commands history.

Stepping Back within a Function

Functions are typically performed in steps. For instance, when drawing a line, Step 1 is to define the first point, and Step 2 is to define the second point. To go one step back, you can:

- Press Ctrl + Z
- Click the middle mouse button (mostly, the same is to press mouse wheel), or press Ctrl + Backspace
- Press ESC or “Cancel” in dialog window (ESC outside a window cancels the current function completely)
- For 5-button mouse, press a corresponding mouse button.

Stepping back enables you to repeat or correct previous input without having to exit the function. Using Undo and Redo does not have the same effect; these functions actually change the 2D/3D object database. See *Undo and Redo (page 21)*

Selecting Objects between Functions

You can also select objects between functions (commands) and after the right-click, select a function from the pop-up menu. You can also right-click an object and then select a function from the pop-up menu. The offer of functions is, however, limited. Mostly editing functions are available.

You can use these additional methods to select objects between commands:

- To select all objects, press Ctrl + A.
- To select a part of solid in 3D (like a hole, fillet...), press Ctrl while moving the cursor.
- To select 3D edges for blending or 2D corners for fillet/chamfer, press Shift and move the cursor.
- To select a drawing plane for 2D drawing (sketching) in space, press Ctrl + Shift while moving the cursor.
- To start a selection window for stretching in 2D or in sketching in 3D, press and hold Ctrl + Shift and move the cursor.
- To select complete 2D view created by export from 3D, including updatable dimensions, axes or hatches, press Ctrl while moving the cursor.

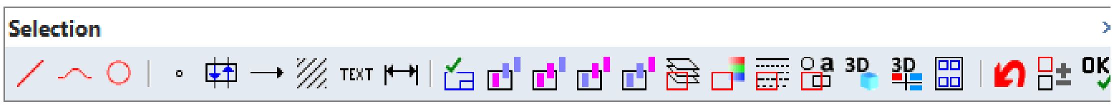
If you once selected solids, you cannot continue selecting edges or drawing planes until you deselect everything. Similarly, if you have selected edges, you cannot continue selecting solids. On the other side, you can combine selection of complete solids and selection of solid parts. Some edit functions support such combined selection.

To deselect all objects completely, press ESC. To change selection method, you can also right-click an empty location and change selection method from pop-up menu.

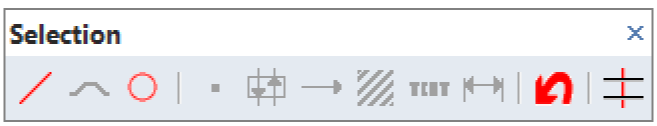
Additional Options within Functions

Many functions provide additional, temporary options. If you need to select an object or define a location, toolbars will appear with options relevant to the current situation. For example, when creating dimensions, you have the option of changing the dimension text or style while defining the dimension position.

All additional or standard options can be reached either by clicking corresponding icon or by pressing Ctrl + Right click (by default). The second method opens a pop-up menu with options.



Example of standard 2D selection toolbar



Example of 2D toolbar available for selection of single objects, and additional option icons

For a list of all functions and commands, see *List of All VariCAD Functions (page 324)*.

For a list of all hotkeys, see *Hotkeys (page 348)*

Creating, Opening and Saving VariCAD 2D/3D Files

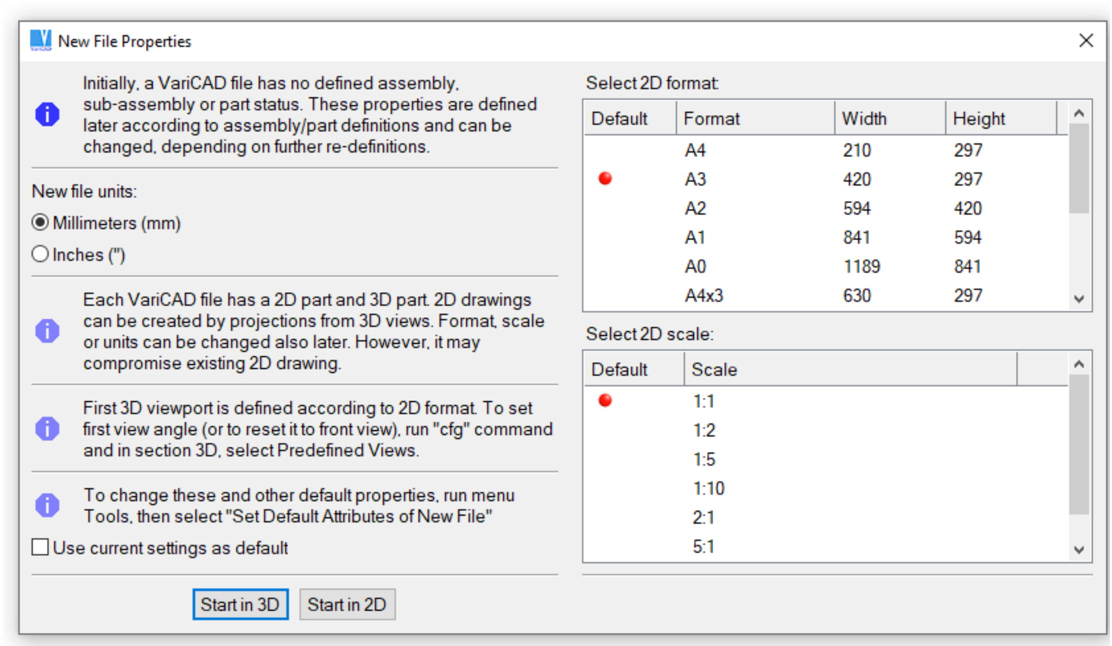
Create a New File – DOP, Ctrl + N

VariCAD always starts with a new, empty file. This file has the default file parameters, and is assigned a preliminary name of “NONAME” plus new file’s serial number (for instance NONAME 1). In order to save this file, you must assign a valid filename. If you want to create another new file, you can confirm or change its parameters. Defined parameters of new file are:

- Units - choose between inches or millimeters. If you change these units later, existing 2D dimensions will not change.
- Drawing format - choose layouts such as A, B, C, A4, A3, A2, etc. You can also define a custom format, and the format can be changed at any time. The drawing format is relevant when printing, if you select “Print according to format,” and for the Zoom Drawing Format tool. In addition, the 2D sheet border is created according to the selected format. In 3D, the format is only used for the initial dimensions of 3D space projection.

- Drawing scale - used only for 2D drawing. Dimensions, arrows, texts and other annotations are created according to the defined scale. The scale can be changed at any time, but be aware that this will change the existing annotation objects.

VariCAD file contains both 3D part and 2D part. Settings of a new file attributes are mostly related to 2D part. If you open a new file, you do not define assembly, sub-assembly or part file status. These attributes can be created later, according to assembly or part definitions.



New file creation window

Current File Attributes as Default - DEF

This function enables you to set the defaults for all new files. The dialog is similar to the creation of a new file (see the previous function).

The mode (3D/2D), units (millimeters or inches), sheet border format, scale and 2D grid distance can be defined in a window. Predefined 2D layers, 2D default line attributes (current layer, color and line style) and predefined angles of the construction lines are copied from the current file.

Open an Existing File – DAD, Ctrl + O

Opens an existing file. You can also open an existing file by pressing Tab, if the previous function is finished and “Ready” is displayed in the Status Bar.

Open Recent Files

This function allows you to open a file from the list of recently used files rather than from the standard file dialog.

Reopen the current document - REOPEN

Opens again the current document. If there are any changes, you are prompted before they are discarded. This commend can be used for instance, if you make changes and if you want to start again differently.

Close – CLO, Ctrl + F4

Closes the current file. Next current file is the file which was active previously. If only one file is open, you are asked whether to exit the session or to create another new file with the default parameters.

Saving and Inserting 2D/3D Files

Save – DSV, Ctrl + S

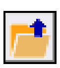
Saves the current file. If the file was created as new or copied to new document window, you must define a valid filename.

Save As – SVA

Saves the current file. You can select a different filename or different file type, like STEP, DWG etc.

Save Only Selected Objects - DPS

Select objects first. Then define a filename. You can save selected objects to any other supported file type.



Insert Objects from File - DPO

Inserts all objects from the selected file to the current file. If you are in 2D, only 2D objects are inserted. If you are in 3D, only 3D objects are inserted.

Selection of 2D objects is described at *Selecting, 2D Objects (page 45) (page 27)*. Selection of 3D solids is described at *Selecting Solids (page 204) (page 154)*. 2D objects from another file are inserted similarly as 2D blocks – see *Insert Block (page 108) (page 27)*. 3D objects from another file are inserted the same way as the solids are transformed and copied – see *Transforming and Copying Solids (page 229) (page 154)*.

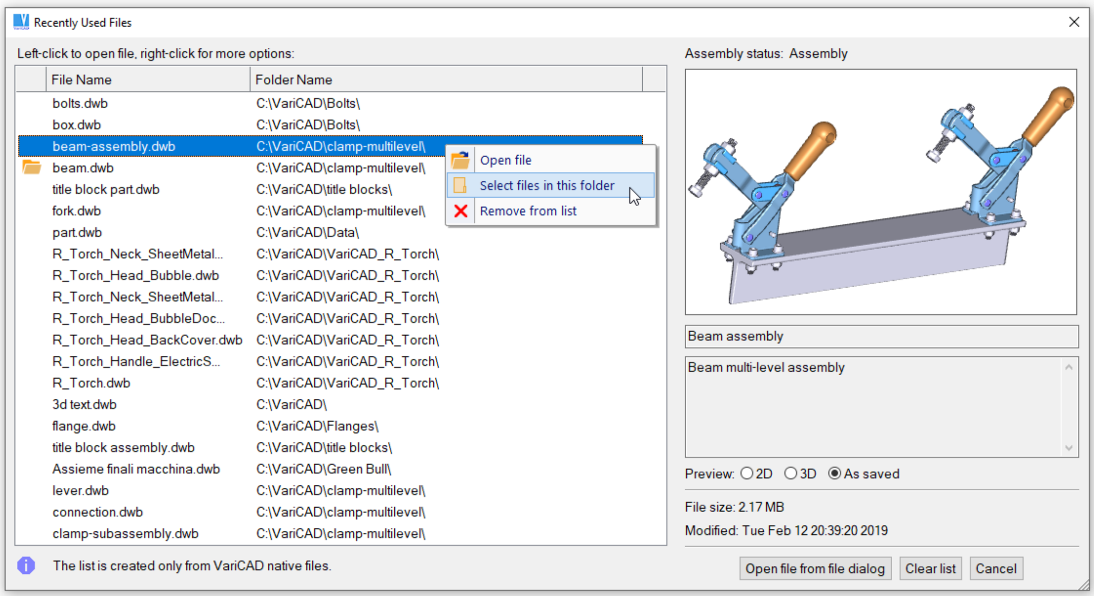
Open/Save Dialog and Recent Files Dialog

"Open an existing file" dialog panel and "Open recent files" dialog panel contains also a file preview. Moving cursor over list of files, the preview displays highlighted item information:

- Content image (for folders only if they are defined).
- Assembly status or folder prompt.
- Title (if defined).
- Description (if defined).

Both panels can be switched between each other. Apart from opening of previously used files, Recent files dialog allows you also to open files from folder the highlighted item belongs to. Such an option is available from right-click.

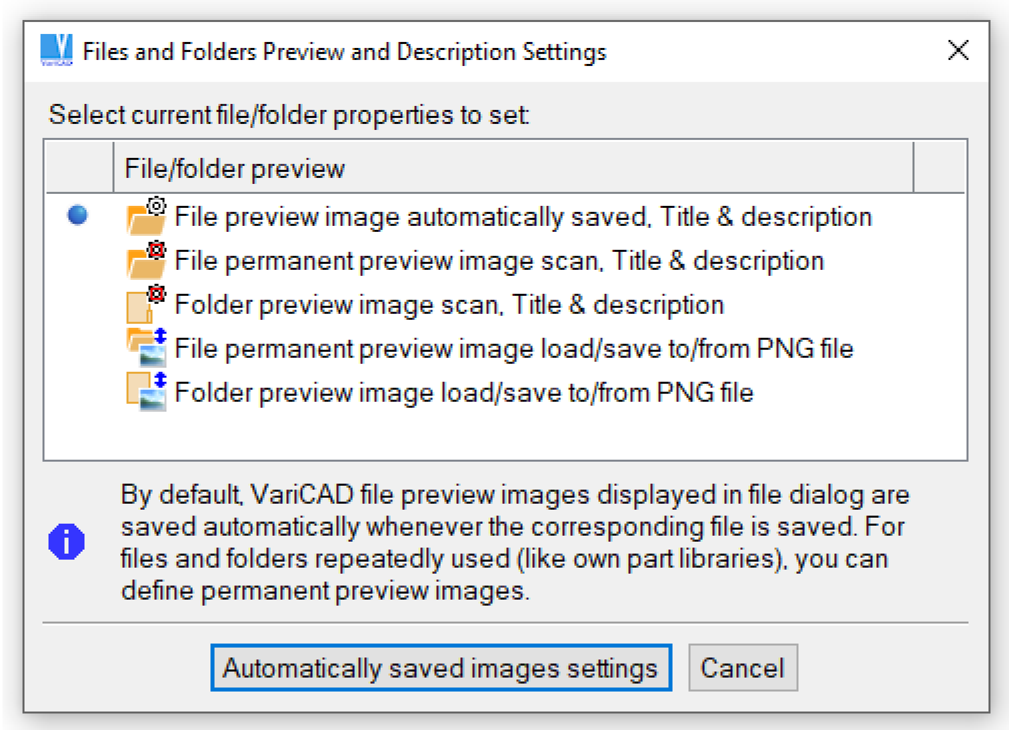
Recent files dialog is also accessible from File save dialog (command “Save as”). In this case, you can select a recently used file – it will be rewritten. Or, select a folder. The file to be saved is saved into corresponding folder (directory).



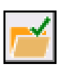
Recent files dialog, right-click options

File or Folder Preview Definition

While saved, a file stores also bitmap image which is later displayed in file dialog preview. You have multiple possibilities to define how a preview image is created.



Files or folders preview settings



File/Folder Preview Definition - FDP

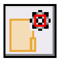
Following options are:

File preview is created automatically

File preview image is saved automatically, whenever a file is saved. The image content is the file 3D part. Here, you can define whether the image is created from currently used angle of view, or from predefined view (like front view, top view, user defined view…). File title and detailed description can be defined, too.

File preview image is permanent

Contrary to previous option, the image remains the same, independent on current file content. This option is useful especially for repeatedly inserted solids, if they are not changed anymore. The image defined here is scanned from display, under selected view and optionally, using enhanced rendering. File title and detailed description can be defined, too.

 Folder preview image

You can define preview image also for current folder. The method is the same as for permanent file preview image definition. Also, folder title and detailed description can be defined. Folder preview data are stored in file type *.dwb_fpv. Filename is the folder name, the file is located in the parent folder (one level up, where the corresponding folder belongs to).

 File preview permanent image load/save

If a permanent file preview image is defined, you can save it into PNG format. Then, you can edit the image outside VariCAD and load it back. Or, you can load PNG image completely created outside VariCAD (for example, a logo). It is necessary to save/load a pair of images. Images dimensions are 400x250 and 800x500. The larger is used for 4k resolution.

 Folder preview image load/save

This option is the same as described above. Images are used for folder preview.

Automatically saved images settings

For automatically saved previews, you can select whether a current zoom is used or “Zoom all” is preferred. Also, one color for entire solid vs. multiple colors (if used) can be selected. In settings described above, angle of view is defined, but not zoom vs. entire view. Settings above are related for each file or folder individually, settings here are used for all files.

VariCAD files created in versions up to 2018 do not contain preview images or titles and descriptions. Previews are displayed only as wire-framed 3D scene.

Backing up Your Files

Backup saves 2D/3D files after a specified number of changes. If the session ends unintentionally, you can recover your backup data the next time you use VariCAD. To turn safety backups on/off, run “CFG” command.

Working with Multiple Files

Changing Active File

To change the current (active) file, you can always use pull-down menu “Windows” and select the file to be active from the list. The list of open files is limited to 10 items. If the number of open files is greater, use the following function instead:

 **Windows – WIN, Ctrl + 3**

This function offers you a clearly arranged list of the open files. You can save or close any selected file or activate selected file from the list.

Previous Document Window – SWD, Ctrl + TAB

Activates a previously active file. Repeatedly used, this function allows you to easily switch between the two files.

Save All Changed – SVALL

Saves all changed open files to VariCAD native format. If the file is created as new or copied to a new document window, you must always define its real name. If the file is imported from another format, you must confirm or redefine the file name. If the current files are all from the native format, no dialog is displayed.

New Document from Current Document - NDW

Creates a new file, copies all objects from the current file and activates this file.

You can set working with multiple files from command “CFG”. Available options are:

- Whether the last open files are open automatically in the next session startup
- How the 2D objects are inserted from another file
- How the 2D objects are inserted from clipboard

Copy and Paste

VariCAD works with separate clipboards for 2D objects and for 3D objects. During work with VariCAD, you can store objects to a corresponding clipboard and whenever insert them to any open file.

Copy – CPY, Ctrl + C

Stores selected objects to the clipboard.

Paste – PAS, Ctrl + V

Insert objects from the clipboard to the current file.

Cut – to Clipboard – CCUT, Ctrl + X

Cut (delete) and put objects to clipboard (Copy)

Switching between 2D and 3D

Switch to 2D - 2D, Alt + 2

Switch to 3D - 3D, Alt + 3

You can switch between 2D and 3D at any time. To switch, you can:

- Click the 2D or 3D icon
- Use hotkey Alt + 2 to switch to 2D, and Alt + 3 to switch to 3D.

Switching between 2D and 3D also changes the available toolbars and menus. There is no direct link between 2D and 3D data, but you can update 2D views after making changes in 3D by using *3D View Exports (page 262) (page 154)*.

Sketching - 2D Drawing in 3D Planes, Alt + S

Sketching is used for creation of contours, later extruded, rotated or lofted into 3D space. Sketching is used also for editing of those contours of existing solids. To start sketching of new contours, click axes displayed in lower left corner of VariCAD 3D window. See *Sketching (page 169) (page 154)*.

Undo and Redo

Undo – UND, Ctrl + Z

Redo – RED, Ctrl + Y

When creating 2D or 3D data, you can use Undo to return backward step-by-step to previous states of your 2D/3D objects. You can also return to where you began work, or to the point at which the file was loaded. Once Undo has been used, Redo can be used to step forward. Undo/Redo history is separate for 2D and 3D objects. When working with assembly connections, this history is lost after parts are reloaded into the assembly after changes have been made. If this occurs, you will receive a message informing you of the problem.

Dragging Objects

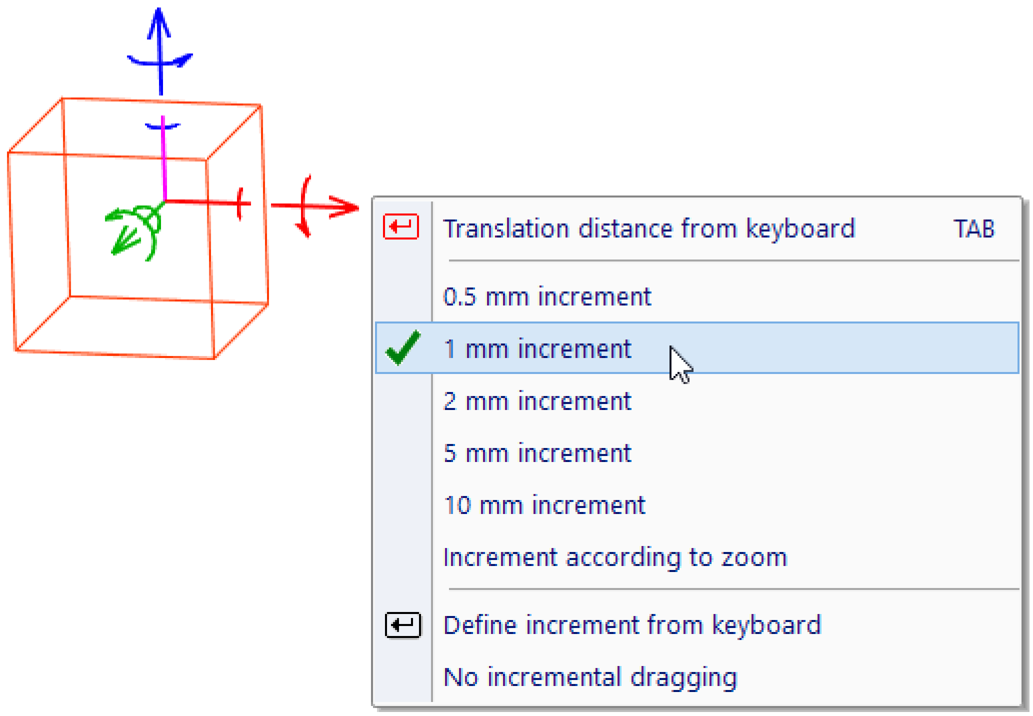
Many 2D functions and some methods of 3D transformation use dragging. In command “CFG”, you can choose between two methods of dragging. In both methods, the cursor defines the position of the reference point or insertion point.

- Dragging without clicking - position change is defined by cursor movement. If the cursor approaches snap points (such as endpoints), the reference point will “stick” to this point until the cursor is moved by at least half the aperture size. Left click ends the dragging movement.
- Clicking and dragging - objects are moved while the left button is pressed. If this button is not pressed, you can define any location simply by clicking on it. Right click or pressing Enter ends the dragging movement. This option is not recommended – it is obsolete (legacy) option.

Dragging Increments

Dragging increments can be always set or turned off during objects dragging. To do so, right-click mouse button. Select dragging increment from menu. Also, you can select input of exact location from keyboard.

Dragging increments are defined as fixed length increments. However, you may also select dragging increments changed according to zoom. In such case, the larger zoom is, the smaller increment is used.



Example of dragging increments settings, in 3D transformation

Turning off Detection Temporarily

Objects are detected by cursor automatically. If you drag selected objects and if you move cursor over other 2D lines or 3D solids, location is changed according to detection. In 3D dragging, objects may jump unexpectedly, because the detected location is projected to dragging vector. In 2D or in 3D, you may need to select a location regardless the other objects. If dragging increment is used, an exact distance is obtained by cursor movement.

Automatic objects detection can be temporarily turned off by following methods:

- Press and hold F1 while moving cursor. This turns detection off only for dragging.
- Hold left-mouse button pressed, Then, dragging is finished when you release the button. Again, this turns detection off only for dragging.
- Press and hold F2 while moving cursor. This turns detection off always – not only for 2D or 3D location, but also for detection of any objects.

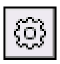
Run command “CFG” if you need to manage how detection is temporarily disabled.

Chapter 6. System Settings (Preferences)

All system settings are available in the Tools menu. If you change 2D drawing parameters such as text height or dimension style, the settings are saved when the session is finished. If you change system parameters such as colors, warning sounds, or file backup, the settings are saved immediately.

In most cases, new settings take effect immediately. There are a few settings, such as OpenGL settings, that do not take effect until the VariCAD session is finished. In these cases, you are informed by a message. Mostly, VariCAD offers automatic restart so the change also works instantly, if restart is accepted.

This section describes a few functions for general system settings. Functions used to manipulate settings are described in greater detail in other sections of this help system.

 **System Settings - CFG**

This command allows you to perform most of system settings. There are sections of general settings, settings of 2D and settings of 3D. Settings of files input or output (like configuration of DWG input etc.) are also available in file-dialogue window, under button File Options.

General settings available in command CFG:

- Pathnames of VariCAD files
- Working with multiple documents (files)
- Automatic safety backups
- Icons size, dialog texts
- Multiple monitors settings
- Right-click pop-up menu, display of options available within commands
- Cursor coordinates, displaying of coordinates together with cursor
- Settings of mouse wheel and mouse buttons
- Space mouse settings
- Cursor increments settings (for dragging)
- Automatic detection, dragging and stretching
- Settings of step back (undo)
- Configuration of keyboard shortcuts
- Values input settings
- Selection of objects between commands
- Theme settings (appearance of panels, buttons, frames etc.)
- Sound settings
- Warning messages settings

- Colors and palettes (available also from separate commands)
- OpenGL settings
- Performance and usage of Multi-threading
- BOM mask, solid attributes and title blocks
- List of materials settings
- Library objects selection settings

2D settings available in command CFG:

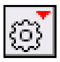
- Display of 2D objects
- Creation of 2D exporting 3D views
- Temporary construction lines
- Options of DWG/DXF format input or output
- Dimensions behavior (not style of dimensions)
- Printed lines thickness and color, print task names, printing to files
- Selection of 2D objects
- Border of 2D format (not sheet border style)
- Automatic layers switching

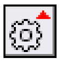
3D settings available in command CFG:

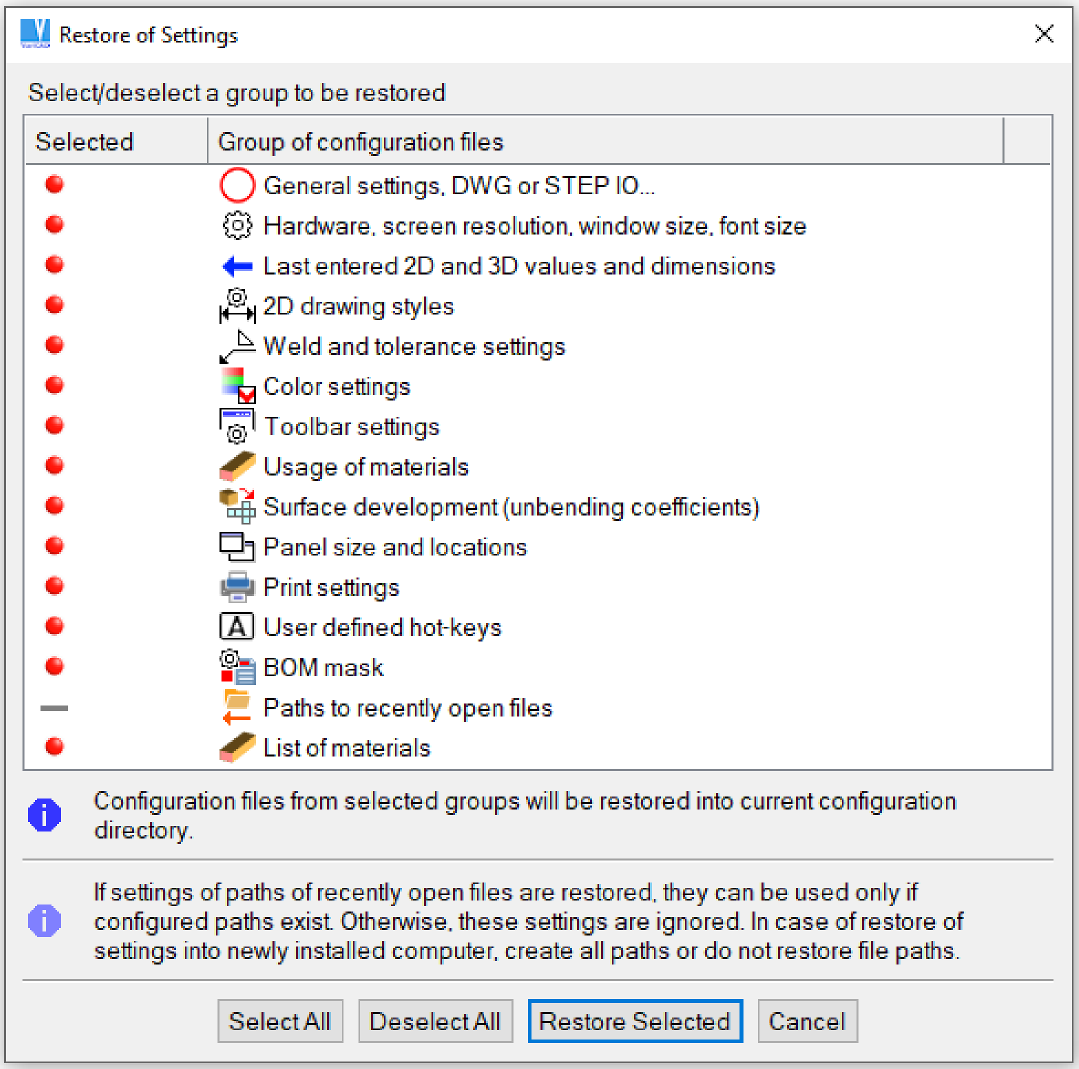
- Selection of solids
- Location of solids
- Definition of solid insertion point (for solids created from profiles)
- Surface shading and reflectivity
- Predefined 3D view (initial view for new file)
- Animation of view changes
- View rotation by arrow keys
- Assembly links settings
- Insertion into assembly settings
- Settings of format of calculation results
- Volume, mass and moment of inertia settings
- Options of STEP format input or output
- Options of STL format output

Backing up Your Configuration

You can create a backup of all your configuration files. The configuration is stored in one file. When you need to restore the configuration, you can select the settings you need to renew. Typical usage of this feature is to save and restore the working environment, if you need to reinstall operating system or if you migrate into another workstation. Also, you can use this feature if you want to share some settings with other users (like 2D drawing and dimensioning styles commonly used in entire company). Save/Restore of configuration is available from pull-down menu Tools or from following commands:

 **Create Backup of All Settings – CBS**

 **Restore Settings from Backup – RBS**



Restore working environment

Colors

Colors - COL

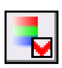
All colors used by VariCAD can be modified, such as colors of 2D or 3D objects, colors of 2D drawing in 3D, colors used for auxiliary images, and highlighting colors. You can save a modified color scheme as a new palette. There are predefined palettes as well.

The following colors can be set:

- Colors for 2D objects. You can modify color 1-9 for 2D objects. You can also set colors for auxiliary objects, the grid or construction lines used in 2D.
- Colors for 3D objects. You can modify colors 1-32 for 3D objects. Using this option, you can also set colors for auxiliary objects used in 3D, a color for the background and a type of the cursor used for objects selection.
- Colors for 2D drawing in 3D. You can modify all colors used for 2D drawing in 3D (this drawing is used, when you define or edit a profile for extrusion, rotation etc.).

Except of 2D colors, all settings allow you interactive changes of a selected color. You can change a selected color’s coordinate dynamically, dragging a scrollbar associated with the red, green or blue color coordinate. A pattern drawing is displayed and you can visually check changes of the color. For rough settings, you can use a standard color dialogue.

Important note: If you change the background color, change, or at least, check the color of the crosshair cursor and other colors, which may be inexpressive on the changed background.



Palettes - PAL

This function allows you to define a new color scheme (palette), rewrite an existing palette or select a defined palette as current. If you created your own color settings previously and if you want to select a new palette, the current settings will be rewritten. We recommend saving the current settings as a new palette, if you want to use them again in the future.

Chapter 7. 2D Drawing

Displaying the 2D Drawing Area

To change the view, you can use the zoom functions or use the corresponding combinations of mouse buttons and keys.

Changing the View Using the Mouse and Keyboard

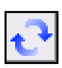
The following keyboard and mouse combinations can be used to manipulate the view:

- Zoom – using mouse wheel
- Zoom - Shift + left mouse button, or right then center mouse buttons. Moving the cursor up enlarges the objects; moving down shrinks them.
- Pan – press mouse wheel or middle mouse button, if it is separate. Then move the cursor.
- Pan - Ctrl + left mouse button or center then left mouse buttons. Moving the cursor shifts the view.

Display Functions

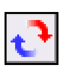
All functions controlling the display are available in the View menu or in corresponding icons.

Rebuilding Functions



Redraw - F6

Quickly refreshes the 2D area.



Regenerate - REG

Regenerates and redraws all 2D objects.

Zoom Functions

You can change the size of the view by using the following zoom functions:

- Window - the view is defined by the two opposite corners of the desired view window.
- Zoom All - the view is sized so that all visible 2D objects will fit inside.
- Zoom Format - the view size is set according to current drawing format.
- Previous View - the display returns to the previous view.

By default, view change is animated. In command “CFG”, you can set animation properties or turn the animation off.

Saving Views

The Save View function enables you to save the current view for future use. To display a saved view, use the Restore View functions. You can save up to eight views. The Predefined Views toolbar makes it easy to switch between saved views. See also *Saving Views (page 157) (page 154)*.

2D Display Settings

2D Circle Display Settings

Settings of 2D objects displaying are available in command “CFG”, in 2D section. It enables you to increase the number of segments used for drawing of 2D circles. When the number of segments is higher, circles will always appear smooth even when zoomed closely. A high number of segments may slow down drawing speed, especially on slow computers and extremely large 2D drawings.

Cursor Settings and Coordinate Listing

Sets the aperture size of 2D cursor.

Displayed Cursor Coordinates - DCC

Coordinates can be measured relative to a user-defined origin, or as DX, DY from the last point. Angle and radius from the last point are displayed automatically in most cases.

Types of 2D Objects

In 2D drawing you work with basic objects. These objects behave like individual entities when selected, and they can later be combined into blocks. The basic 2D objects are as follows:

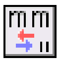
- Line - includes single lines defined by two points or multiple (chain) lines
- Spline – 2D interpolated NURBS, used for curves, includes ellipsis
- Arc - includes arcs and circles
- Point - used mostly as construction aid
- Arrow - similar to lines, with arrowheads at endpoints
- Hatch - for filling closed areas
- Text - single-text lines or a note containing up to ten text lines
- Symbol - can contain lines, arcs, arrows and/or texts
- Dimension - can contain lines, arcs, texts or arrows
- Axes – created as axes of a circle, or by 2 points, or as axes of rotation surface exported from 3D

For more information about creating blocks, see *2D Blocks (page 107)*.

2D Drawing Attributes - Units, Formats, Scale

File attributes can be defined when creating a new file (see *Creating and Opening VariCAD 2D/3D Files (page 14) (page 10)*). This section describes the functions used to change 2D drawing parameters such as drawing units, format and scale. Attribute functions can be found in the Tools menu.

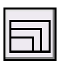
Units



Change Units - CHU

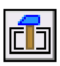
Change units in the current file by toggling between inches and millimeters. For example, an object defined to be 1” long will convert to 25.4 mm. Dimension text values do not change, nor do attributes of inserted mechanical parts. For example, Screw M10 will always have the same attributes, even if units are changed.

Format and Sheet Border



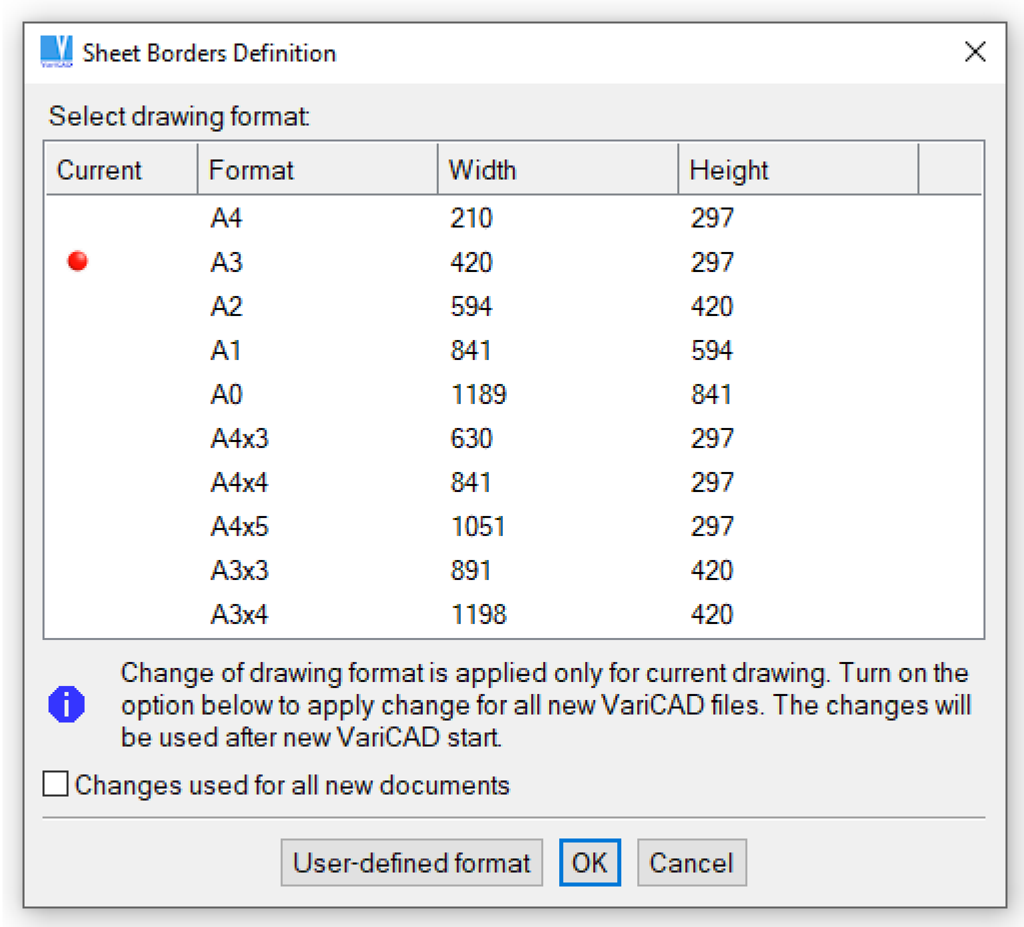
Drawing Format - FMT

Changes the current drawing format. The format controls the view area, sheet border, and print attributes.

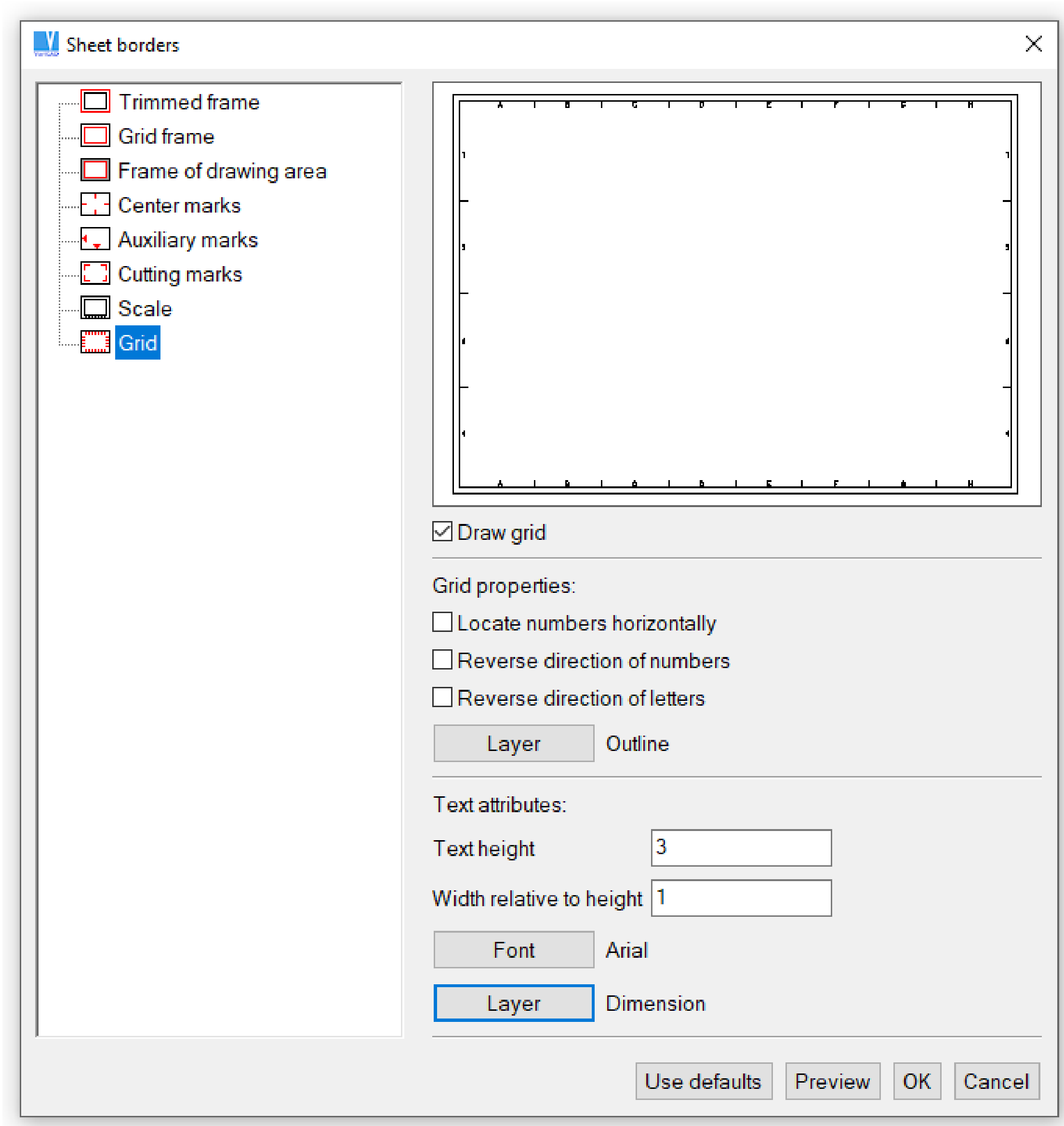


Sheet Borders Definitions - SBD

You can define custom sheet border formats, modify existing formats, and define the method of drawing sheet borders.



Formats window



Sheet Border window

Sheet Border - BOR

Creates a sheet border around the 2D drawing area. The lower left corner of border corresponds to the origin in global coordinates. Border width and length are determined by the drawing format. Border drawing method setting is defined in Units and Sheet Border.

Sheet Border in 2D Background

System draws a rectangle around the current sheet border, regarding the current drawing scale. The rectangle is drawn in the same style as an auxiliary grid. This format border is not printed and cannot be detected, it is only displayed. Contrary to the format border in the drawing background, you can insert a border created in a configured style from detectable and printable 2D lines. Such a border is a part of the current 2D drawing.

You can turn on/off drawing of the background sheet border in the command “CFG” (system settings). If turned on, the background sheet border is always drawn - it is not a part of a 2D drawing.

2D Drawing Scale

Drawing scale only affects 2D annotation objects such as text, dimensions, symbols and arrows. The scale affects the proportions of these objects. For example, with a 1:2 scale, a 100-mm line will print as 50 mm long. Text 3 mm high will print as 3 mm high. Changing the scale does not affect dimensions.

Change Drawing Scale - SCH

Changes the 2D drawing scale. The scale is defined when the file is created, and this function can be used to change the scale. All objects in the file remain unchanged. New annotation objects such as dimensions and text are created in different proportions. We recommend finalizing the drawing scale before starting to assign dimensions.

Attributes of 2D Objects

2D objects have the following attributes:

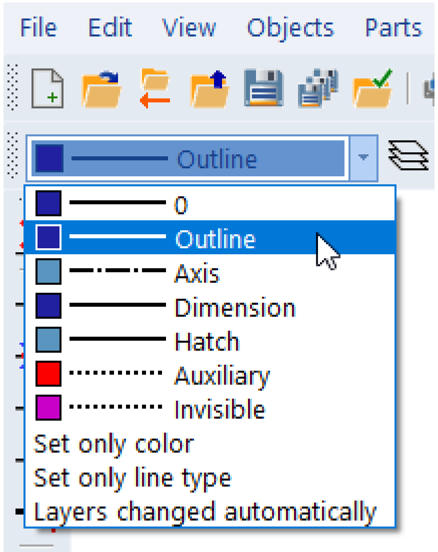
- Layer
- Color
- Line Type
- Visibility (blanked or unblanked)

Working with 2D Layers

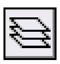
You can define up to 250 layers in each file. Each file contains one predefined layer named “0.” In 2D assembly, layers can be used to distinguish between separate details. For detail views or 3D view exports, layers should be used for distinguishing outlines, axes, dimensions, hatches etc.

Each layer is defined by name, color and line type. New objects are always created in the active layer. The active layer can be changed at any time, even during object creation. You can also change the current color or line type without changing the layer. For each object, its layer, color or line type can be changed at any time.

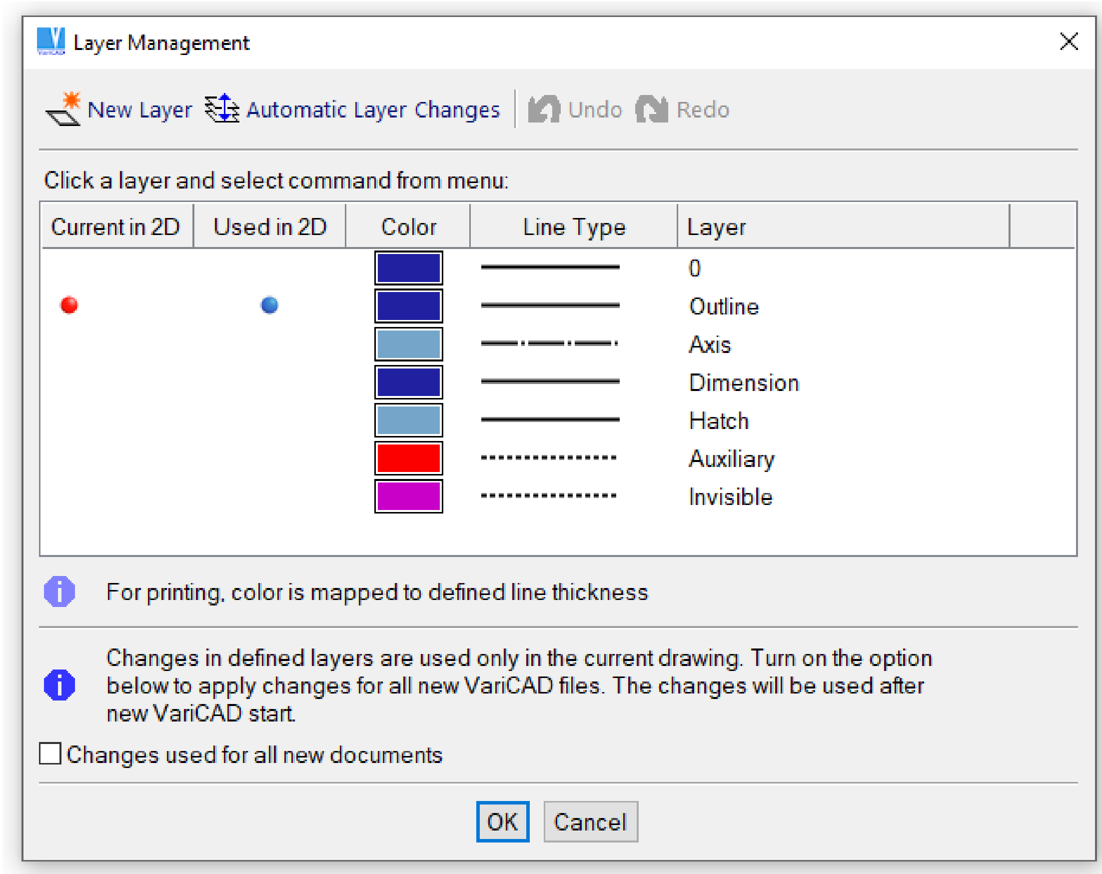
If objects from another file are inserted into the current file, objects from unknown layers are inserted into Layer 0.



Setting the active layer

 **Layers - LAY**

Creates new layers, and edits or deletes existing layers. You cannot delete the active layer, a layer containing objects, or Layer 0. The command allows you to manage automatic layer switching.



Layers window

Automatic Switching of Layers

Automatic layer switching can be set from the command CFG or LAY (see above). Automatic layer switching is useful for drawing 2D details. Layers are switched according to the executed function. Drawing functions like Line or Arc tools create objects automatically placed in Layer “outline.” Hatches are placed in “hatches;” dimensions are placed in “dimension.”

By default, automatic layer changing works with following layer order:

- The second layer is named “outline,” and is active during drawing functions
- The third layer is named “axis,” and is active while creating axes
- The fourth layer is named “dimension,” and is active while creating dimensions
- The fifth layer is named “hatches,” and is active while creating hatches

You can redefine the automatic layer switching. It is possible to select a single 2D drawing command or a group of 2D drawing commands and to assign a new layer automatically activated whenever the command is used.

 **Change Layer - MLA**

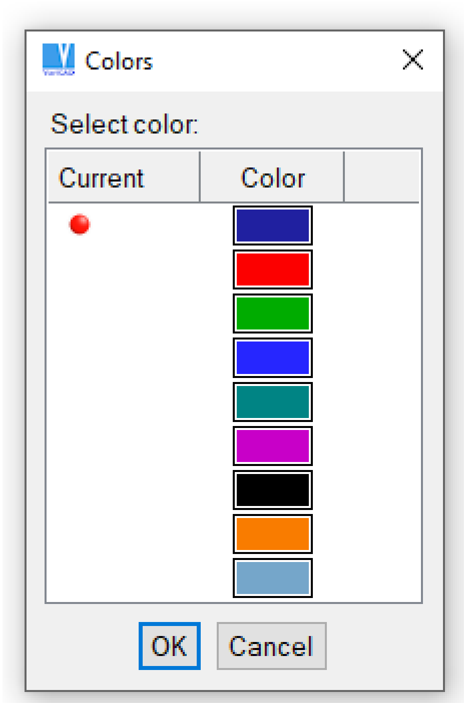
Changes the layer of selected objects to that of another object, or to a layer selected from the list of layers.

 **Highlight Layer - CHL**

Shows all objects on a specified layer, enabling you to check that the layer contains the correct objects.

2D Object Colors

For 2D objects, there are nine colors you can use. For 3D objects, there are 32 colors. Following dialog window contains list of colors available for 2D objects. The displayed color set is used for light background.



Line colors used in 2D drawing, for light background

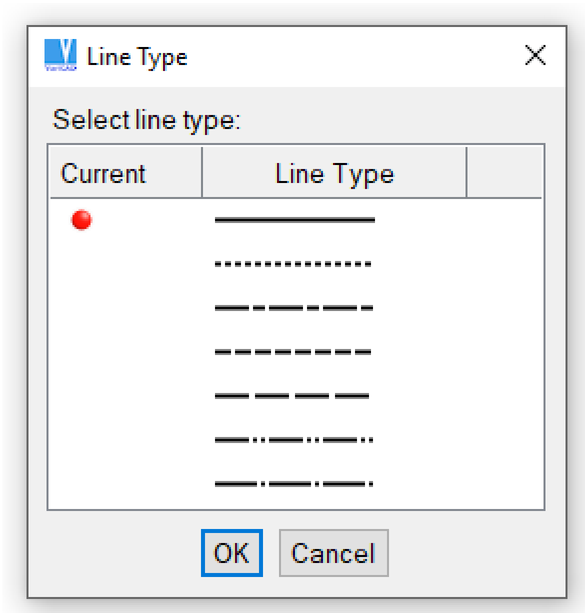
For printing, line thickness is set according to color number. For color printers, you can map colors to other colors.

 **Change Color - MPE**

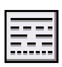
Changes the color of selected objects.

Line Types

For 2D objects there are 7 line-types available. Following dialog window contains list of line styles available for line style selection.



Line styles used in 2D drawing

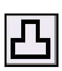
 **Change Line Type - MLT**

Changes the line type of selected objects.

Visibility of 2D Objects

 **Blank 2D Objects - BLA, Ctrl + B**

Makes selected objects invisible. You can blank temporarily redundant objects, or objects that cannot be selected. Blanked objects are not printed, nor are they selected in selection windows.

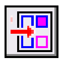
 **Unblank 2D Objects - UBL, Ctrl + U**

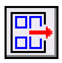
Unblanks objects that were blanked, either in the entire drawing or in a specified area. You can also unblank individual objects, or all objects on a specified layer.

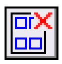
Work Sets

Work sets can be used to hold 2D objects. Work sets are useful when you want to delete or translate an entire set of objects. When inserting 2D objects from another file, you can place all new inserted objects into their own work set.

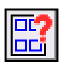
The following functions are used to manage work sets:

 **Add to Work Set - ATW**

 **Delete from Work Set - RFW**

 **Clear Work Set - CLW**

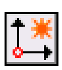
Deletes all objects from a work set

 **Highlight Work Set - CHW**

Helps you verify objects in the work set

2D Coordinate System

VariCAD uses two types of coordinate systems. The absolute coordinate system has its origin at the lower left corner of the drawing area. The user-defined coordinate system is, by default, identical to the absolute system until a new origin is defined. The user-defined coordinate system is unique for each file. When 2D coordinates are used, they are always relative to user-defined origin.

 **User Origin - UCO**

Locates the user-defined origin or resets to the absolute origin.

More convenient method to change or reset the user-defined origin is to use *Increment Cursor Mode (page 41)* and select the change from dialog panel.

2D Drawing Aids

The following aids are available to help you while creating 2D objects:

- Grid
- Construction lines
- Temporary construction (leading) lines
- Transient leading lines
- Ortho mode

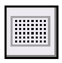
- Increment cursor movements

Ortho mode or increment cursor movement is displayed at right end of status bar. Also, corresponding icons in tool-bar are checked.

Grid

The orthogonal grid provides better drawing orientation and enables snapping to grid points. The grid is displayed in two levels of brightness. When the grid density is too high to display, the grid is automatically turned off. Snap distances can be smaller than the grid spacing.

The grid can be especially useful when translating entire sections of a 2D drawing or inserting 3D view exports. Grid snapping is available even when the grid is not displayed. Entered values are rounded to the nearest multiple of the snap distance. If you want to use the same grid settings in new documents save the current settings as default.

 **Grid – GRI, Ctrl + G**

Sets the grid spacing or turns the grid on or off.

Construction Lines

Construction lines are “auxiliary” or temporary lines, independent of any other objects. You can create individual constructions lines or a mesh of them. Objects or other construction lines can be placed at the intersection points of other constructions lines – either intersection between two construction lines, or intersection between a construction line and a line, arc or NURBS curve.

Construction lines can be created as vertical, horizontal or angular. You can predefine two angles for angular lines. If you create a construction line or if you delete one or multiple lines, the step can be undone or redone again – similarly, as for any other 2D or 3D object.

Construction line functions are available from the Construction Lines toolbar, and from the Objects / Drawing Aids menu.

Creating Construction Lines

You can create construction lines as:

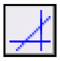
- Single lines passing through a selected point
- Groups of lines that have a specified distance from a specified origin (a negative distance creates new construction lines in the opposite direction)
- Groups of lines that are offset at a specified distance from the previous line (a negative distance creates construction lines in the opposite direction)
- Single lines tangent to a selected circle or arc

Deleting Construction Lines

You can delete all construction lines, delete a selected line, or delete all lines by type (all horizontal, all vertical or all angular).

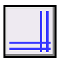
Creating Multiple Construction Lines

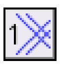
Apart of creation of individual lines or group of lines, you can create multiple construction lines – as the most convenient method.

 **Create Multiple Construction Lines - CCL**

This function creates one or multiple construction lines of all types – horizontal, vertical or angular.

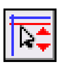
Following options are available:

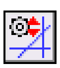
 Create Horizontal/Vertical Construction Lines – select a location and then direction the new construction line (lines) will be created at. The line or lines are created according to current selections – at a distance from the selected location, at a distance from the previous line or they are dragged by cursor

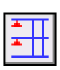


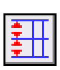
Create Angular Construction Lines – under predefined first angle, similarly as horizontal/vertical lines

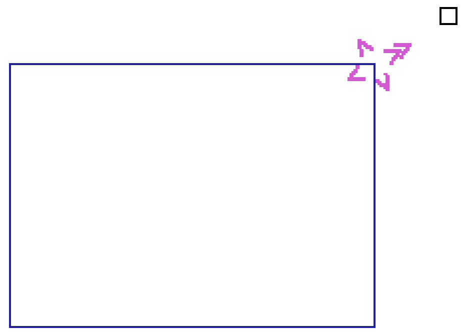
Create Angular Construction Lines – under predefined second angle, similarly as horizontal/vertical lines

 Drag Construction Line by Cursor – if this option selected, a new construction line is dragged by cursor until you select a location. By default, increment mode of dragging is used – for instance, the new line is dragged in increments of 1 millimeter. Distance from location is displayed near the cursor. If this option is turned off, the distance of a new construction line is defined by keyboard input or measurement.

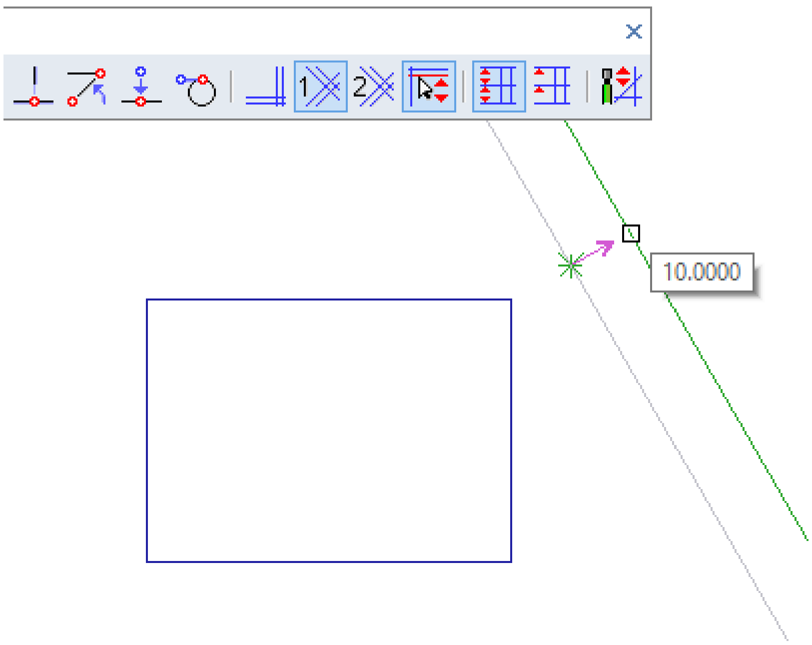
 Set Dragging Increment or Angles – sets increment of dragging, or predefined first and second angle of angular construction lines.

 Measure Distance from Selected Point

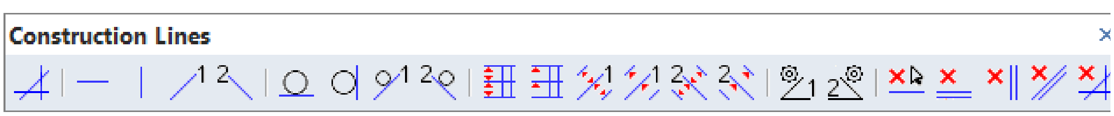
 Measure Distance as Offset from Previous Line



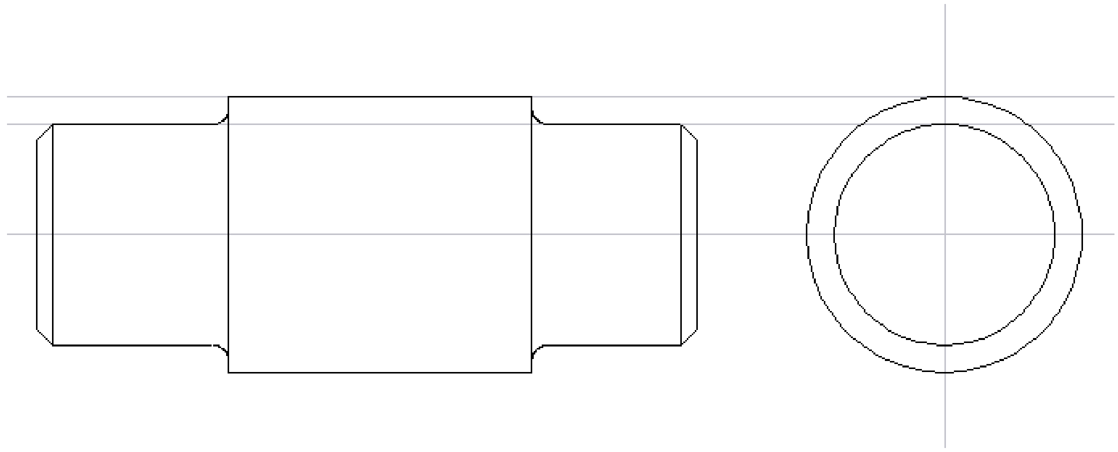
Selecting a direction of a new construction line



A construction line dragged by cursor, according to the previous selection



Construction Lines toolbar



Example of construction lines used to create a side view

Temporary Construction Lines

Temporary construction lines (leading lines) are automatically created at the last defined location and in these situations:

- Drawing of 2D lines, poly-lines or multi-lines
- Creation of temporary boundaries for extensions, or temporary cutting lines
- During dragging of objects, at a location of dragging point definition
- During stretching, at a location of dragging (reference) point definition

By default, these temporary construction lines are active in 2D drawing in 3D (in sketching). You can stick cursor at these construction lines and easily follow horizontal or vertical direction. In 3D sketching, you may not recognize directions of X or Y axis well, because of view rotation.

Transient Construction Lines

Transient construction line or lines are temporarily displayed, if X or Y cursor coordinate approaches:

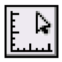
- Location x=0, y=0 (coordinates origin)
- Marked point, like end-point or mid-point
- Poly-line start point.

Transient lines disappear, if cursor moves away from them more than defined distance. On the other side, you can follow a transient line by cursor movement until you reach an intersection of the transient line and a 2D object.

To turn on or off temporary or transient construction lines for sketching or 2D mode, run command "CFG", or select *Ortho Modes, Leading Lines (page 44)*

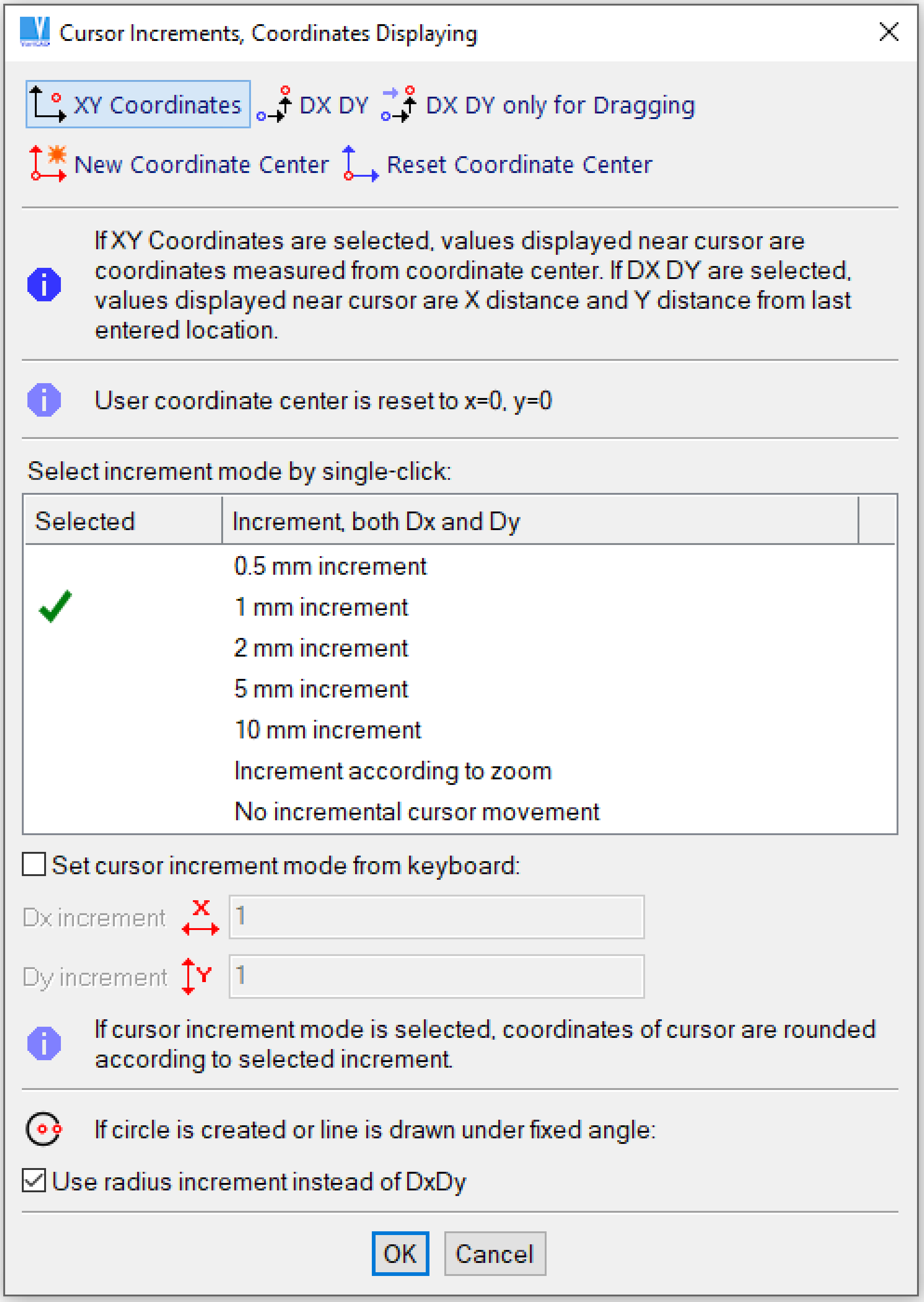
Increment Cursor Mode

When not in Increment mode, the cursor moves smoothly and defined locations are based on display resolution. When using Increment mode, cursor locations are rounded to the nearest multiple of the increment distance. The square or arrow cursor movements are still smooth; only the resulting locations are rounded. Crosshair cursor movements “jump” in the defined steps. Increment mode is indicated in the Status Bar. Increment mode is especially useful when used in conjunction with a user-defined origin.



Increment Cursor Mode - STP, F9

Turns Increment mode on or off, or sets the XY increments of cursor movement. This command also allows you to set or reset 2D coordinate origin.



Setting cursor increments or coordinates origin

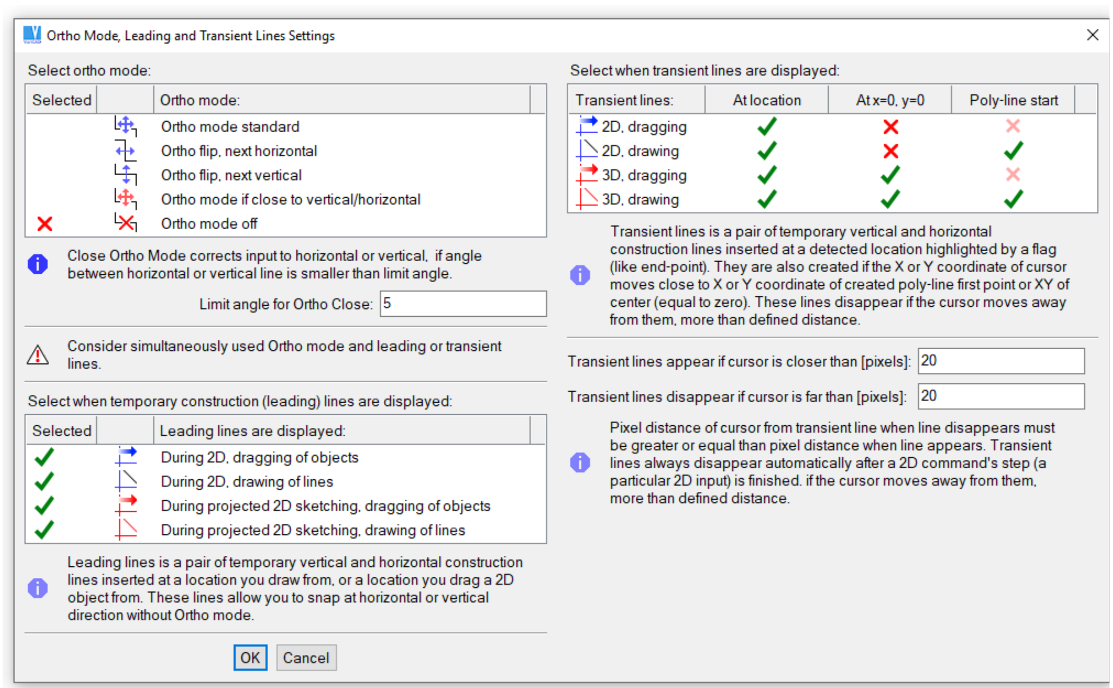
Ortho Mode

Ortho mode is an obsolete method. It is available as a legacy option. We recommend to work with temporary and transient construction lines rather than with Ortho mode. Besides, Ortho mode may be inconvenient in stretching, where you may have drawing plane displayed under various view angle.

In Ortho mode, created lines are always horizontal or vertical, according to current position of cursor. You can use Alternating Ortho mode, in which lines alternate between horizontal and vertical, regardless of cursor position. Ortho mode is indicated in the Status bar. Also, a corresponding icon in tool-bar is checked.

Ortho Modes, Leading Lines

This command sets Ortho mode, turns off Ortho mode or configures temporary construction lines (leading lines).



Ortho modes and leading lines settings.

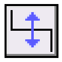
Following commands manage various Ortho modes individually:

Ortho On - F11

Turns on Ortho mode.

Ortho Alternating H/V

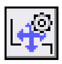
Turns on Ortho mode, alternating horizontal and vertical lines. The first line is horizontal.

 **Ortho Alternating V/H**

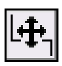
Turns on Ortho mode, alternating horizontal and vertical lines. The first line is vertical.

 **Ortho if Close to Vertical/Horizontal**

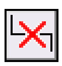
Ortho mode is used, if the angle of the current cursor location measured from the last input is close to horizontal or vertical direction. The angular limit can be set.

 **Set Close Angle for Ortho**

Sets the angular limit for the mode described above.

 **Ortho Modes – Shift + F11**

Opens pop-up menu with all Ortho mode possibilities.

 **Ortho Off**

Turns off Ortho mode.

Selecting 2D Objects

While working with 2D objects, you almost always need to select other objects. In a typical function, you must select one or multiple objects, finish the selection, and then the function processes the selected set. For example, selection is used when deleting objects, translating objects, changing object color, etc. Temporary toolbars provide selection options.

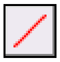
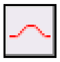
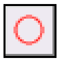
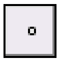
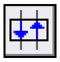
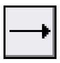
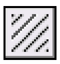
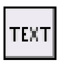
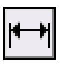
Methods of Selecting

The most direct way to include objects in the selection set is to left-click on them. Objects are selected if they are within the cursor aperture, and selected objects are highlighted in a different color.

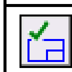
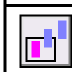
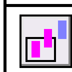
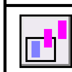
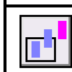
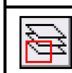
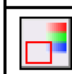
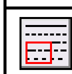
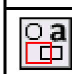
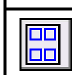
A temporary toolbar appears during object selection, which provides additional selection options. All options are also available on the Select menu. You can select single objects, or groups of objects that share attributes such as a particular color. You can select groups of objects from the entire drawing, or you can use selection windows. You can also access selection options by entering the relevant command keys (which are not case-sensitive). When using commands, the desired object must already lie within the cursor aperture.

Selecting Types of Objects

These options enable you to select single objects of a certain type that are found within the cursor aperture. When using these selection options, automatic detection is irrelevant.

Icon	Key	Use
	L	Selects a line
	Q	Selects a spline
	A	Selects an arc or circle
	P	Selects a point
	S	Selects a symbol
	Shift + 6 (^)	Selects an arrow
	C	Selects a hatch
	N	Selects a text
	D	Selects a dimension

Selecting Groups of Objects

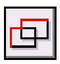
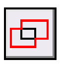
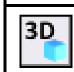
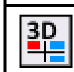
Icon	Key	Use
	N/A	Selects all objects
	R	Selects objects completely inside the selection window
	I	Selects objects completely or partially inside the selection window
	U	Selects objects completely or partially outside the selection window
	O	Selects objects completely outside the selection window
	V	Selects objects on a specified layer
	B	Selects objects of a specified color
	Y	Selects objects of a specified line type
	T	Selects a group of objects of a specified type
	1…8	Selects objects from work set 1 - 8

Using Selection Windows

Run command CFG, section 2D and sub-section “2D Object Selection Settings”. Here you can determine when and how selection windows will be used. If, during a selection, you click in the drawing and nothing is selected, you can set the system behavior to do one of the following:

- Display a warning message
- Start a selection window whose behavior must be determined
- Start a selection window whose behavior is defined by which corner the window starts (upper left, lower right, etc.)

Selecting Objects Related to 3D Solids

Icon	Key	Use
	E	Selects a profile to be used to create a solid. The profile is selected segment by segment.
	F	Selects a profile to be used to create a solid. The profile is identified automatically.
	M	Selects all objects linked to a single 3D solid. These objects are created by exporting a 3D view.
	G	Selects all objects belonging to a 3D view export.

When you create a 2D documentation from 3D parts or assemblies, you may need to select entire view (Top view, Front view…) quite often. To do so, press and hold Ctrl and move cursor over 2D objects. All objects belonging to an exported view are detected. Left-click them to finish selection.

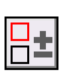
This way, you can select 2D lines, circles or arcs displaying corresponding 3D objects, and also dimensions, hatches or axes, if they are connected to 3D export.

If the view is once exported and you want to move it into a different location, it is necessary to select all objects from the view. Otherwise, connections to 3D may be lost.

Limited 2D Selections

Some functions, such as Fillet and Chamfer, require you to select a limited number of objects. In this case, there is no need to finish the selection group. If you need to select two segments that share a corner, you can select the segments individually or select the corner itself. To select by using the corner, click when the V symbol appears on the corner. See also *Creating Corners, Chamfers and Fillets (page 71)*. Prior to calling command, you can press and hold Shift, move cursor over a corner and when detected, right-click to select fillet or chamfer.

Deselecting Objects



To delete objects from a selection set, click Select/Deselect icon and use the normal methods to select objects - each selected object will return to its unselected state. Pressing X while selecting will also deselect objects.

Finishing the Selection

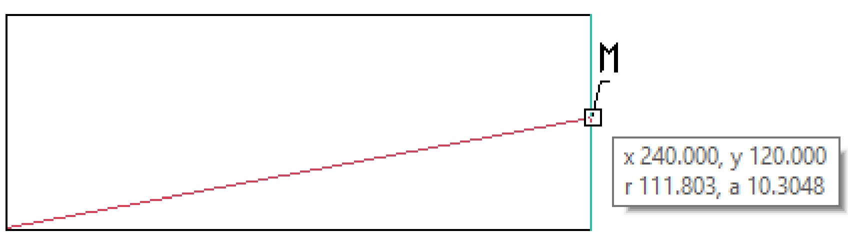
Press Enter or right-click to complete the selection set, or click the corresponding icon in a toolbar.

Selecting 2D Locations

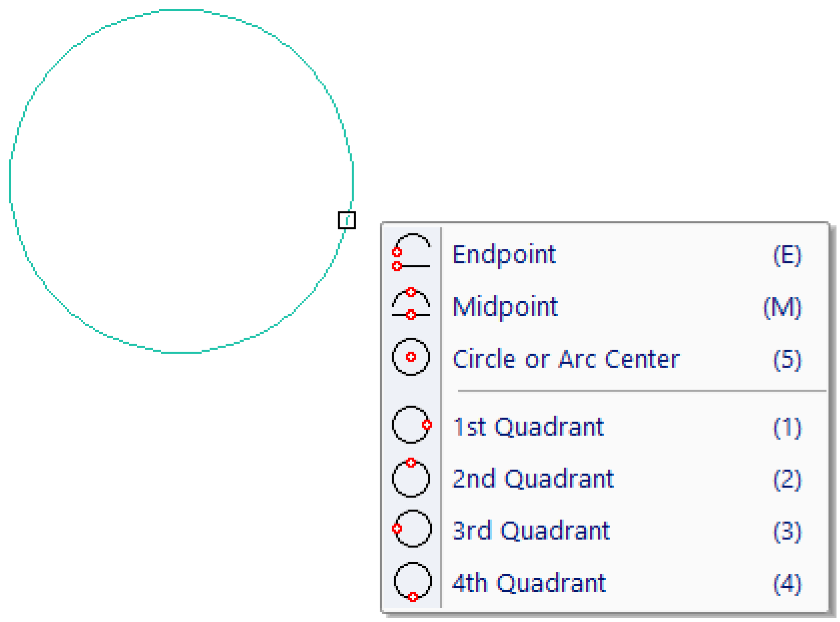
Most 2D objects require geometric input in the form of X, Y coordinates. VariCAD makes it easy to select certain significant locations relative to existing objects. To select a point on an object, move the cursor over the object so that it is highlighted. Clicking on the object will select the point closest to the cursor. If cursor approaches a snap point, such as an endpoint or midpoint, a symbol appears next to the cursor. Clicking when you see this symbol selects the point. The following letters indicate snap points:

Letter	Snap Point
E	Endpoint
M	Midpoint
5	Center of circle or arc
X	Intersection
S	Connection point of a symbol or block
1	0-deg point of a circle or arc
2	90-deg point of a circle or arc
3	180-deg point of a circle or arc
4	270-deg point of a circle or arc

You can also use icons, keys, and Snap menu items to specify snap points or other geometric locations. If you select a snap-mode by key pressing, the desired point or related object must already be within the cursor aperture. The Snap menu appears if you press simultaneously right and left mouse button while 2D location is defined.



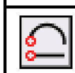
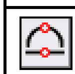
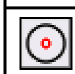
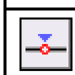
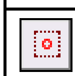
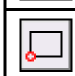
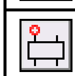
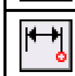
Automatic detection of a snap point

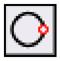
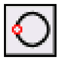
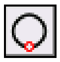
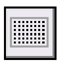
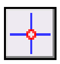


Pop-up menu with snap options – displayed after right-click of detected object.

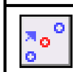
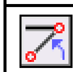
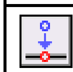
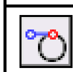
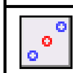
If you need temporarily turn off automatic detection during a location input, press and hold F2 - see *Turning off Detection Temporarily (page 22) (page 10).*

2D Snap Points

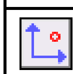
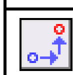
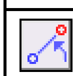
Icon	Key	Location
	E	Nearest endpoint
	M	Midpoint
	5	Center of an arc or circle
	O	Nearest point on an arc or line
	P	Point (must lie within cursor aperture)
	Z	Insertion point of a block or symbol
	S	Connection point or insertion point of a block or symbol
	N/A	Nearest dimensioned point

	N/A	Leader, text-side point
	1	0-deg point of a circle or arc
	2	90-deg point of a circle or arc
	3	180-deg point of a circle or arc
	4	270-deg point of a circle or arc
	F	Intersection of two segments (select both segments, can also find intersection of segment extensions)
	Spacebar	Nearest grid point
	C	Intersection of nearest construction lines

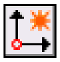
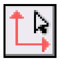
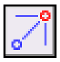
Combination of Location Points

Icon	Key	Location
	G	Defined distance from the nearest line endpoint
	A	Intersection of a selected object and a line created from the last point at a specified angle
	6	Intersection of a selected object and a line created from the last point, perpendicular to this object
	T	Tangent point on a selected object, directed from the last point
	B	Halfway between two defined points

Points Defined by Keyboard Input

Icon	Key	Location
	K	Enter X, Y coordinates
	D	Enter DX and DY from the last point
	R	Enter the distance and angle from the last point

Other Points and Functions

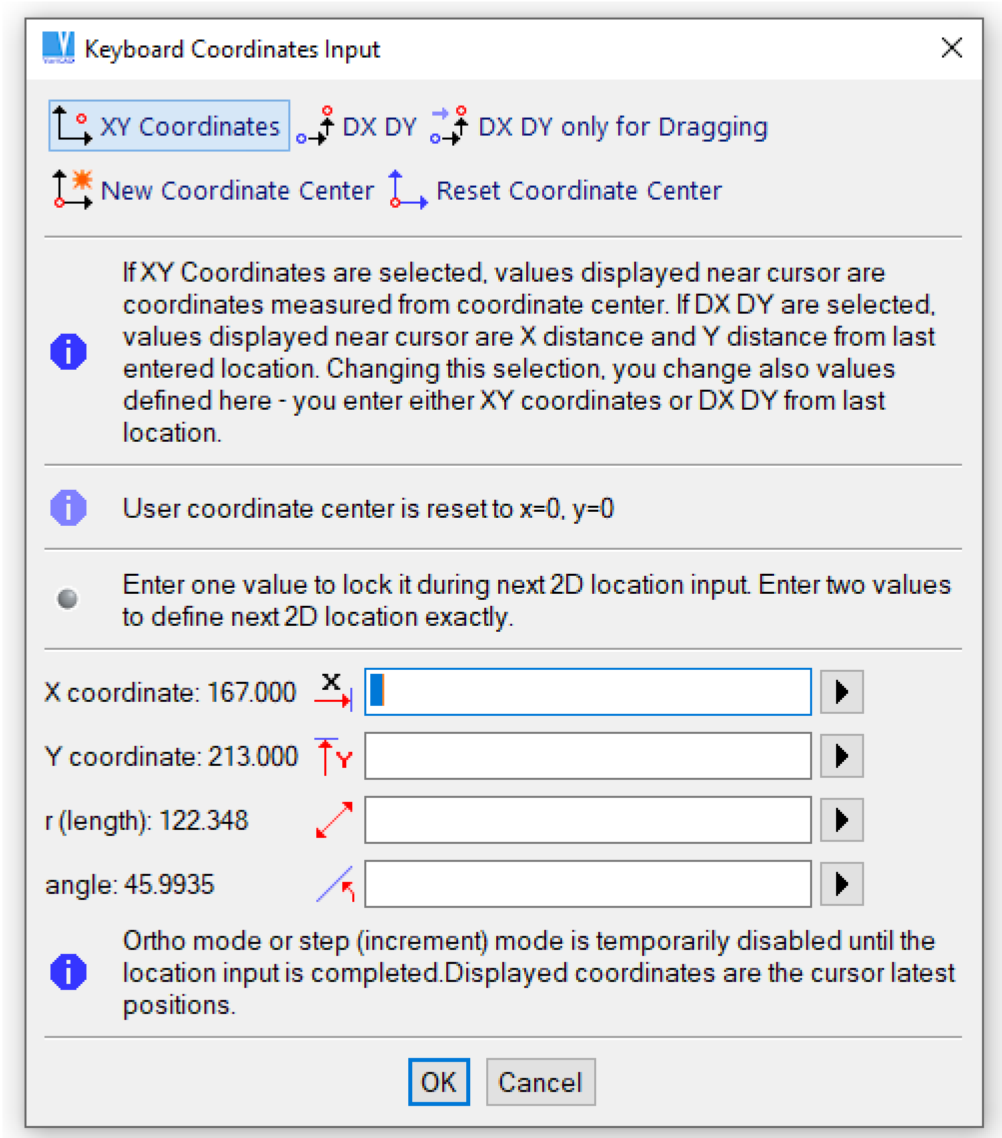
Icon	Key	Function
	W	Redefine the user origin
	I	Exact current cursor location
	TAB	Fixed length, angle or coordinate

Fixed Length, Angle or Coordinate

This method of 2D input allows you to use many convenient combinations. It is always available by pressing TAB. Then, a dialogue panel is open. If you continue pressing TAB, you change focus of input of X, Y, radius or angle value.

If you enter only one value, for instance X coordinate, next 2D input has fixed X coordinate. Or, if you enter only radius (length), next 2D input is always rounded to fixed distance from previous point. During drawing of line, you can create a fixed length segment.

If you enter two values, input is exactly defined. This allows you a combination of input of fixed coordinates, input of location in distance and under an angle etc.

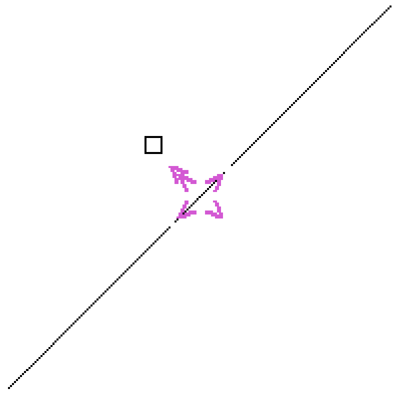


Keyboard coordinates input

Defining Angles and Directions

Some functions, such as symbol insertion or mechanical part insertion, allow you to determine the rotation angle of inserted objects relative to line segments. Angles are measured counterclockwise from the +X direction (to the right of the origin).

To define an angle according a line direction, click a line. Then, four arrow icons appear – two along line, opposite each other. Next two arrow icons are perpendicular to line, opposite each other. Move cursor around the center of icons. Arrow in direction close to cursor is highlighted. Left-click a desired location, the current direction of angle is used.



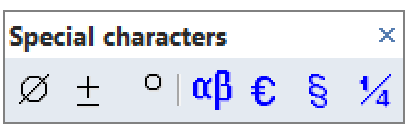
Angle input, measured from a line

Writing Special Characters

Special characters can be used during dimensioning or creation of 2D text lines. Whenever a text input is required, a toolbar containing special characters is available. Click a corresponding icon to enter a special character.

The following special characters are available:

- Diameter sign
- Degree sign
- Plus/minus sign
- Set of selected Greek letters
- Set of selected currency symbols
- Set of selected general symbols
- Set of selected numerical symbols



Special characters toolbar

Mathematic Expressions

If you need to specify a numeric value, you can enter a number or a mathematic expression. If either the expression or number contains errors or invalid characters, a warning message is displayed.

Mathematic expressions can contain the following operators:

- + (plus)
- - (minus)

- * (multiply)
- / (divide)
- ^ (exponent)

Numbers and variables can be written in brackets or parentheses with unlimited insertion levels.

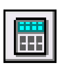
Mathematic expressions can also contain the following functions:

- sin, cos (sine, cosine)
- tan, atan (tangent, arc tangent)
- asin, acos (arc sine, arc cosine)
- log (decimal logarithm)
- ln (natural logarithm)
- exp (exponent of e)
- rtd (converts radians to decimal values)
- dtr (converts decimal values to radians)
- sqr (square)
- sqrt (square root)

You must enclose arguments in brackets or parentheses. Trigonometric function arguments are entered in degrees.

Example of correct expression: 1 + 2*sin (30) + 2*(2^2+sqrt (9))

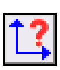
The result is 16.

 **Calculator - CAL, Shift + F9**

Enter and solve mathematic expressions.

Checking Objects, Distance, Angle and Coordinates

Checking functions for 2D objects are accessible from the Objects / Check menu or from the 2D Check toolbar.

 **2D Coordinates - COO**

Displays X, Y coordinates of a selected point.

 **2D Distance - DIS**

Displays the distance between two points.

 **Angle - ANG**

Displays the measurement of an angle. You can define an angle by following methods:

- By line direction
- By two points
- Between two lines
- By three points
- By tangent of an arc or a spline

Angles are always measured counterclockwise, and their values are displayed in degrees.

 **2D Object Information - ODT**

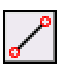
Displays information about a selected 2D object. Object type, layer, color, line type, and work set (if any) are always listed. Other displayed data depends on the type of selected object.

Drawing 2D Objects

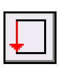
This section describes the various 2D objects you can create. See *Selecting 2D Locations (page 48)* for details on defining and selecting locations.

Drawing Lines

The following functions enable you to create lines and objects created from lines, such as rectangles and polygons:

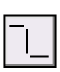
 **Line – LIN, Ctrl + L**

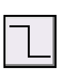
With this function, you can create a single line, a multi-segmented line, or a freehand curve. While creating lines, the following additional options are available:

 Close - joins the last point and the first point to close the line.

 Freehand - creates a freehand curve by holding the mouse button.

 Endpoints – the default mode, in which each mouse click defines a segment endpoint. This option cancels the freehand mode.

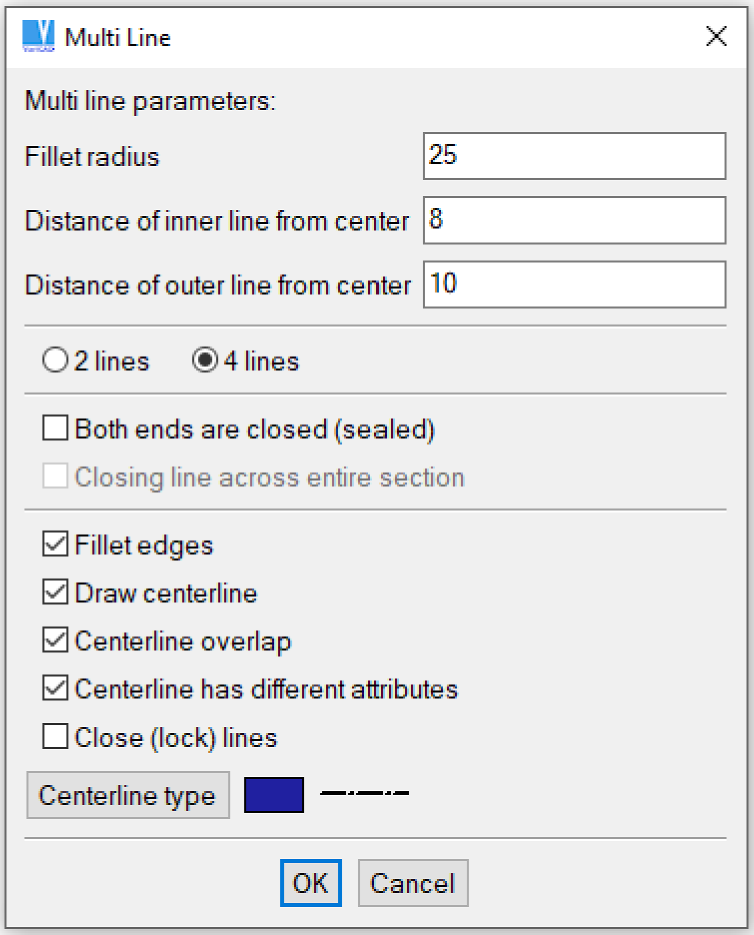
 Separate Objects - creates each line segment as a separate object.

 One Object - creates one object that contains all the line segments.

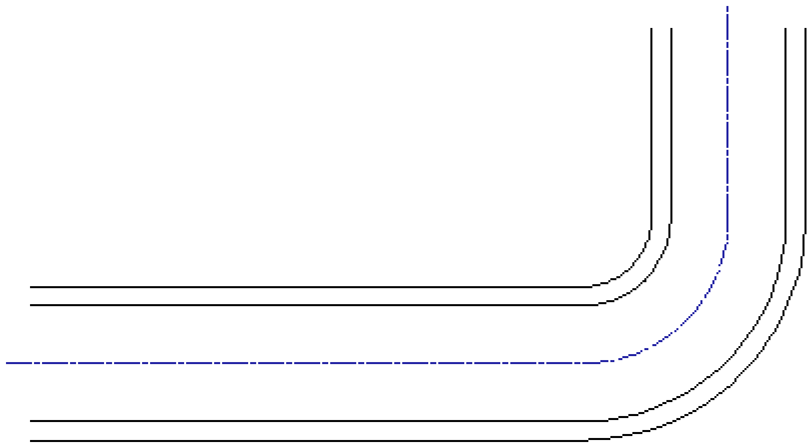
Multi Line - MLL

A multi-line is a group of parallel lines that acts as one line. Before creating multi lines, you can set parameters such as number of lines, line distance from the center, and rounding.

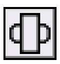
Multi Line Attributes - change properties during multi line creation.



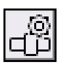
Multiple lines options

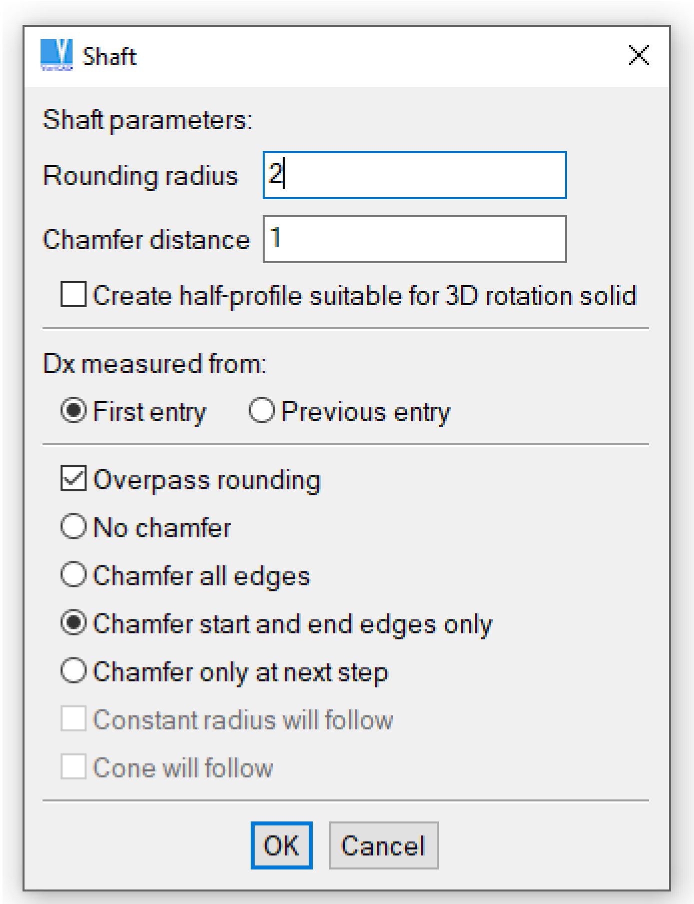


Example of multi lines created within one function

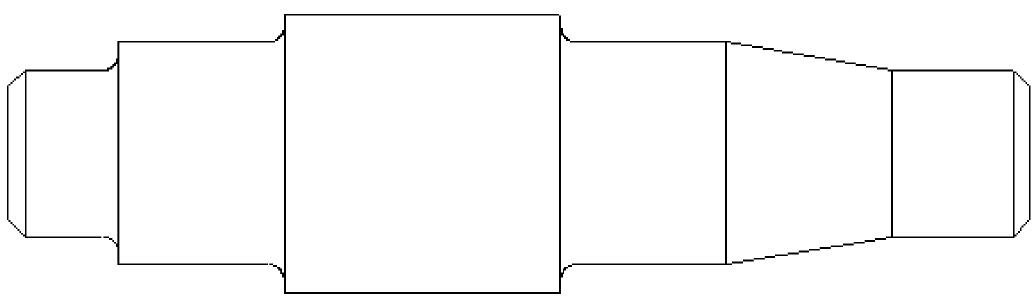
 **Shaft - SHAFT**

Creates shafts or other symmetrical 2D objects. Before creating shafts, you can define coordinate display, rounding, chamfering or creation of conical or cylindrical parts. When drawing shafts, coordinates are displayed as DX, DY. You can choose whether DX is measured from the first shaft point (total length from beginning) or from the last point (length of the created segment).

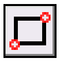
 Shaft Attributes - change properties during shaft creation.



Shaft options

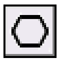


Example of object created by Shaft



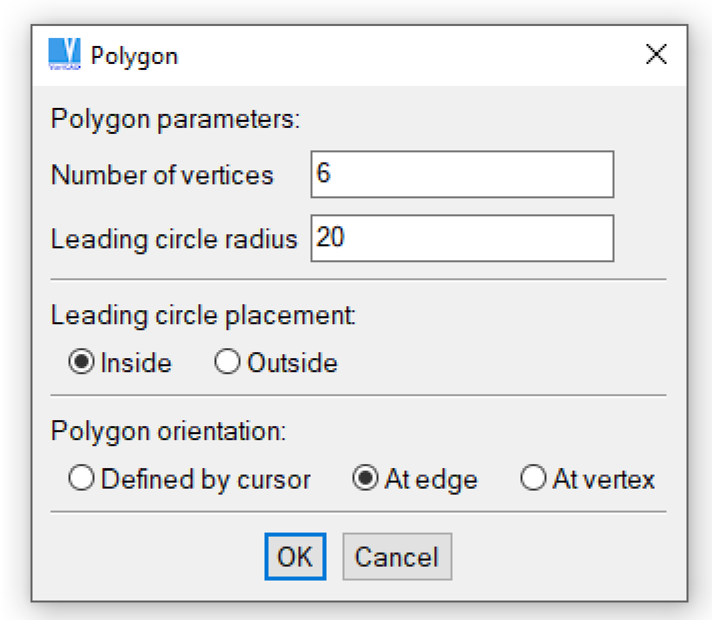
Rectangle - RECT

Creates a rectangle by two opposite corners definition. If you right-click an empty area when a first rectangle or a second rectangle corner is defined, a pop-up menu appears and you may select cursor increment mode or enter exact rectangle dimensions from keyboard.



Polygon - POL

A polygon is an object in which all sides have the same length. The polygon can fit inside or outside a specified radius, and you can define the number of sides (vertices) and vertex location.

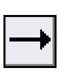


Polygon options

Tangent Line - TAN

Creates a tangent line by selecting two circles or arcs. The tangent line endpoints are located where you select the circles/arcs.

Arrows



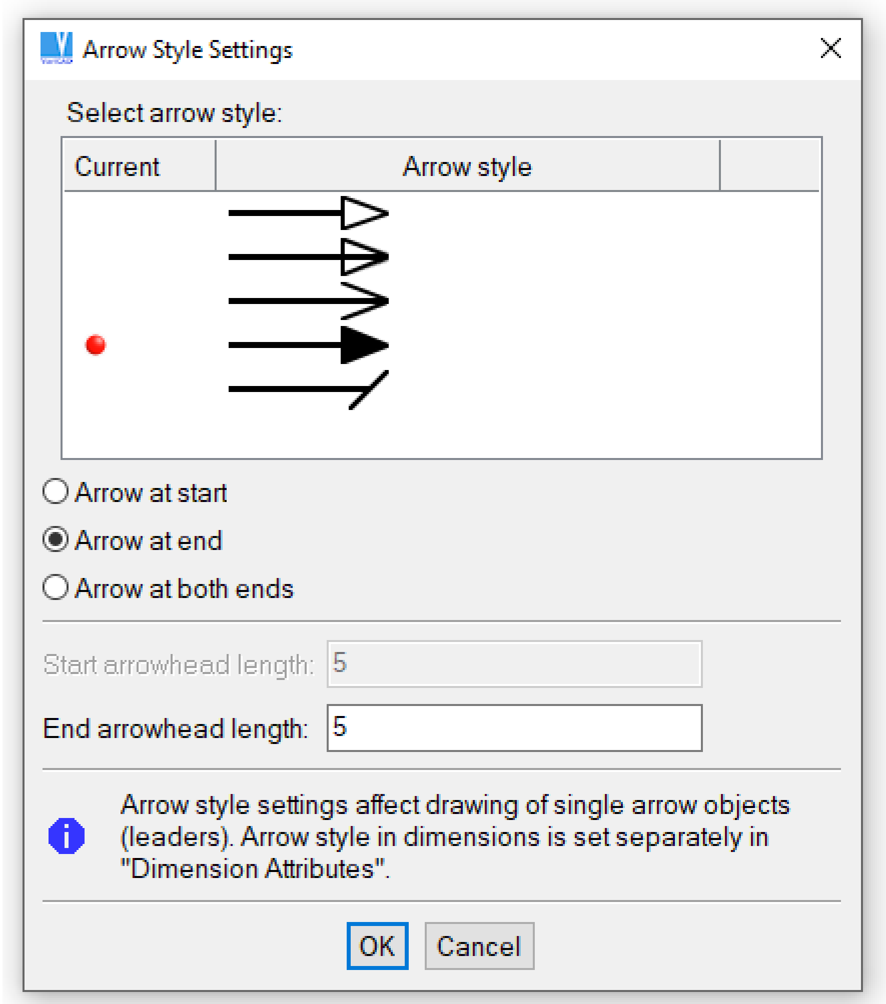
Arrows - ARR

An arrow is a single- or multi-segment line that has an arrowhead at the end of its last segment.

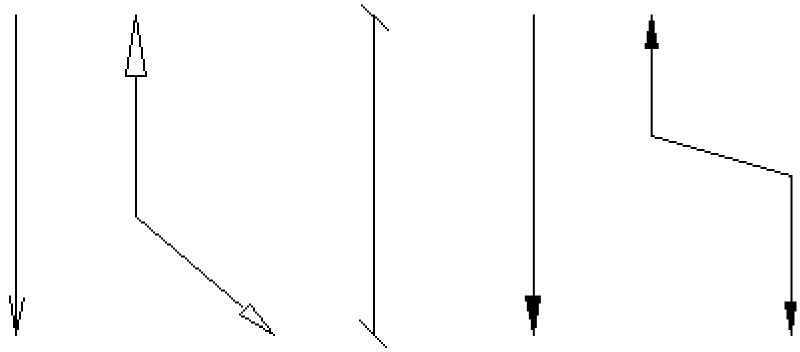
Arrow Attributes Setting – ARA

This function is available on the Tools menu. You can set the following attributes:

- Style of arrowheads
- Length of arrowheads
- On which ends of line the arrowheads are created

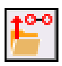


Arrow style settings

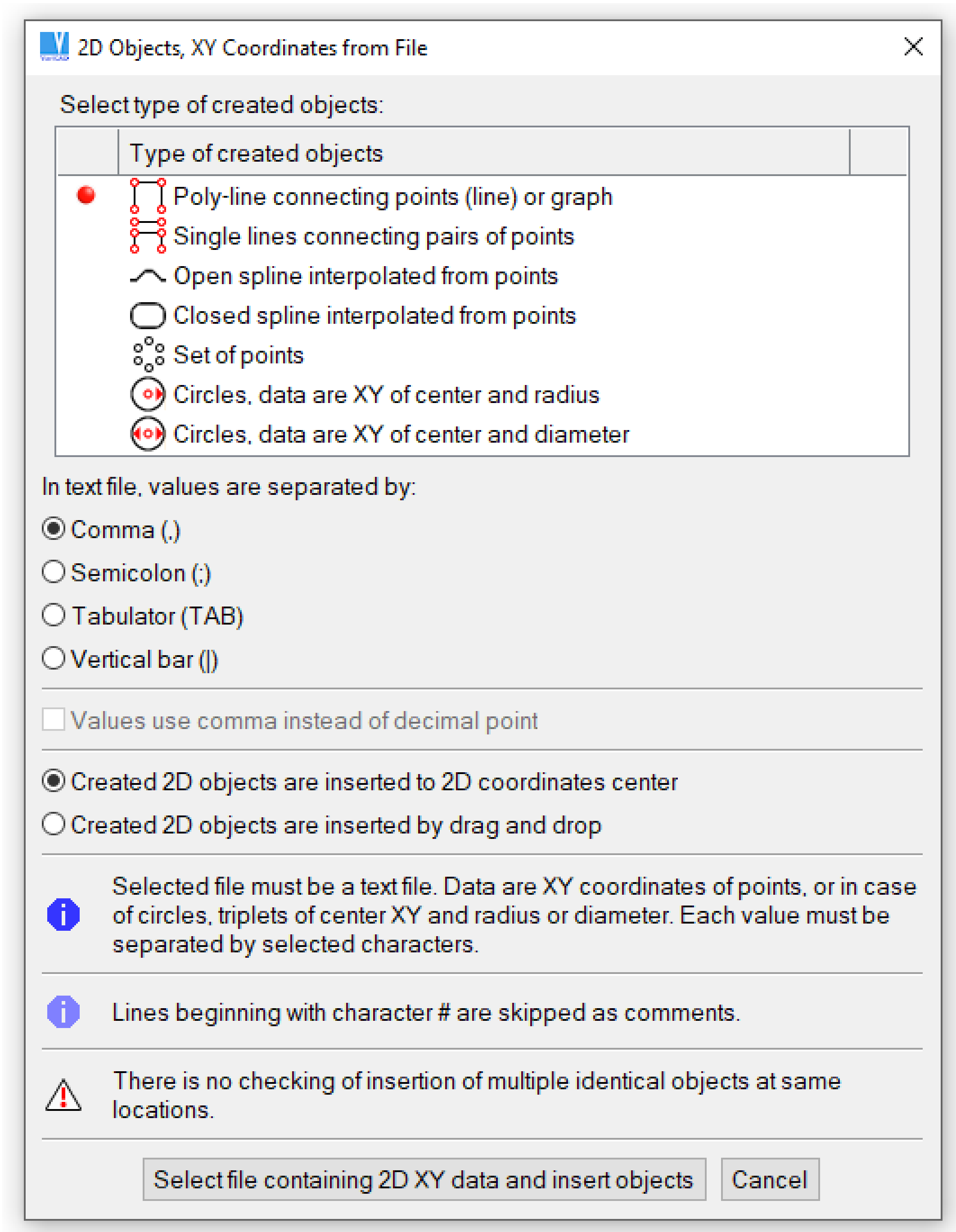


Example of various types of arrows (each arrow is a single object)

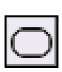
Drawing Curves

 **2D Objects, XY Coordinates from File – 2DFF**

Various types of 2D objects can be defined by X and Y coordinates loaded from a text file. See image below. After selecting type of 2D objects and other properties, select a file-name. Values are loaded from the file, and you can insert a new 2D object.

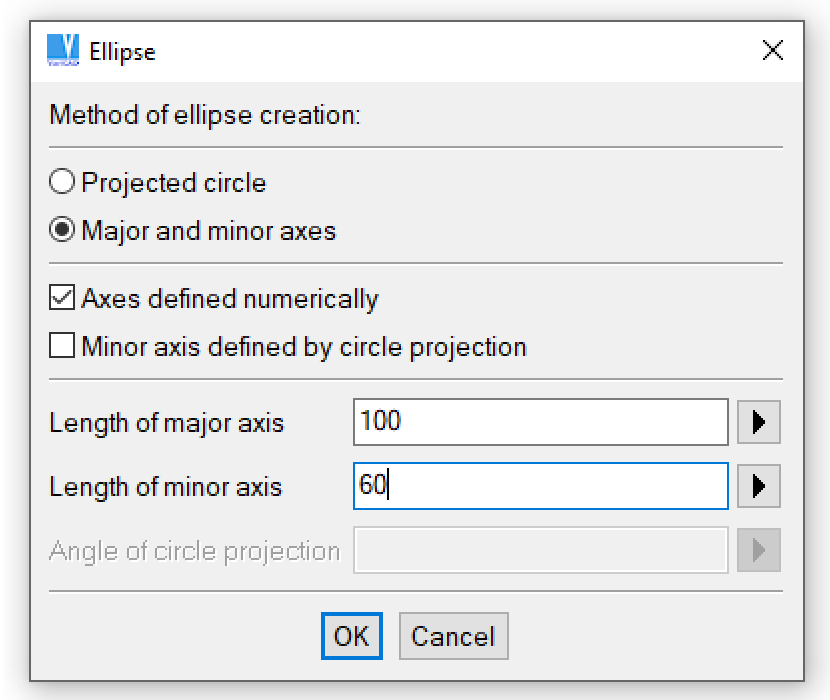


2D Objects, XY Coordinates from File

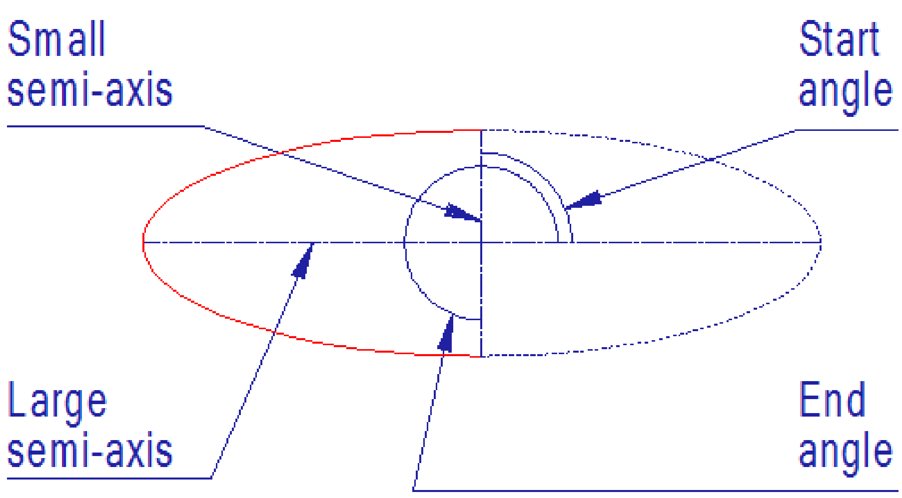
 **Ellipse - ELL**

An ellipse can be created by defining major and minor axes, or by projecting a circle. To create an elliptical arc, you can define start and end angles. Identical angles will produce an entire ellipse. You can easily specify identical angles by pressing the Space Bar twice at the same location – first, while entering the first angle and again (without moving the cursor) while entering the second angle.

Ellipsis is created as a NURBS curve.



Ellipsis definition



Example of ellipse

Spline - SPL

A spline is a curve defined by a set of points. The spline passes through the defined points. You can choose whether to create an open or closed spline. Closed curves are connected smoothly – gap between first and last point is spanned.

Splines are created as interpolation NURBS curves.

Creating Points

Points are used in 2D as auxiliary objects.

Point - POINT

Creates single points by clicking on point locations.

Points on Arc - POC

Creates points along an arc. You can specify a number of equally-spaced points, or define the distance between points.

Points on Line, Number - PLN

Creates a specified number of equally-spaced points along a line.

Points on Line, Distance - PLD

Creates points along a line, separated by a specified distance.

Points from File - PFF

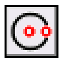
Inserts points from a text file, listed as X, Y coordinates. Each set of coordinates in the text file must be separated by a space.

Creating Circles and Arcs

Circles and arcs are the same type of 2D object; a circle is a 360 degrees' arc. Angles of arcs are measured counterclockwise, with zero degrees along the right direction of X axis. If a circle or an arc is defined by cursor dragging, you may press right mouse button. Then, menu appears and you can select cursor dragging increment or define exact radius value from keyboard. In most situations, cursor increment is measured as DX DY from last defined point or from coordinate center. But when you drag a circle or arc from its center, cursor increments are measured as increments of radius.

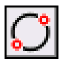
Circle Center Radius - CCR

Creates a circle defined by a center point and radius.



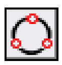
Circle Center Point - CCP

Creates a circle by defining the center point and a point on the circumference.



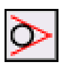
Circle 2 Points - CR2

Creates a circle by defining two circumference points and the radius. The circle is created when you specify the side of the circle center, relative to the line connecting the two circumference points.



Circle 3 Points - C3P

Creates a circle by defining three circumference points.

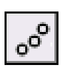


Circle Tangent to 2 Objects - CT2

Creates a circle tangent to two objects (lines, circles, arcs) with a specified radius.

Circle Tangent to 3 Objects - TG3

Creates a circle tangent to three objects (lines, circles, arcs).

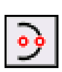


Group of Holes - HOL2

Creates a group of holes (circles) along a circle or line.

Arc Center Radius - ACR

Creates an arc by defining the center point, radius, and start and end angles. To create an entire circle, the start and end angles must be identical. You can easily specify identical angles by pressing the Space Bar once while entering the first angle and again (without moving the cursor) while entering the second angle.

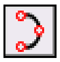


Arc Center Point - ACP

Creates an arc by defining the center point, point on circumference, and start and end angles. To create an entire circle, the start and end angles must be identical. You can easily specify identical angles by pressing the Space Bar once while entering the first angle and again (without moving the cursor) while entering the second angle.

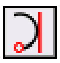
Arc 2 Points - AR2

Creates an arc by defining the radius and two endpoints. The arc is created when you specify the side of the arc center, relative to the line connecting the two endpoints.



Arc 3 Points - A3P

Creates an arc by defining the first endpoint, a point on the circumference, and the second endpoint.



Arc Point Tangent - APT

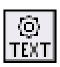
Creates an arc by defining the radius, point on circumference, and a tangent line or arc. The arc is created when you specify the side of the arc center.

Arc Tangent to 2 Objects - AT2

Creates an arc tangent to two lines or arcs, specifying the arc radius.

Creating Text Objects

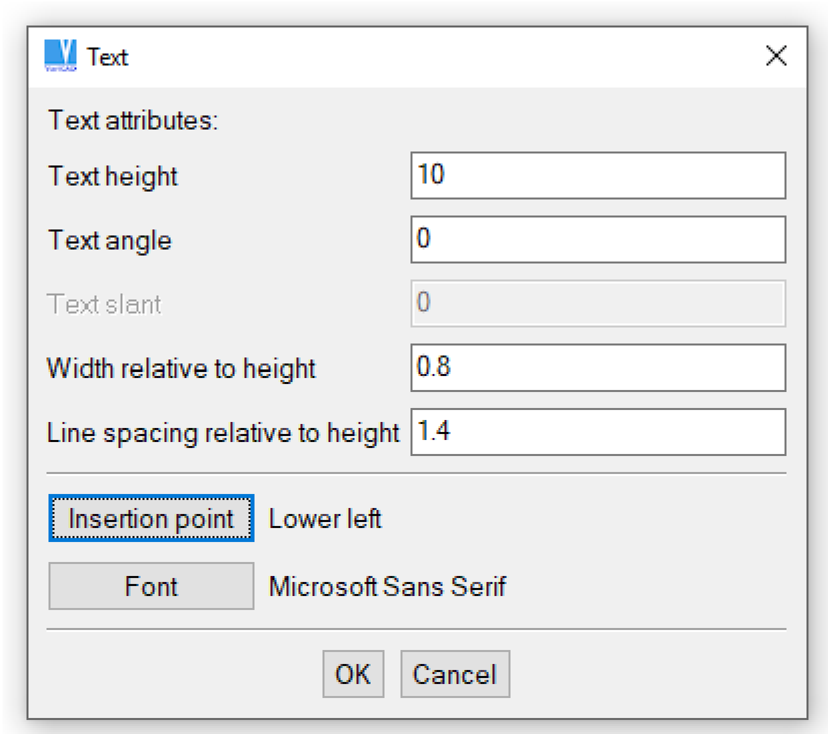
There are several ways you can add text to your drawing: as single text lines, as notes of a few lines, or inserted from a text file. Text can be horizontal or drawn at a specified angle. See *Editing Text (page 73)* for details on modifying existing text.



Text Attributes - TXA

You can set the following text attributes:

- Text height
- Text angle - angle of the text line (single-line text or text file inserts are horizontal, but the angle can be changed later)
- Text slant - the angle of individual letters
- Line spacing - relative to text height
- Text width - relative to text height
- Text insertion point
- Text font

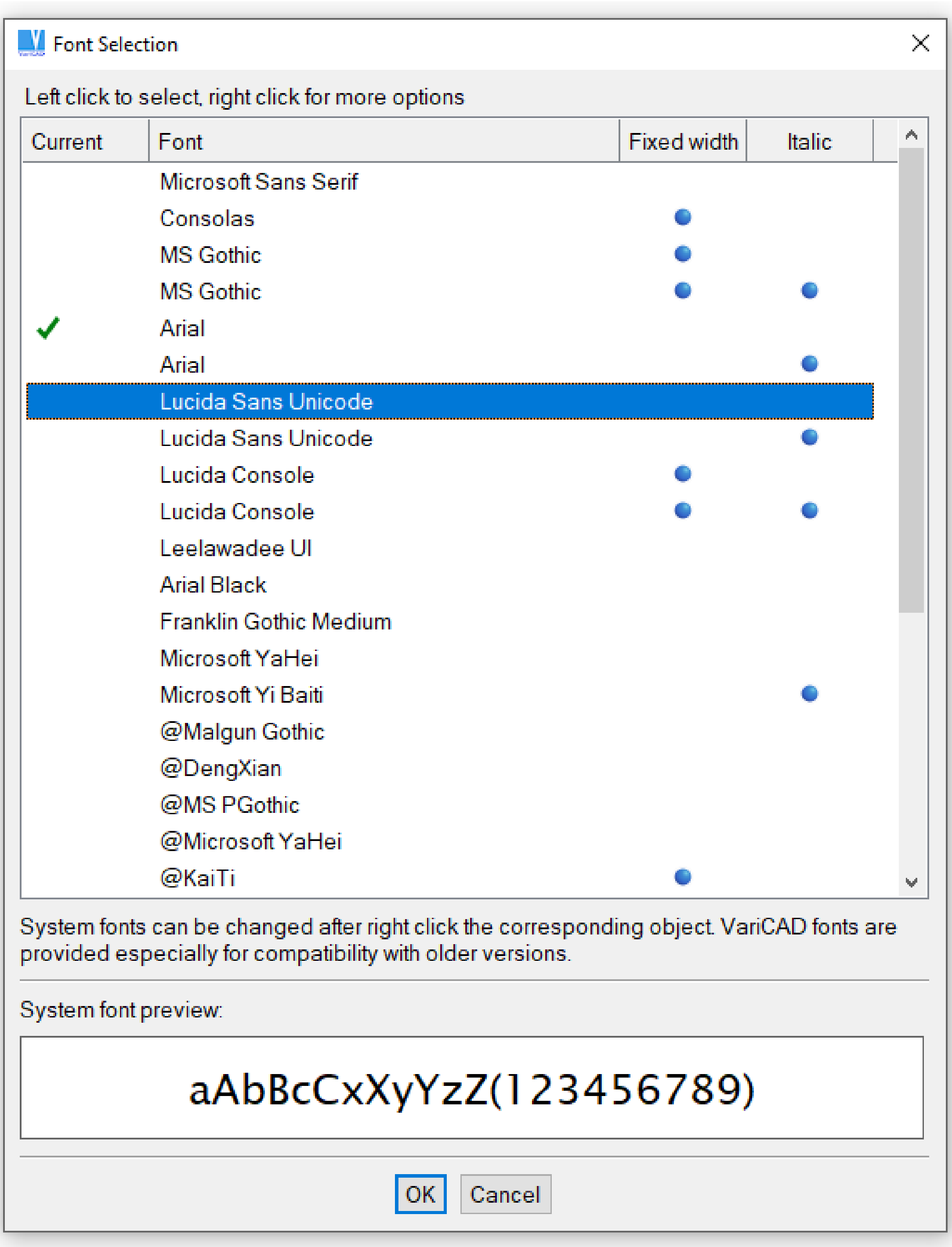


Text attributes window

Text Encoding, Text Fonts

All text objects used in VariCAD are encoded in Unicode. However, there are some limitations if you select text fonts. VariCAD works with two types of fonts:

- VariCAD fonts, like VariCAD standard. These fonts are always available, but they contain only limited character set. VariCAD Standard font supports Western European encoding, Central European encoding and the Cyrillic.
- True type fonts. These fonts are available from operating system. Windows operating systems offer different fonts than Linux systems. True type fonts usually contain most characters from Unicode character set.



Text font selection

Writing Text from Up to Down

Text in VariCAD is written from left to right. If necessary, there is also a possibility to write single lines from up to down, in Chinese, Japanese or Korean languages under Windows. Select a true-type font which name begins with character @ - for instance, font named @Gulim. Then, set text angle to -90 degrees. True type fonts which name begin with @ have Chinese, Japanese or Korean characters rotated by 90 degrees.

There is no possibility to write texts from right to left in current version of VariCAD.

Windows 10 may not have all predefined fonts installed by default.

Note (Multiple Lines) - NOTE

A note is a single object containing lines of text. Notes are inserted by drag and drop, and you can define text width as a ratio of two distances.

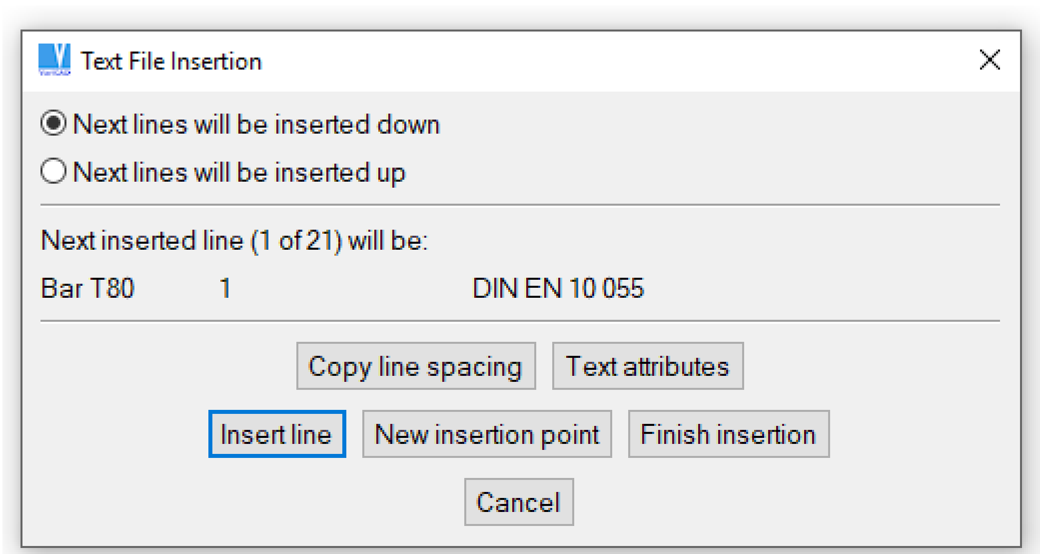
Single Text Line - TEX

Single text lines are drawn horizontally. First locate the start point of the text, then type the text. Press Enter to finish each line. Before selecting an insertion point of text, you can also:

Copy Text Attributes - use attributes from a selected text object.

Insert Text File - TXI

Inserts text from a file into the drawing. Select a file, and then select the lower left point of the first text line. You are asked to confirm each line, and you can interrupt insertion and redefine the position of subsequent lines. You can also cancel the insertion before reaching the end of the file.



Inset Text File window

Editing and Deleting 2D Objects

This section describes how you can change the shape of selected 2D objects and text. For information on how to select objects, see *Selecting 2D Objects (page 45)*.

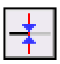
You can select a command, and then select objects to be edited. Or, you can select objects first and then select a command. Also, you can select an object or objects, then right-click an empty area and select a command from pop-up menu.

Deleting Objects

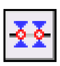
 **Delete 2D Objects – DOB, Ctrl + D**

Deletes one or multiple selected 2D objects.

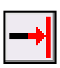
Changing Objects Geometry

 **Trim - TBO**

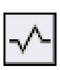
Removes sections of lines, splines or arcs that lie on one side of a trimming curve. You can also define a temporary trimming line by defining two points.

 **Remove Segment - RSG**

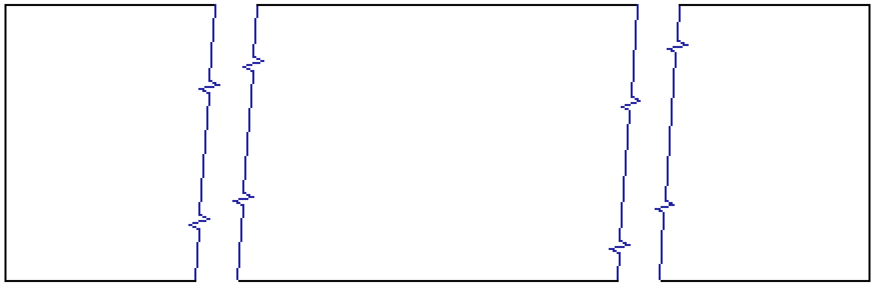
Removes a segment from an arc, spline or line. The removed segment is defined by two points.

 **Extend - EBO**

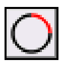
Extends lines or arcs to a defined curve. You can also define a temporary extension line by selecting two points.

 **Break Line - BLN**

Creates break marks on a selected line, by selecting the location of each break mark.



Example of break lines

 **Circle from Arc - CEC**

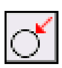
Creates a full circle from a selected arc.

 **Edit Spline - ESP**

Edits the selected spline. Select one of the spline definition points and drag it to a new location. While dragging, the spline shape updates dynamically. Press Enter or right-click to finish editing.

 **Change Line Length - CHLL**

Changes length of a selected line or performs stretching.

 **Change Arc Radius - CHAR**

Changes radius of a selected arc or a circle. If the arc is created as a rounded corner, VariCAD changes the fillet radius and preserves the filleting. You can select multiple lines, multiple arcs or multiple 2D rounded corners. Change is performed at one step for all selected objects.

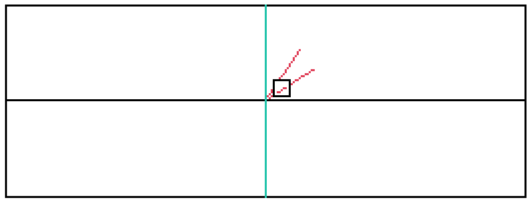
Creating Corners, Chamfers and Fillets

Each of these functions can be applied to line, arc or NURBS segments. You can select both segments at once by selecting the intersection snap point, indicated by a V-shaped symbol at the cursor. If segments are selected separately, you can choose to apply the function to the first, second, or both segments. If any segments will be trimmed as a result of the function, select the segment on the non-trimmed portion.

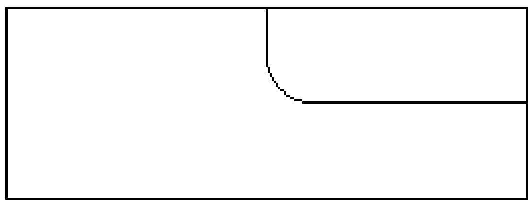
Selecting Corners, Chamfering or Filleting at One Step

Instead of selecting a particular command (for instance, by clicking a corresponding icon), you can press and hold Shift key and move cursor over corners or intersections of 2D objects. Whenever a V-shaped symbol appears, you can right-click and then select corner, chamfering or filleting from menu.

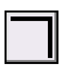
Some situations do not allow all three possibilities. If you already selected a function (like filleting), you may continue to fillet corners after finishing each step, until you select another command or interrupt by pressing ESC key.



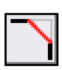
Selecting an intersection to be rounded



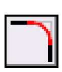
Result of rounding

 **Corner - CCO**

Creates a sharp corner at the intersection of selected segments. Segments will be trimmed or extended to create the corner, so select the segment on the non-trimmed portion.

 **Chamfer 2D Corner- CHM, Ctrl + R**

Creates an angled chamfer between two lines. You can define the chamfer by distance along each segment, or by distance and angle for one segment. You can choose whether to trim or extend segments.

 **Fillet 2D Corner – RND, Ctrl + F**

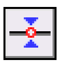
Rounds the corner between lines or arcs. When applying a fillet to two arcs, you can choose a convex or concave result. You can also choose whether to trim or extend segments.

Breaking and Dividing 2D Objects

 **Explode - EXP**

Explodes selected 2D object into their basic elements. Objects are exploded according to their type as follows:

- Lines, rectangles, polygons become single line segments
- Polylines become lines, arcs or splines
- Arrows become lines
- Hatches become lines
- Axes become lines
- Dimensions become lines, arrows, circles and text
- Text becomes single text lines
- Symbols become the objects they contain
- Blocks become the objects they contain; nested blocks become single blocks

 **Divide by Point - BPO**

Divides a line, spline or arc at the selected location, creating two separate objects.

 **Divide by Curve - BBO**

Divides a line, spline or arc by a specified curve, creating two separate objects. You can define a temporary curve by selecting two points.

Editing Text

 **Edit Text - ETX**

Modifies the selected text. The text lines appear in the editing window where they can be edited.

 **Text Attributes - TAC**

Changes attributes of the selected text. Before selecting text to change, define the new attributes and whether the text parameters, font or insertion point will be changed.

 **Align Text - JTX**

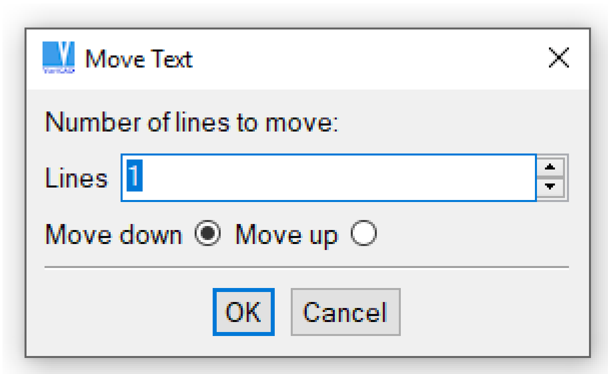
Moves text lines horizontally, aligned to a center point or to another selected point.

 **Text Width - TWD**

Changes the width of the selected text width. The new text width is determined as ratio of two distances you defined with the cursor.

Move Text Vertically - MTI

Moves text lines vertically. The move distance is a factor of line spacing, and you must enter the number of lines to move. Text can be moved after text is deleted, or in order to insert text between lines.



Move Text window

Explode Font into Segments - BTF

Explodes 2D text letters into line segments. In older versions, exploded font contours were used for creation of 3D texts. Now, 3D text objects can be created by commands related to font extrusions – see *3D text (page 269) (page 154)*

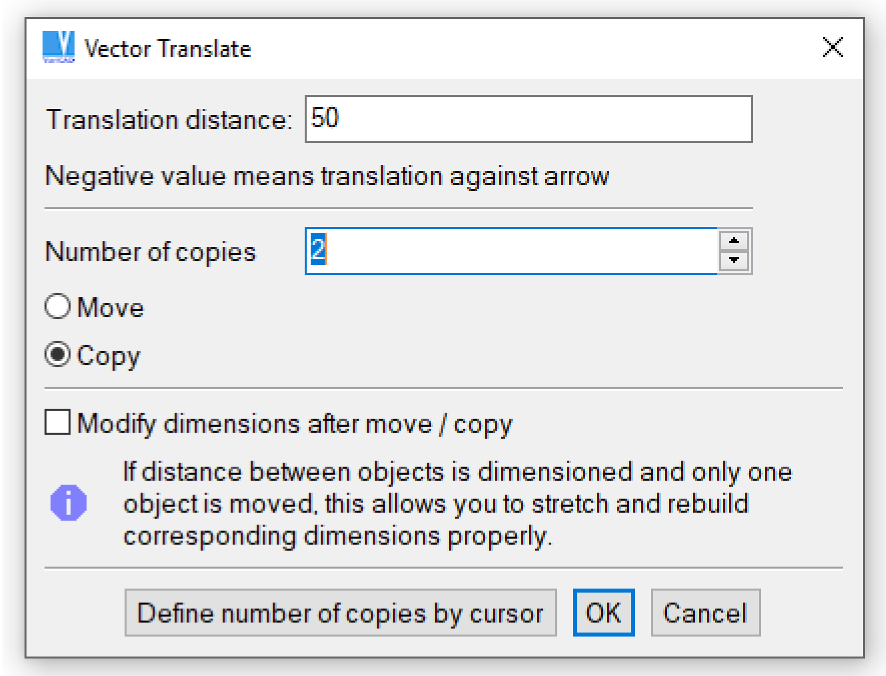
Transforming and Copying 2D Objects

Transformation functions enable you to change an object's location or scale, and move, copy and rotate objects. All functions allow you to define transformation parameters first and then apply them, or to perform the transformation dynamically. See *Dragging Objects (page 21) (page 10)*.

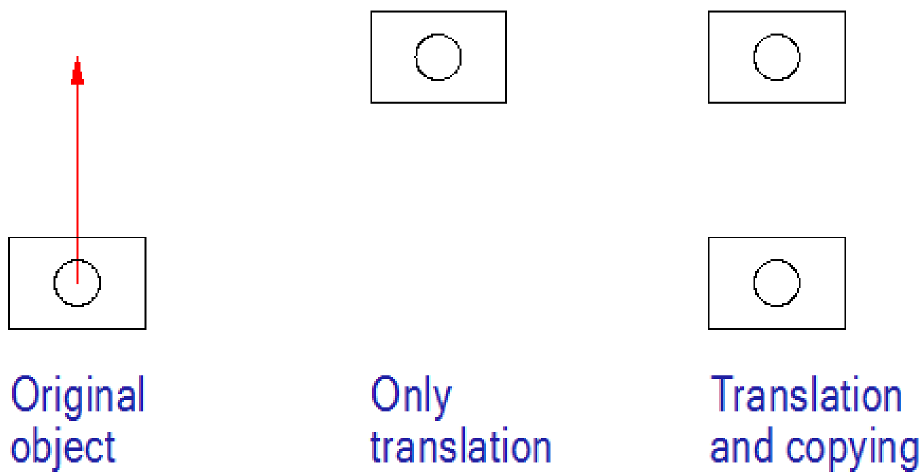
Translating, Rotating and Scaling

Translate or Copy 2D Objects– MOV, Ctrl + T

Moves or copies objects along a defined translation vector. The vector is defined by two points and indicates distance and direction. If objects are to be copied, you can specify the number of copies, and the original object is preserved. Multiple copies are made along a row. You can modify the length of translation, so the original vector defines only a direction.



Vector Translate window



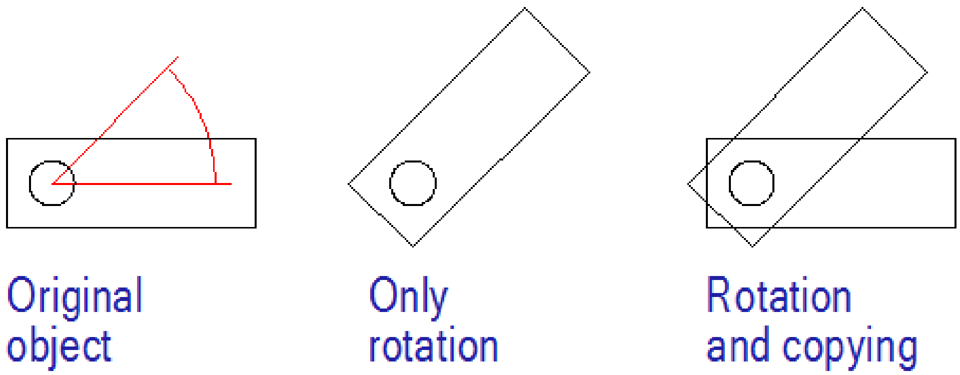
Moving and copying

 **Dynamic Translation - DRG**

Moves or copies objects, with cursor movement defining the new object locations. Select the objects, define their insertion point, and select new locations of the insertion point. If you choose not to delete the original objects, copies will be made.

 Rotate or Copy 2D Objects - ROT

Rotates objects around a defined point by a specified rotation angle. You can choose to rotate only, or to rotate and copy. If multiple copies are made, each copy is separated by the rotation angle.



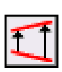
Rotating and copying

 Dynamic Rotation - DRO

Rotates dynamically using the cursor. Select objects, define the center of rotation, and select a reference point. The cursor movement defines the direction from the rotation center to the reference point. You can choose to rotate only, or to rotate and copy.

 Translate and Rotate - DTR

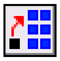
Both moves and rotates objects. Select objects and define the first and second reference points. Then define new locations of both reference points. If you choose not to delete the original objects, copies will be made.

 Scale - SCA

Rescales objects. Select the center of scaling and the scale value. You can choose to scale only, or to scale and copy. If multiple copies are made, the scale value is applied to each copy progressively.

 Dynamic Scaling - DSC

Scales dynamically using the cursor. Select objects, define the center of scaling, and select a reference point. The cursor movement defines the scale value. If you choose not to delete the original objects, copies will be made.

 **Array Copy - CTA**

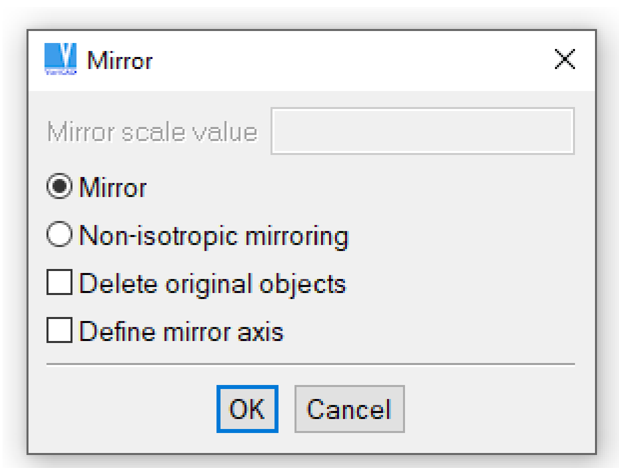
Creates an array of copied objects. Define the origin of rows and columns, the direction and distance between columns, and the direction and distance between columns. Then specify the number of copies in rows and columns. This value is the number of new copies; the original object is not included. You can also define the number of copies by using the cursor to select the location of the last objects in rows and columns.

Mirroring Objects

 **Mirror - MIR**

Creates mirrored objects from selected ones. The mirror axis can be an existing line, or you can define a temporary line by selecting two points. You can choose whether to delete existing objects.

If you select non-isotropic mirroring, the mirrored object will be created on the same side of the mirror axis (in case of positive mirror value), and will be scaled perpendicular to the mirror axis. Non-isotropic mirroring is not possible for dimensions. Non-isotropic mirroring scales objects only in one direction.



Mirror window

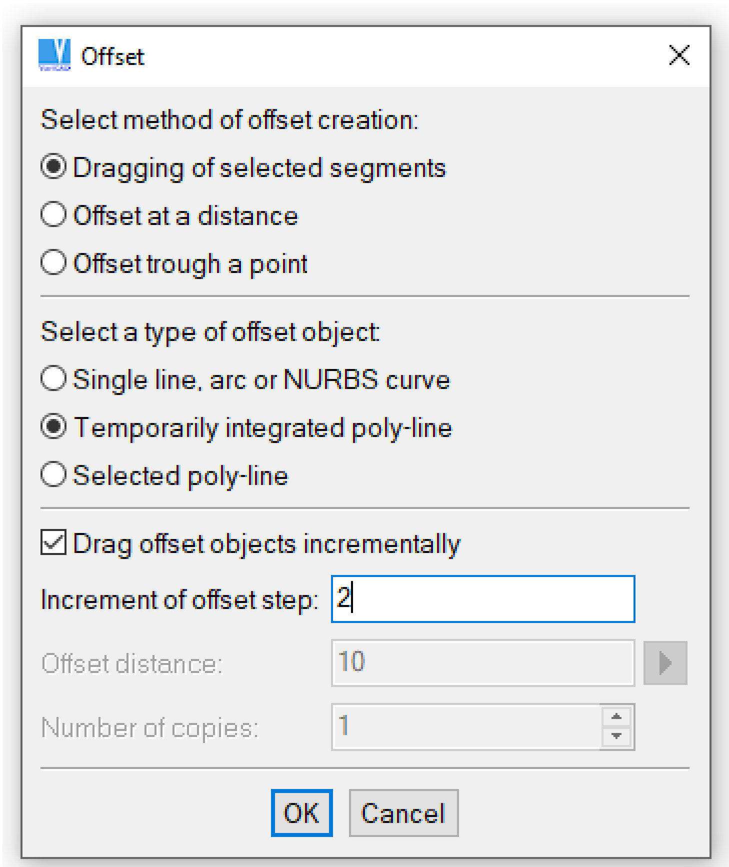
Offsetting Objects

 **Offset - OFFS**

You can use following methods of offset definition:

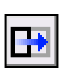
- Dragging of selected segments. Offset copies are changed dynamically, according to cursor movement. It is very convenient to set dragging of offset objects in increments. In such case, offset distance is changed in defined steps – for instance per one millimeter, and the offset distance is displayed near cursor.
- Offset at a distance. Select a distance and a side the offset copy is created at.
- Offset through a point.

Offset copies can be created for single objects, temporarily integrated poly-lines or existing poly-lines.



Offset window

Stretching Objects

 **Stretch - SOB**

This type of stretching uses the cursor movement to define a new shape of selected linear objects. Use a selection window to encompass the endpoints of line segments you want to move. After selecting objects, select a reference point and move the cursor to stretch the objects. To stretch horizontally or vertically, turn on temporary construction lines (see *Temporary Construction Lines*) or use Ortho mode. Then, vertical or horizontal direction is easy to follow.

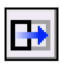
To start stretching of objects, you can press Ctrl + Shift and move the cursor whenever system waits for a new command. A selection window is started immediately. Select objects and a reference point.

You can select objects in advance. After selection, press and hold Ctrl + Shift while moving cursor. In next step, VariCAD process the pre-selected objects, looking for lines end-points. No other objects are automatically selected from defined rectangle. Next, select the reference point. This way you can exclude some objects which would be otherwise stretched.

Stretching 2D Dimensions

If you need to stretch linear dimensions (like horizontal or vertical dimensions), you can select them in advance - by single click or by selection rectangle. Then, create a selection window by pressing Ctrl + Shift and moving the cursor. If you define a selection rectangle and only one dimensioned point is inside, it is automatically considered as the reference point. Or, you have to select a reference point using mode “Nearest dimensioned point” – see *Selecting 2D Locations (page 48)*.

If you stretch a dimension connected into 3D, all connections are lost and after changes in 3D, the dimension will not be updated.

 **Stretch to Direction - DST**

Stretches objects along a defined direction. You can stretch objects horizontally, vertically or by a diagonal vector defined by two points. Define the line dividing the preserved and moved points of objects which will be stretched, select objects and the reference point. Move the cursor to define a new shape. This type of stretching can only be done on lines; other types of objects are moved without being stretched.

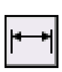
Dimensioning

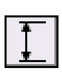
Dimensioning enables you to describe geometry by displaying measurements. All dimensioning functions can be found on the Dimensioning toolbar or in the Objects / Draw menu. To create dimensions, select the objects to measure and drag the mouse to locate the dimension text (see *Dragging Objects (page 21) (page 10)*). Dimensioning tools have several options for additional formatting and settings.

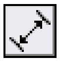
If dimensions are created in 3D views exported into 2D area, connections between dimensions and corresponding 3D objects are established. After changing 3D geometry and returning back into 2D mode, dimensions are automatically updated. See *Automatic updates of dimensions, axes and hatches after changes in 3D (page 112)*

Once created, the dimensions can be easily edited (see *Editing Dimensions (page 94)*). Format and style of dimensions and dimension text can be modified using *Dimension Attributes (page 85)*. Changing dimension text height affects the length of leader lines arrows.

Single Dimensions - Horizontal, Vertical and Diagonal

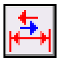
 **Horizontal Dimension - HDI**

 **Vertical Dimension - VDI**



Diagonal Dimension - SDI

Single dimensions are defined by selecting start and end points, then locating the dimension text.



Change Dimension Shape - Before the text location is defined, you can change the current dimension format. You can also use Dimension Format to set the format of future dimensions.

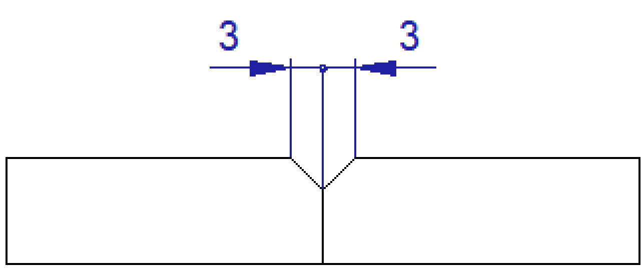
Dimension Format provides the following options:

- Center dimension text - for the current dimension, all texts will be automatically centered. If turned on, this option turns centering off.
- Toggle points and arrow - switches between points and arrows on the leader lines. This option is useful when space is limited.
- Cancel predefined level - cancels all other previously selected options, the dimension is created at level exactly defined by cursor location.

These options of Dimension Format allow you to define the level (text location and dimensioning line location) of created dimension or all future dimensions:

- Created dimension at the same level as selected (according to a selected dimension)
- All dimensions at the same level as selected
- All dimensions at the same level as created (next dimensions are according to the currently created dimension)
- Created dimension parallel to selected
- All dimensions are parallel to selected
- All dimensions are parallel to created

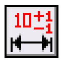
The last option from menu allows you to redefine the dimension style.



Dimension arrows changed

Change Dimensioning Scale

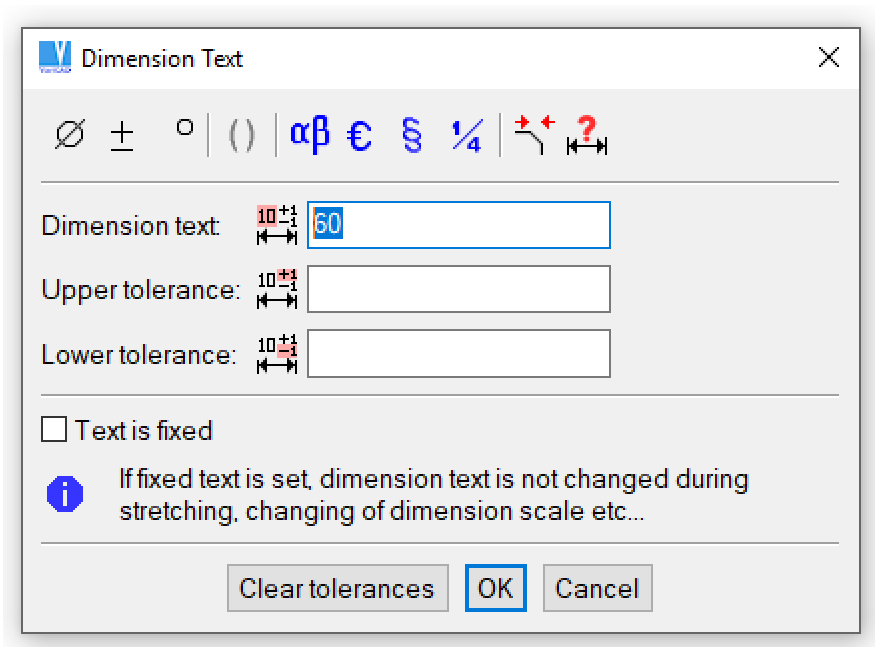
Dimension scale value multiplies measured values of linear, diameter or radius dimensions. It must be used, if a part of 2D drawing is created in different drawing scale than entire drawing (for instance, in case of enlarged details).


Change Text or Tolerance - modify the dimension text or tolerances.

Dimension text is created automatically, and you can use *Dimension Attributes (page 85)* to define its parameters. For currently created dimension, you can change text in the text input window.

You can select this option by clicking the corresponding icon in toolbar, from pop-up menu (available by default after Ctrl + Right-click), or, very conveniently by pressing TAB during dimension text location. The last possibility allows you to easily select the input text field, if you press TAB repeatedly – input is changed from dimension text to upper, then lower tolerance.

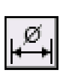
Dimension text window offers you to enter special characters, by clicking the corresponding icons. Also, you can click parentheses icon – then, entire text is closed into parentheses. Dimension text is properly updated after changes and remains closed in parentheses, too.

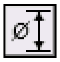


Change dimension text window

Predefined Horizontal, Vertical and Diagonal Dimensions

Predefined dimensions contain standard text characters such as diameter symbols. These are useful when dimensioning objects like circles or threads.


Horizontal Diameter Dimension - HDM

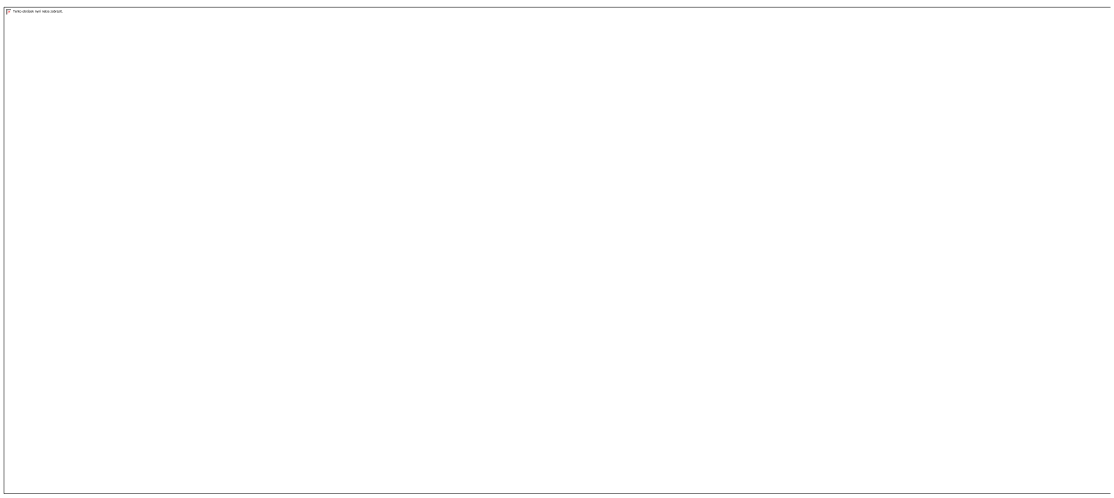
 **Vertical Diameter Dimension - VDM**

 **Diagonal Diameter Dimension - SDM**

Horizontal Thread Dimension - HTH

Vertical Thread Dimension - VTH

Diagonal Thread Dimension - STH



Example of predefined dimensions

Serial, Parallel and Datum Dimensions

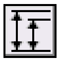
These functions create multiple dimensions. The first dimension in this group is defined as for single dimensions (by 2 points), and subsequent dimensions in the group require only one point.

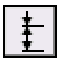
Parallel dimensions are a group of dimensions that all start at the same point. The offset between parallel dimensions is defined in *Dimension Attributes (page 85)*. Datum dimensions consist of a line of points, at each of which the dimension defines the total distance from the start point. Serial dimensions are a chain of aligned dimensions that define the distance from point to point.

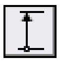
Horizontal Parallel Dimensions - HPD

Horizontal Serial Dimensions - HSD

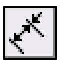
Horizontal Datum Dimensions - HDD

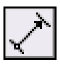
 **Vertical Parallel Dimensions - VPD**

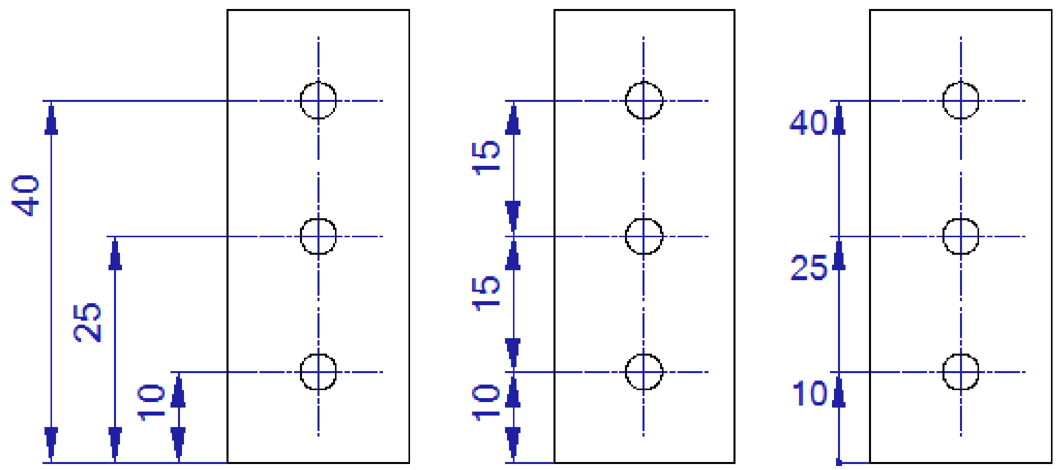
 **Vertical Serial Dimensions - VSD**

 **Vertical Datum Dimensions - VDD**

 **Diagonal Parallel Dimensions - SPD**

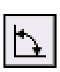
 **Diagonal Serial Dimensions - SSD**

 **Diagonal Datum Dimensions - SDD**



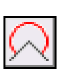
Example of baseline, serial and datum dimensions

Angular Dimensions

 **Angular Dimension - ADI**

Angular dimensions define the angle between two lines. You have the same additional options for dimension shape changes as for the linear dimensions. See *Change Dimension Shape (page 79)*.

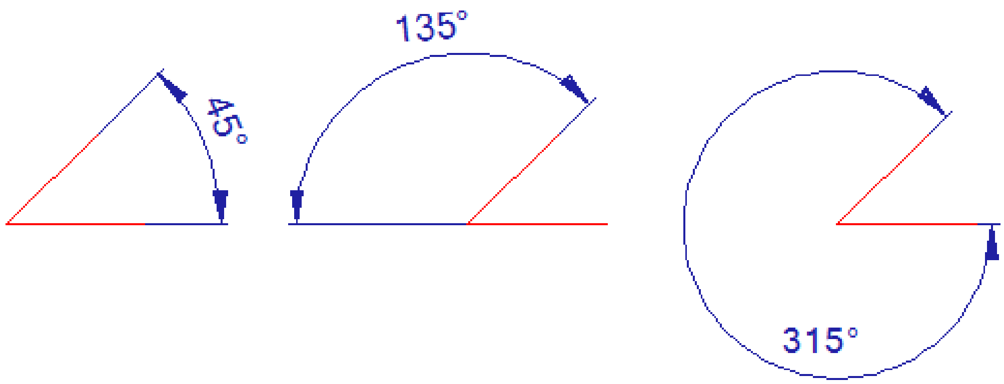
For angular dimension, define first and second line. When the second line is defined, you can optionally select following:

 Greater Than 180 - dimensions the angle greater than 180 degrees.

 Turn off dimensioning of angle greater than 180 degrees, cancels the previous mode.

 Complementary Angle On - dimensions the angle complementary to 180 degrees.

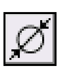
 Complementary Angle Off - turns off the complementary angle dimensioning, cancels the previous mode.



Example of default angular dimensioning, complementary angle dimensioning and dimensioning of angle greater than 180 degrees

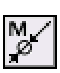
Diameter and Radius Dimensions, Thread Dimensions

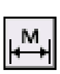
 **Radius Dimension - RDI**

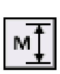
 **Diameter Dimension - DDI**

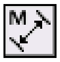
These dimensions are defined by selecting an arc or circle and locating the dimension text. Additional options for radius dimensions are:

- Extend to radius center - creates a mark at the arc center.
- Not to extend to radius center – cancels the previous mode.

 **Thread dimensions - THR**

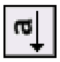
 **Horizontal Thread Dimension - HTH**

 **Vertical Thread Dimension - VTH**



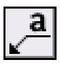
Diagonal Thread Dimension - STH

Thread dimensions behave like diameter or linear dimensions, except the predefined thread symbol is used. If the thread is created in 3D and then exported into 2D, either in sections or in a view, the dimension text is created automatically according to corresponding 3D object. Dimension recognizes thread type (like Metric thread, Pipe thread), and thread size or pitch (like M10, M10x1 …). It is necessary to use these four dimensioning commands to obtain the corresponding thread text, according to creation in 3D. Also, threads should be exported to 2D from basic views (like front view, back view, top view…).



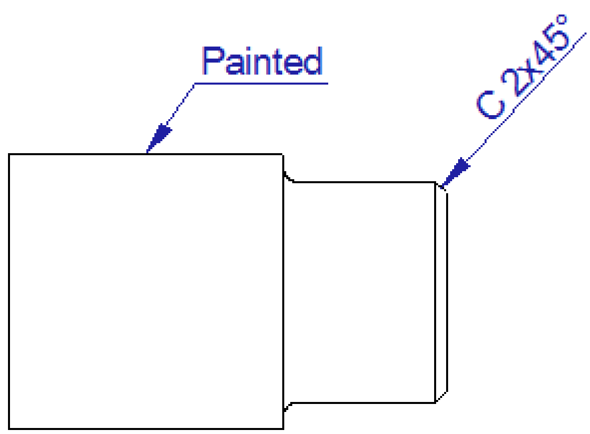
Single Text Arrow - STXA

Creates a text connected to a single arrow.



Multiple Text Arrow - MTXA

Creates multiple lines of horizontal text connected to an arrow.



Example of text connected to an arrow

Dimension Attributes

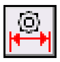
Dimension Attributes - DMA

Available on the Tools menu, Dimension Attributes enables you to change dimension styles and other properties. The following properties can be modified:

- Text attributes - height, slant, width and font. Text height controls the size of leader arrows.
- Dimension text format – number of decimal digits, usage comma instead of period, inch values format

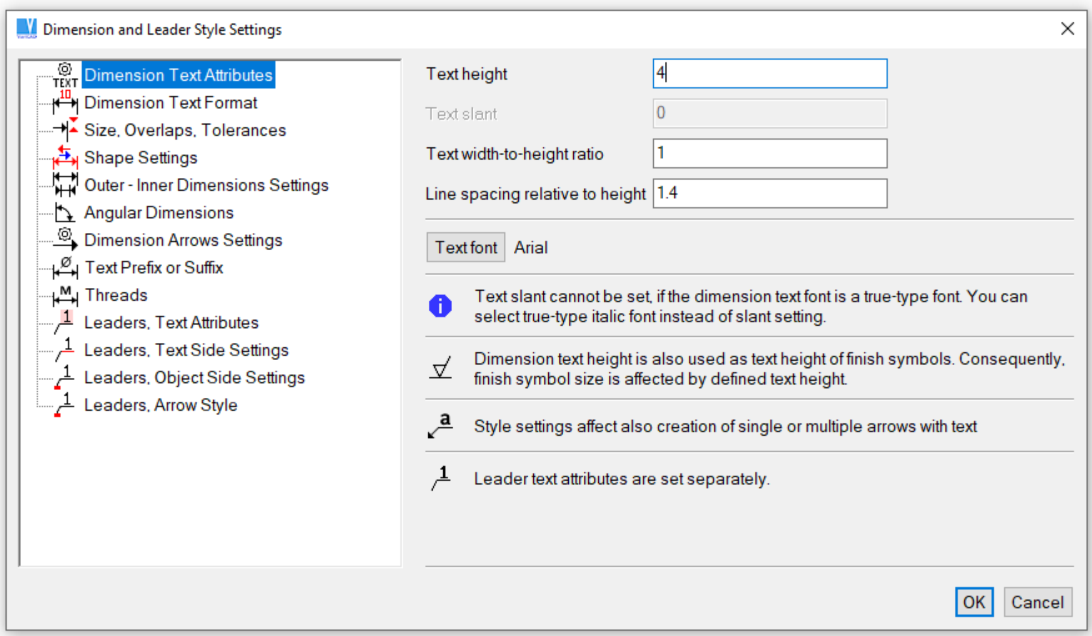
- Size, overlaps, tolerances - distance between parallel dimensions, distance between text and leader lines, witness line creation, tolerance position
- Angular dimensioning
- Arrow style
- Text prefix or suffix – defines usage of radius, diameter or thread marks

Changing dimension attributes allows you to set dimensioning according to various drawing standards, like DIN or ANSI. Dimension attributes are used also for finish symbols or single or multiple text arrows.

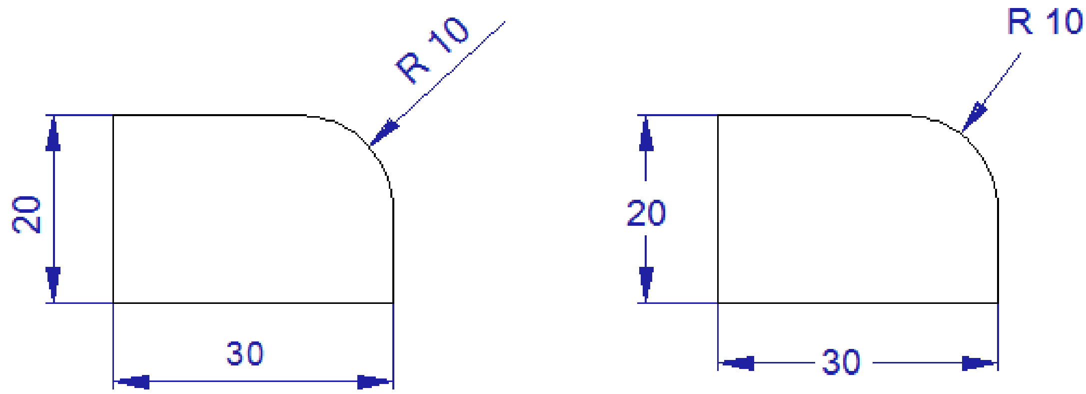


Edit Style of Selected Dimensions - EDS

This command changes the style of selected dimension or dimensions (also from entire drawing) to a style of a selected dimension, or according to current dimension style settings.



Dimension style settings

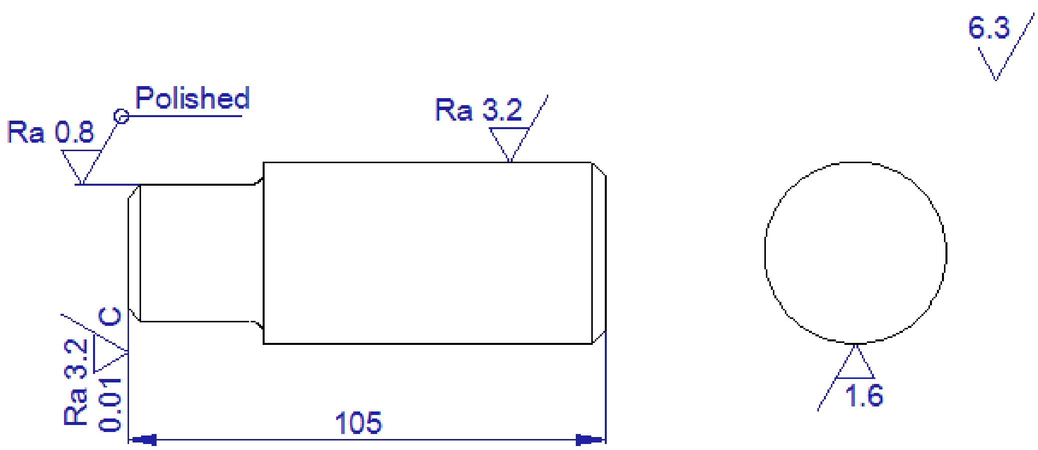


Example of dimension styles

Finish Symbols

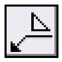
 **Finish Symbols - FSY**

Finish symbols are created by entering a roughness value, or by selecting a symbol for not-machined surfaces. Symbols can be attached to a line, arc or dimension witness line, or they can float. When attaching to a 2D object, you can drag the symbol around the object before selecting its final position.



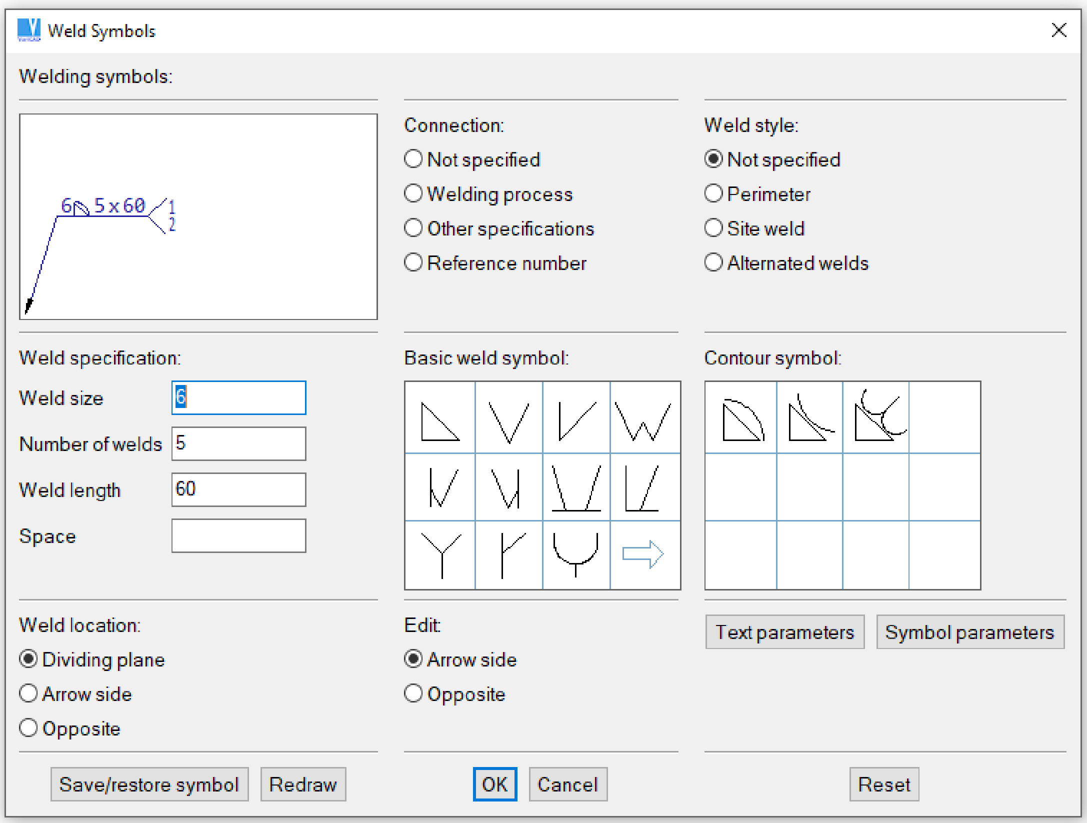
Finish symbols example

Weld and Tolerance Symbols

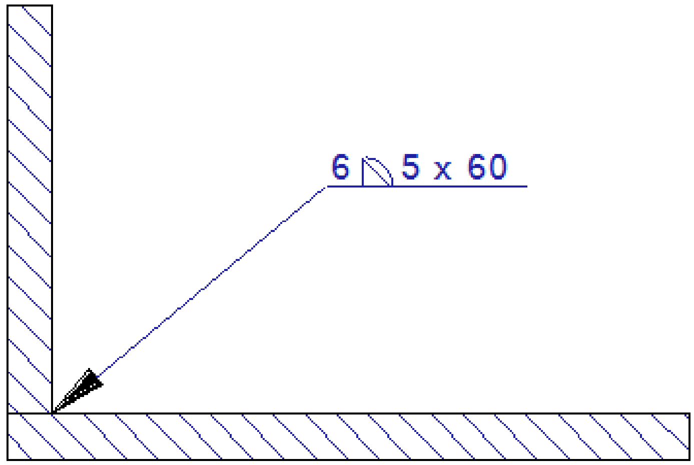


Weld Symbols - WSY

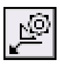
The weld symbol includes a basic symbol, supplementary symbol, dimension of weld size, and weld process symbol. When defining a symbol, you can change text and other attributes, and see a preview before creating it. You can also save up to nine symbols for future use. To create a weld symbol, first select the weld location and then select the symbol location.



Welding symbols definition



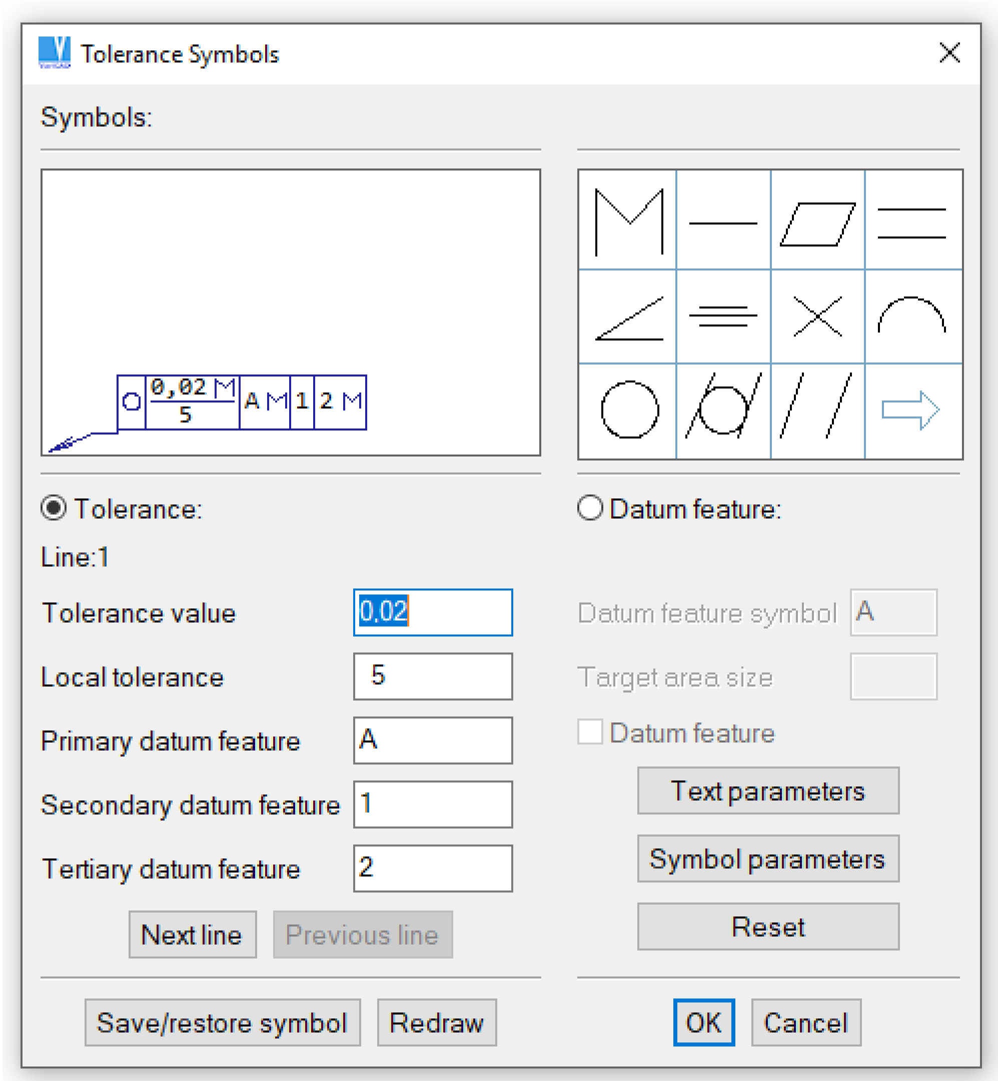
Example of welding symbol; this symbol was created within one function.

 **Weld Symbol Settings - SWS**

Available on the Tools menu, allows you to choose whether the weld symbols will be created according to ISO or ANSI standards.

 **Tolerance Symbols - TSY**

Similar to welding symbols. Datum feature symbols can also be created.



Tolerance symbols definition

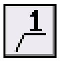
Creating Leaders, Item Numbers

VariCAD has a complex system solving item numbers and their appearance in 2D leaders. If BOM mask contains an attribute prototype of type "Item Number", you can define item numbers of solids together with other solid attributes. Moreover, you can define item numbers in BOM. Here, you may define item numbers for a batch of objects, or for entire assembly at one step. Also, you can remove item numbers the same way. See *Defining Item Numbers (page 309) (page 298)* in BOM section for more information.

If once defined, item numbers are exported to 2D drawings together with corresponding 2D views of 3D solids. In 2D, during leader creation item numbers are detected automatically from 2D objects.

If you change item numbers in 3D solids and if you perform a new export of 3D views into 2D area, item numbers can be checked and updated automatically. If necessary, leaders can be easily deleted from entire 2D drawing or from a selected 2D region. Leaders can be selected as a sub-type of dimension.

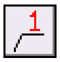
Leader style in 2D can be modified as well as a style of dimensions. Leader style settings are part of dimension settings. Leaders themselves are sub-part of dimensions objects.



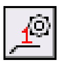
Create Leaders - LDR

This command creates 2D leaders. If 2D drawing was exported from 3D and if 3D solids have defined item numbers, leader texts (numbers) are assigned automatically to each leader. Otherwise, the leader number is assigned from 1 and each next leader has the number increased.

Following options are available during leader definitions:

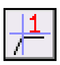


Changes default text of the created leader.

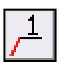


Changes leader (or dimension) style.

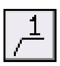
Sets distance between adjacent leaders.



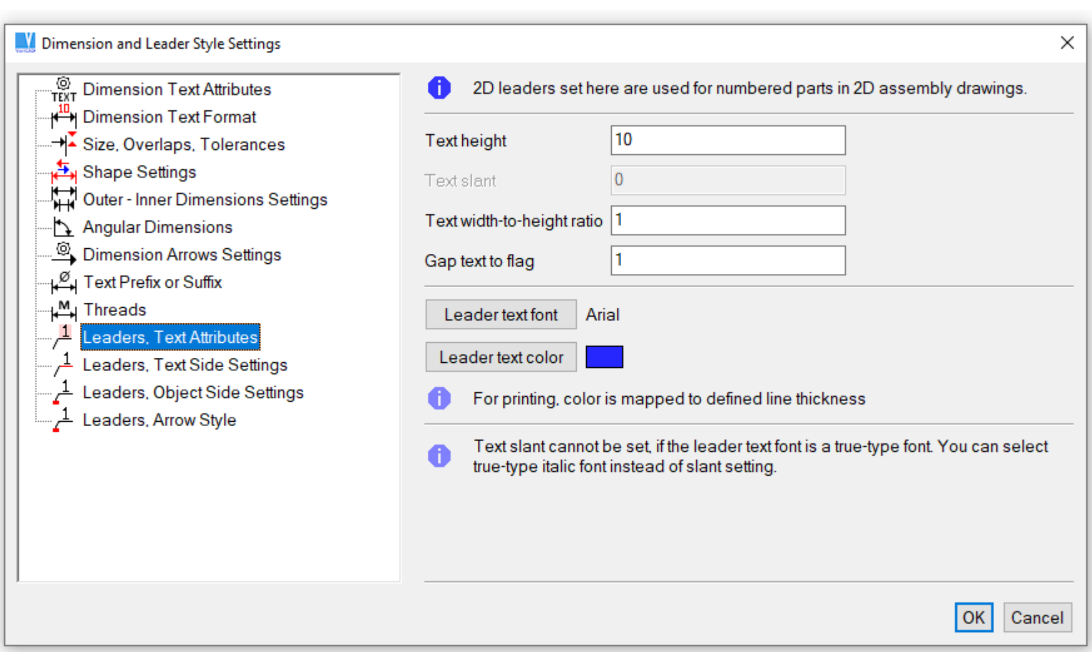
Creates temporary leading lines at the selected leader. This allows you to keep the next leader adjusted horizontally or vertically according the selected leader.



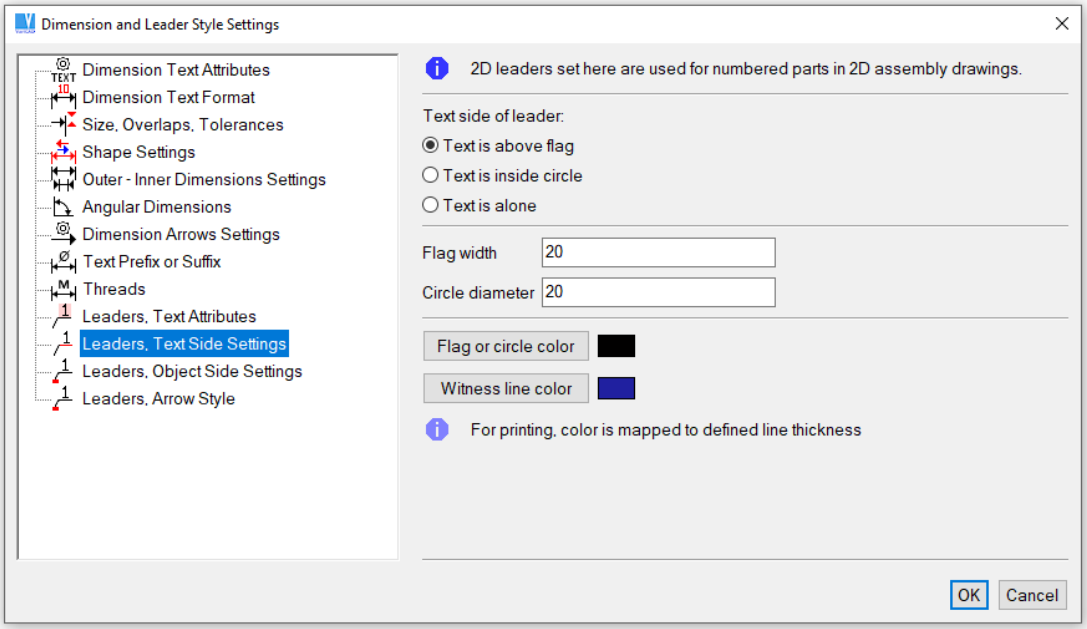
Leader witness line is a spline. This switches witness line creation from a single line to a spline.



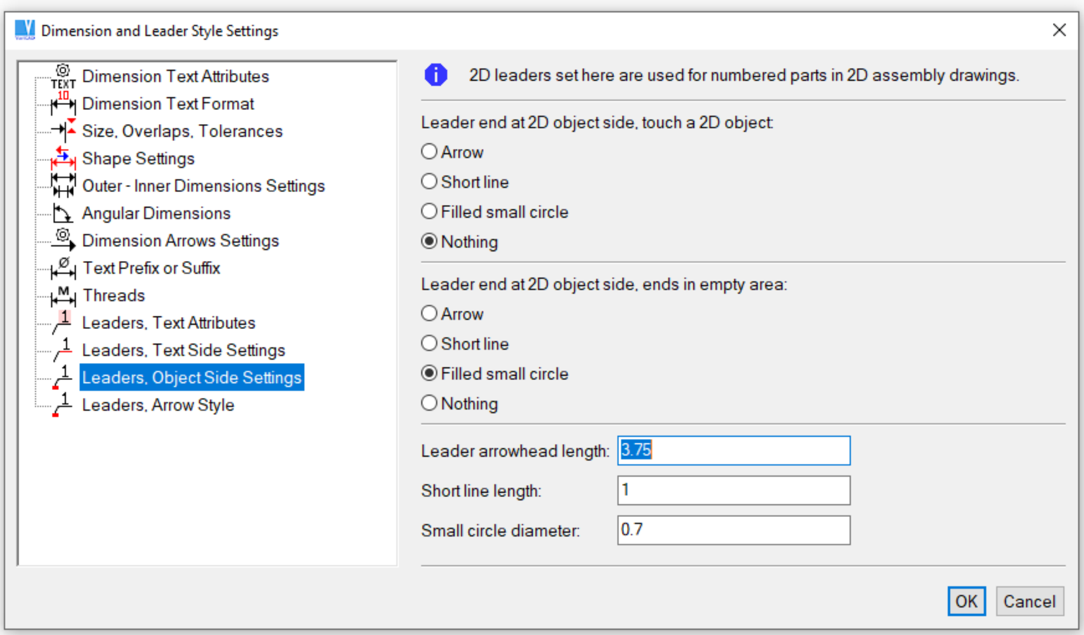
Leader witness line is a single line. It turns off witness line created as a spline.



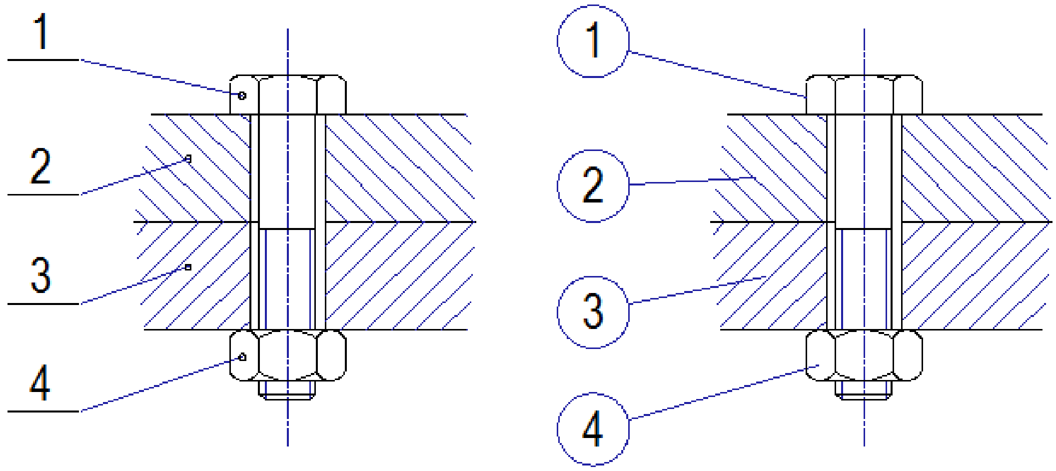
Leader style definition, text style



Leader style definition, text side of leader



Leader style definition, object side of leader

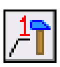


Example of leaders

Editing Leaders

To change existing leaders, right-click a leader and select action from pop-up menu.

 Edit leader. This allows you to change leader shape the same way, as it is created.

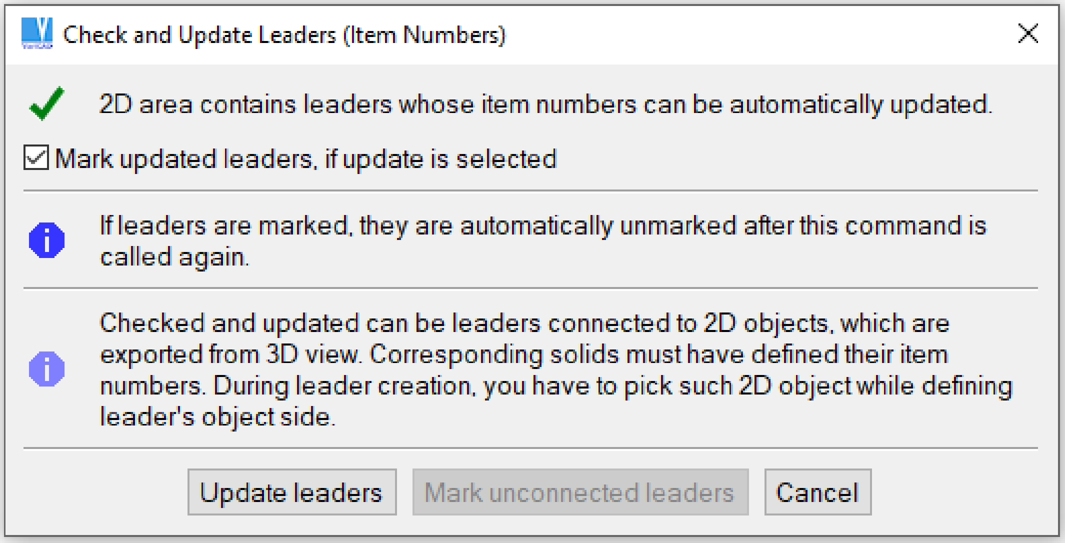
 Edit leader text.

Checking Leaders, Updating Leaders

If you defined item numbers in 3D and a 2D drawing containing leaders was created, you can change item numbers in 3D again. In 2D drawing, leaders are correctly rewritten or checked by following command:

 **Check and Update Leaders (Item Numbers) - CHLDR**

If there are leaders to be updated, following dialog appears:



Update leaders

After confirmation, leaders are correctly rewritten and optionally highlighted. Highlighting of updated leaders is turned off by calling the update command again.

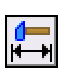
Editing Dimensions

Edit Dimension Text - EDI

Edits the text of a selected dimension. The same text editing options are available as when you created the dimension.

Move Dimension Text - MDT

Uses drag and drop to move the text of a selected dimension. Be careful when using this function, because if you create more dimensions after moving the text, you could have overlapping objects.



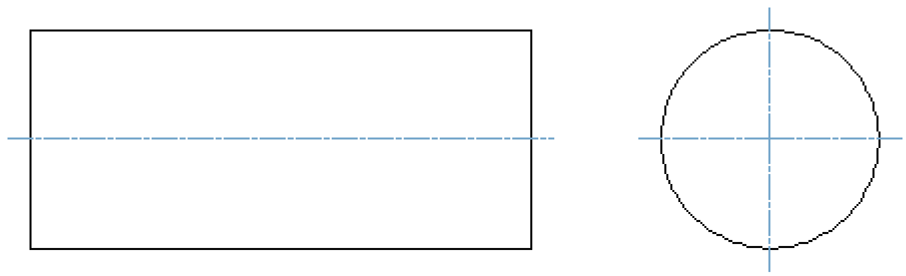
Edit Dimension - EDM

Change any attribute of a dimension. You can change everything about a dimension except for the original dimension definition points or the 2D dimensioned object. Dimension shape is defined the same way as if you create a new dimension. Similarly, you can change style, text or dimension shape.

Axes

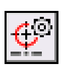
Axes can be created in several ways:

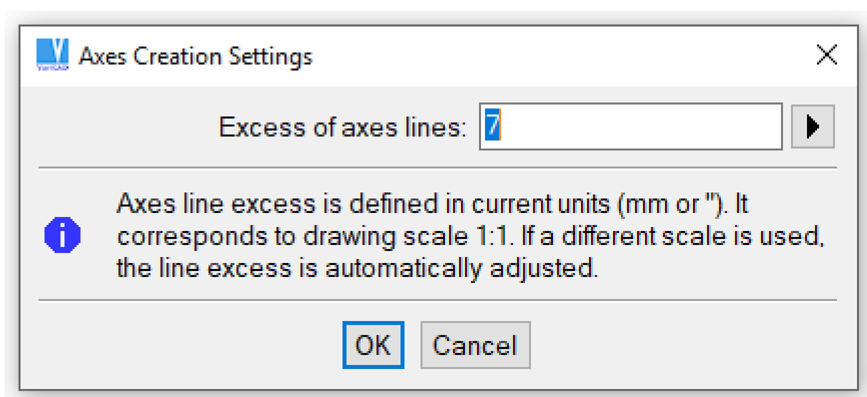
- Two points - the axis is defined as a line between two points, with an extension.
- Arc or circle - a set of four semi-axes will be created at the circle/arc center point.
- Axis of rotation surface exported into 2D
- Pitch circle



Example of axes created at center of circle, and between two points.

If axes are created in 3D views exported from 3D space into 2D area, connections between axes and corresponding 3D objects are established. After changing 3D geometry and returning back to 2D mode, axes are automatically updated. See *Automatic updates of dimensions, axes and hatches after changes in 3D (page 112)*

 Creating axes, you can set excess of line ends. Click the corresponding icon in tool-bar.

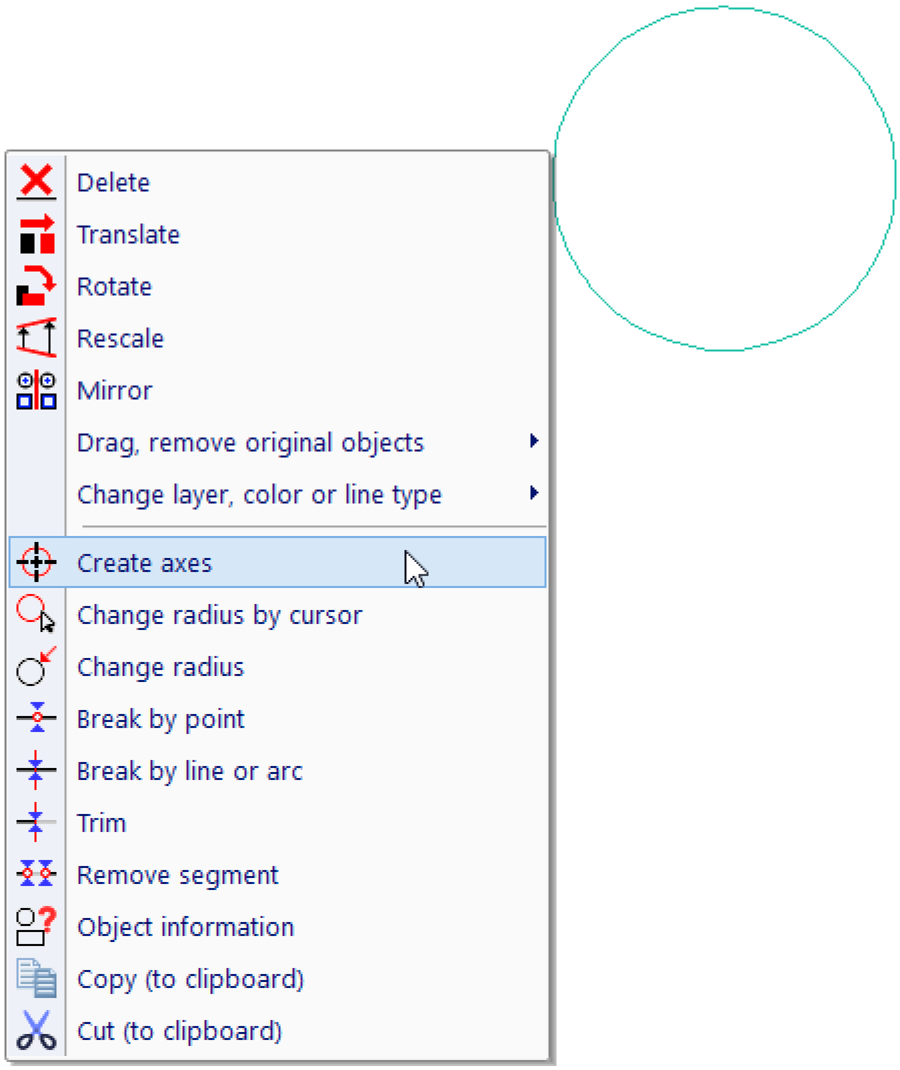


Excess of axes settings

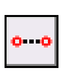
Axes can be created by following commands:

Axes of Circle or Arc - CAX

Select a circle or arc, axes are created as four semi-axes in vertical and horizontal direction. Or right-click an arc or circle and select axes creation from pop-up menu.

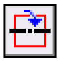


Selecting axes creation from pop-up menu.



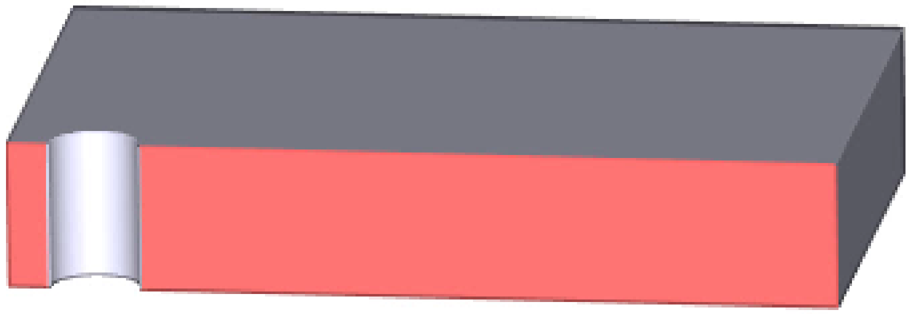
Axis by 2 points – AX2P

Select first point and then second point. Axis is created as connection of selected points with defined excess. If axis is created in 3D view exported into 2D, do not use this method for creation of axes of rotation surfaces. Otherwise, axes may not be updatable after changes in 3D.

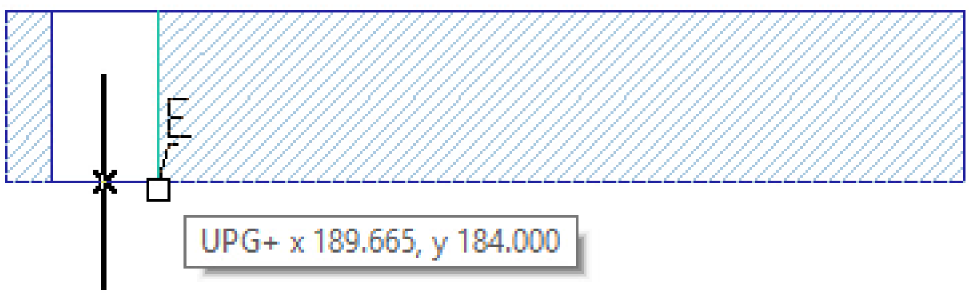


Axis of Rotation Surface - LAX

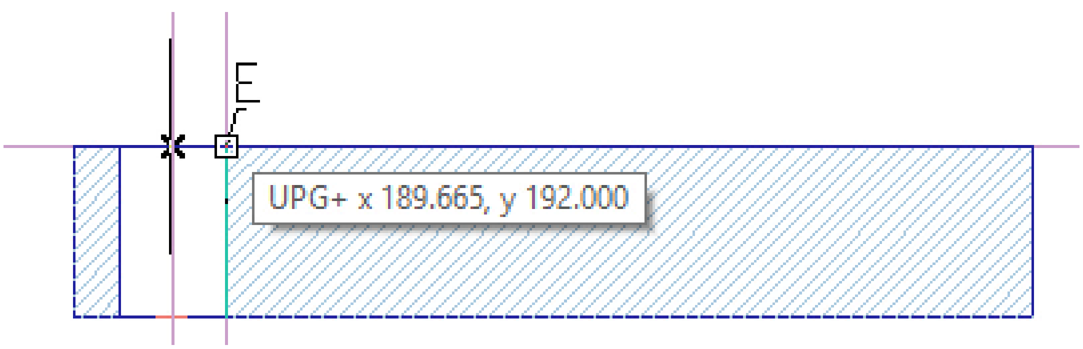
This method is intended for axes of rotation surfaces exported from 3D into 2D views. Axes points are selected, if you move cursor over end of line or circle created by outline of rotation surface. Points are highlighted together with axis segment. Left-click selects highlighted point. See images below.



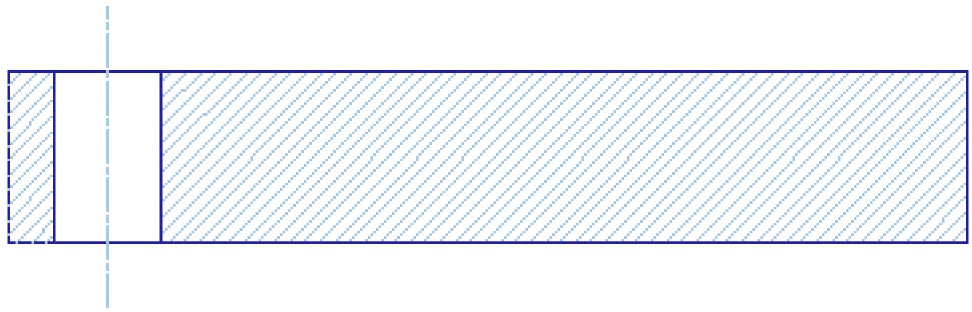
A solid with section turned on, the axis is created as axis of hole.



Selecting first point of axis



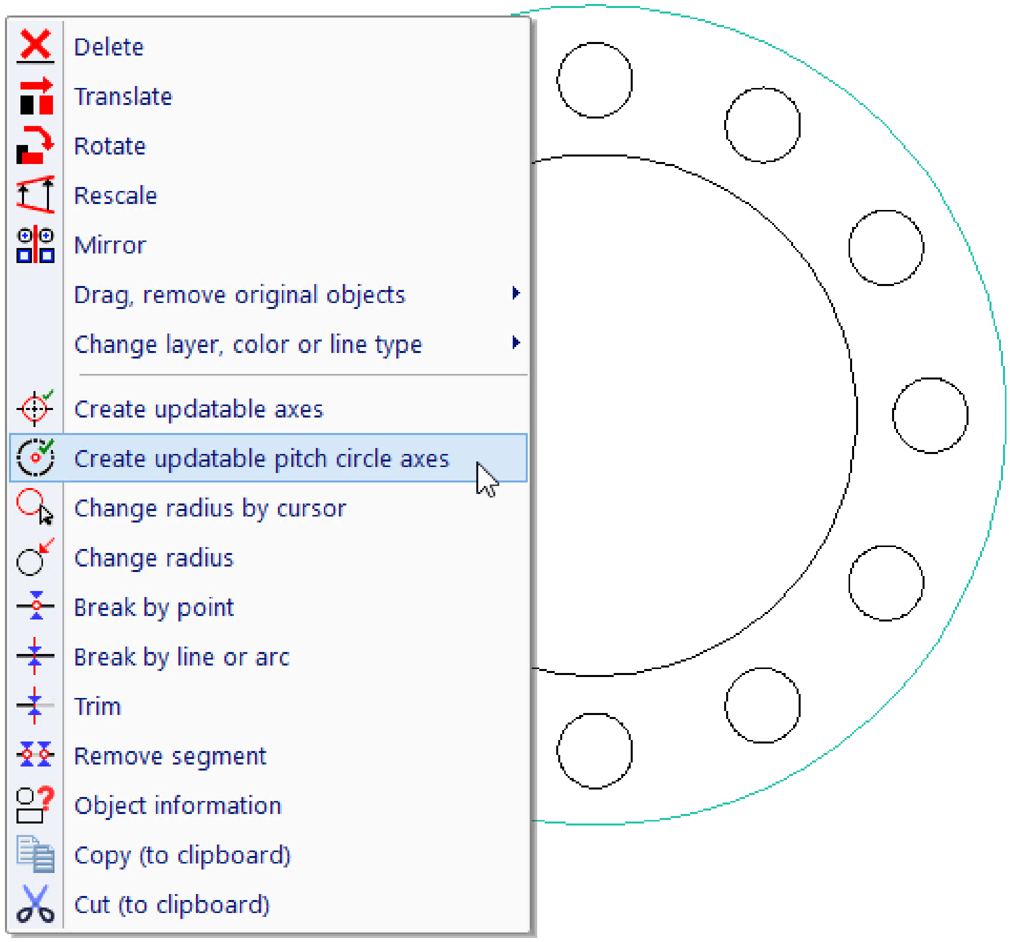
Selecting second point of axis, leading lines are displayed automatically



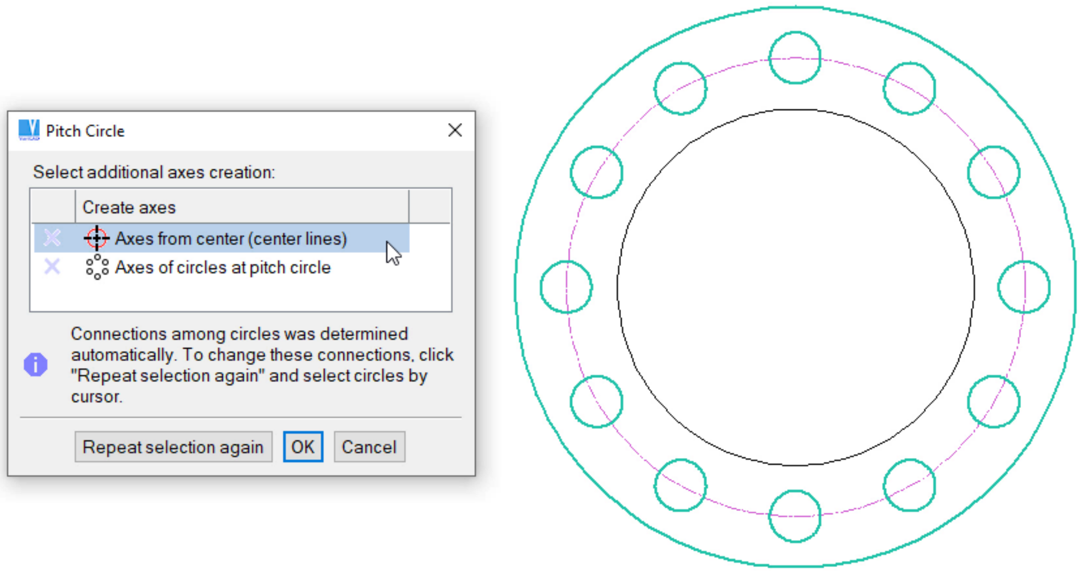
Created axis

Pitch Circle - AXPC

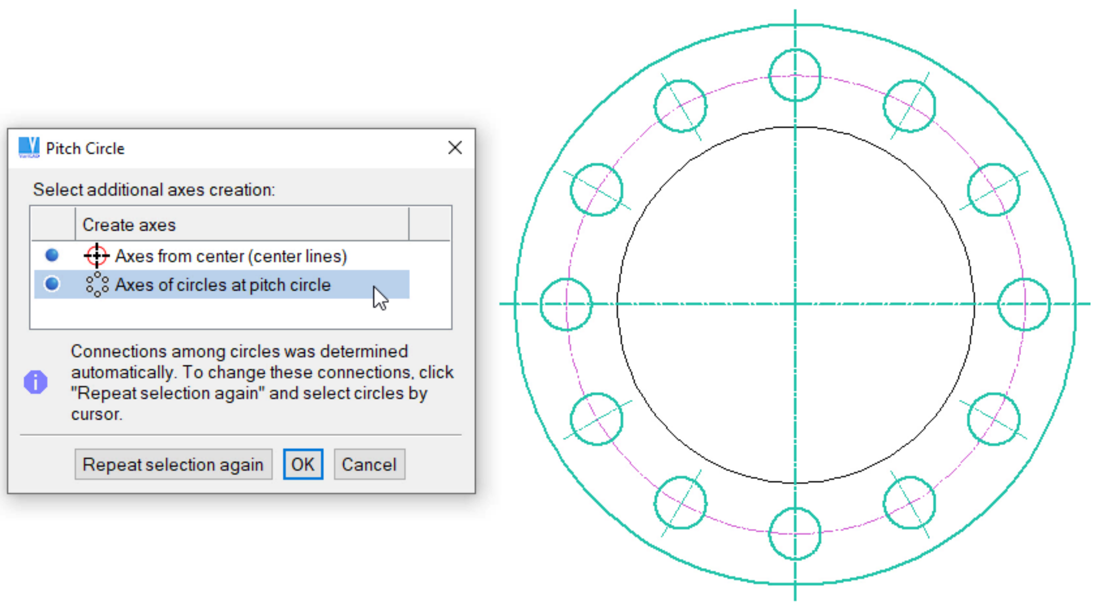
Select an outer circle or inner hole, then select a circle located at pitch circle. Create pitch circle and additional axes. Or, you can right-click outer circle, inner hole or a circle located at pitch circle. Select creation of pitch circle from pop-up menu.



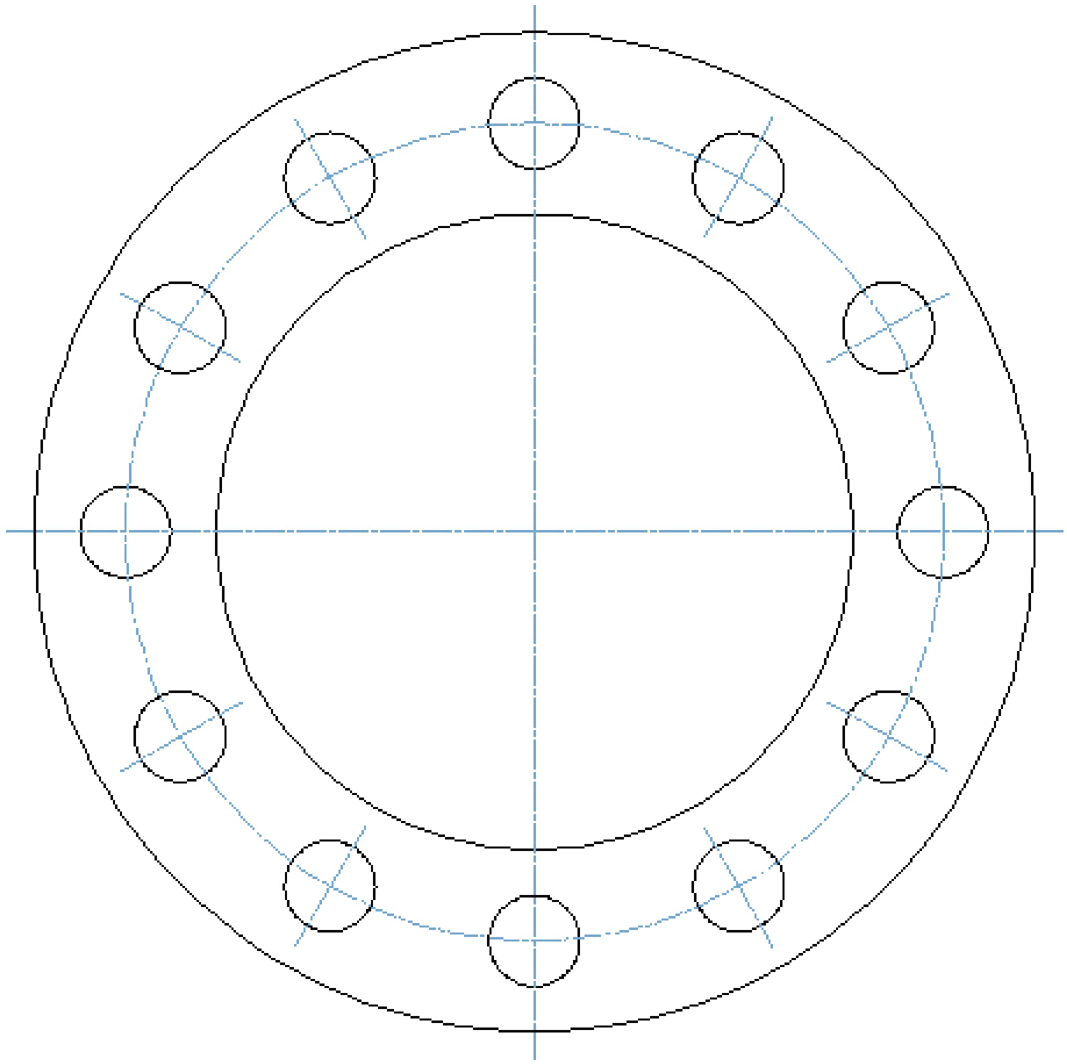
Selecting pitch circle creation from pop-up menu.



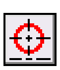
Pitch circle is displayed and detected circles are highlighted. Select axes from menu.



Creating additional axes.



Pitch circle and selected axes are created.

 **Axes - AXI**

This function displays a pop-up menu with all four possibilities.

Hatching

Hatching functions, typically used to identify sections, are used to fill a closed area. A simple hatch consists of lines at a specified angle and separation distance. You can also select a predefined hatch pattern and density, or create your own pattern. Hatch boundaries can be defined by selecting individual segments, by automatic boundary detection, or by automatic detection of 3D section boundaries exported to 2D. Hatch boundaries must be closed, and any entities that are partially inside the boundary are not used. Hatch boundaries may contain islands.

If hatches are created in 3D views exported from 3D space into 2D area (hatching of 3D sections), connections between hatches and corresponding 3D objects are established. After changing 3D geometry and returning back to 2D mode, hatches are automatically updated. See *Automatic updates of dimensions, axes and hatches after changes in 3D (page 112)*

Solid Fill

To use a solid fill, choose the horizontal line pattern and define a hatch distance smaller than the thickness of printed lines.

Hatching 2D Objects

Hatch, Select Boundaries - HAT

Boundaries of hatch area are selected by cursor. Optionally, you can use:

Detect individual area segments – select a segment and a direction the boundary segments will be detected at. Confirm each segment by mouse click. Detection is stopped after the boundary is completed or at the open end of a segment.

Detect continuous area boundary – select a segment and a direction the boundary segments will be detected at. Complete boundary is detected, or detection is stopped at the open end of a segment.

Temporary segment closing area – allows creating a line, lines, circle or arc temporarily. It may be used for closing an open area or creating islands around a text.

Temporary object – this is similar as the previous option except the temporary object is not pre-selected. It can be used conveniently if you need to create an island around a text and the text partially crosses the existing area boundary.

Temporary removes restraining object.

Automatic 3D section detection – turns detection of section boundaries on/off. This option allows hatching of 3D sections conveniently. Selection of 3D section boundaries should be combined with other options, because section boundaries itself may be partially obscured or may contain cross-lines. The option is obsolete and kept as legacy option. Use rather the next possibility.

Updatable 3D sections detection. Unlikely the previous option, this detects entire hatch boundary created by export of 3D section. Cross lines are automatically removed.

Blocks automatic detection of selected line types. This option automatically excludes detection of axes or invisible lines drawn in non-continuous style, for instance.

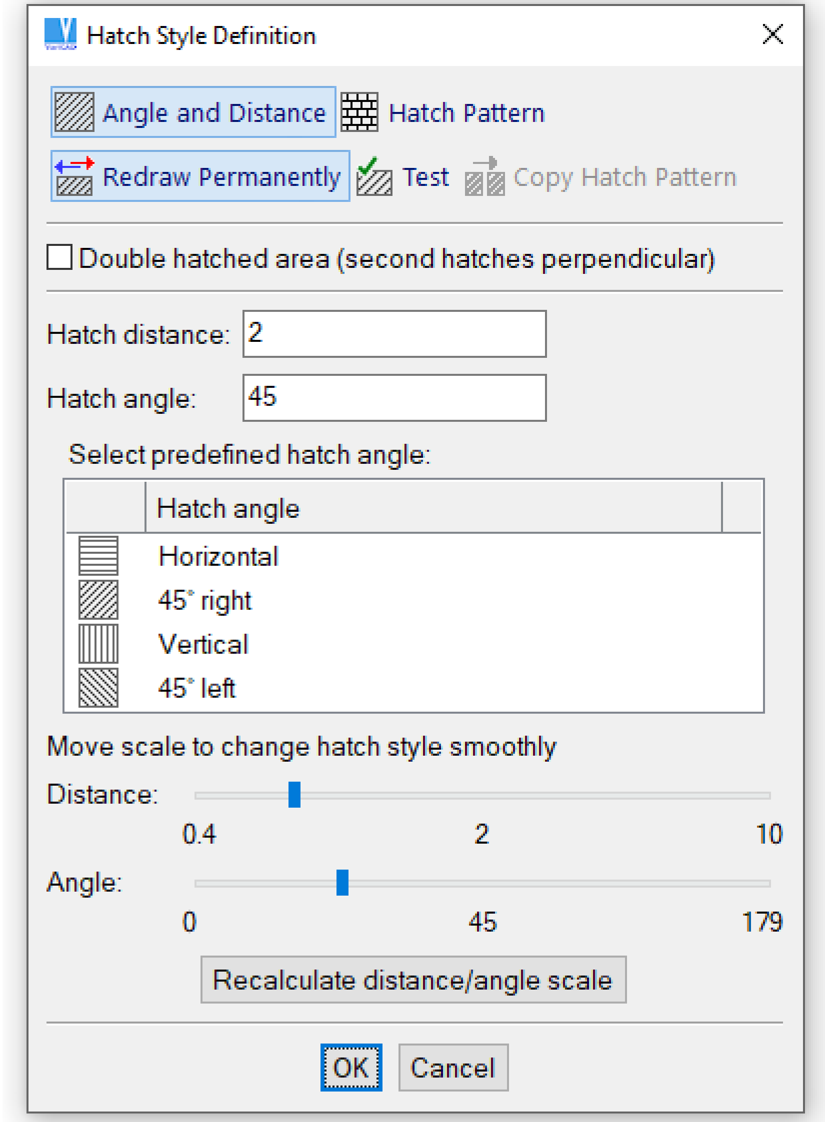
Hatch, Detect Boundaries Automatically - AHB

Contrary to the previous command, this hatching method detects closed hatch boundaries automatically during cursor movement. Automatic detection can be turned on/off. All other options available are the same as for previous command.

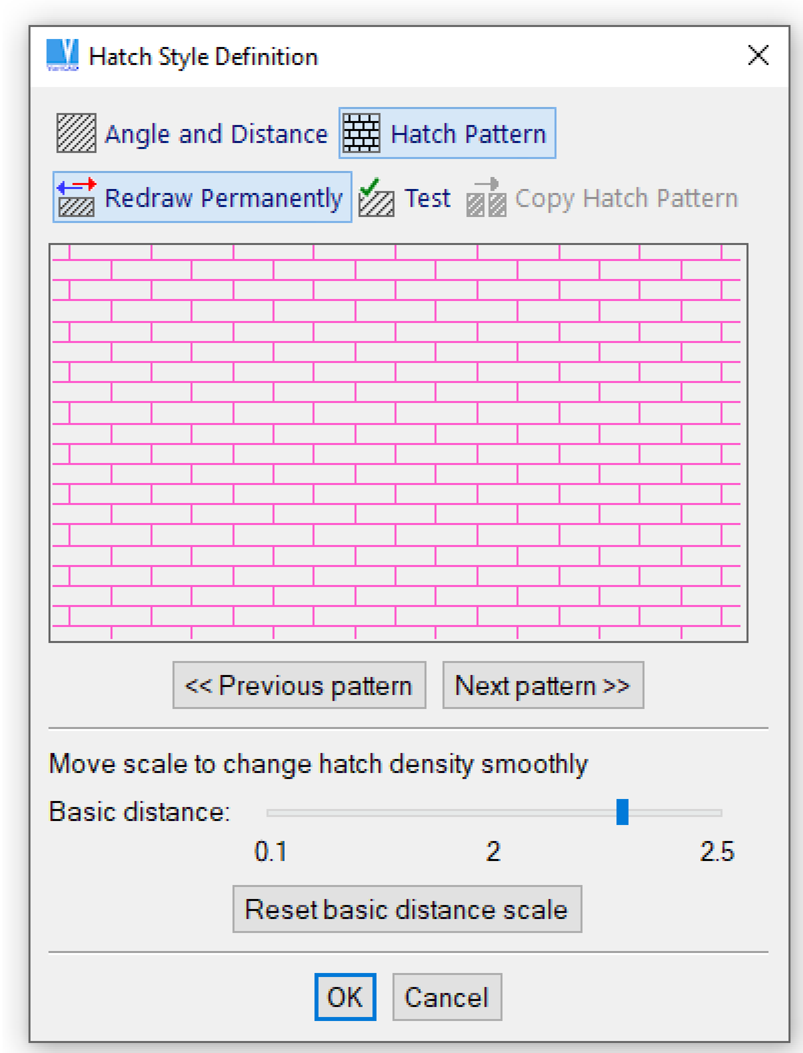
Hatched area boundaries are mostly created by continuous lines. By default, other line styles are excluded from automatic boundaries detection. You can turn on detection of non-continuous lines, if you click a corresponding icon in selection tool-bar, among additional options.

Selecting Hatch Style

You can choose between single hatching and a hatch pattern. If the single hatching is selected, an angle and a hatch distance are defined. In both cases, you can move the slider in hatch style window to change the hatch density continuously. Click button "Recalculate angle/distance scale" to change the hatch density slider margins.



Single hatch selection

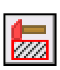


Hatch patterns selection

Editing Hatches

 **Change Hatch Style - CHHP**

Changes style of existing hatches. Style definition is the same as for newly created hatches.

 **Change Hatch Area or Style - CHH**

This command changes style of existing hatches similarly as the previous command. Moreover, it redefines the hatch area boundary. Initial state of the new boundary is defined by the location where you clicked hatches to select them. Boundary can be added or deleted the same way as for newly defined hatches. Command can be conveniently used if the shape of the hatched area was modified.

Creating a Hatch Pattern

Create Pattern - CHP

Create your own pattern or edit existing patterns. Each pattern is created as up to eight groups of lines. Each group is defined by an angle and a basic distance. You can define some lines in the group to be blanked. Each line can be assigned a line style, consisting of a number of line segments of specified length.

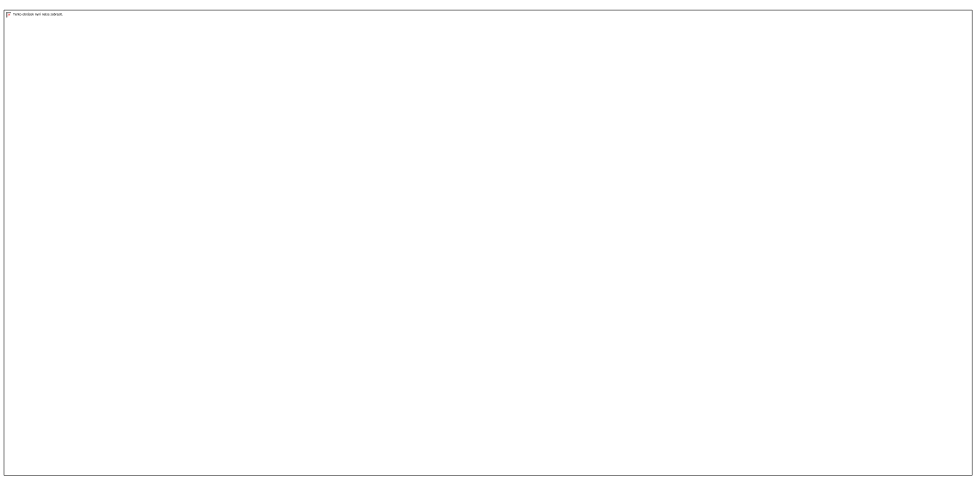
Symbols

Symbols are sets of 2D objects. Symbols are defined by:

- An insertion (reference) point
- Connection points - used after insertion for snap locations
- Name and comment

You can insert symbols from standard libraries or create your own symbol libraries. The standard VariCAD package offers libraries containing hydraulic symbols, pneumatic symbols, and electrical symbols.

Welding, tolerance and finish symbols are managed by different functions, see *Dimensioning (page 79)*.



Working with a symbol library

Symbols - SYM

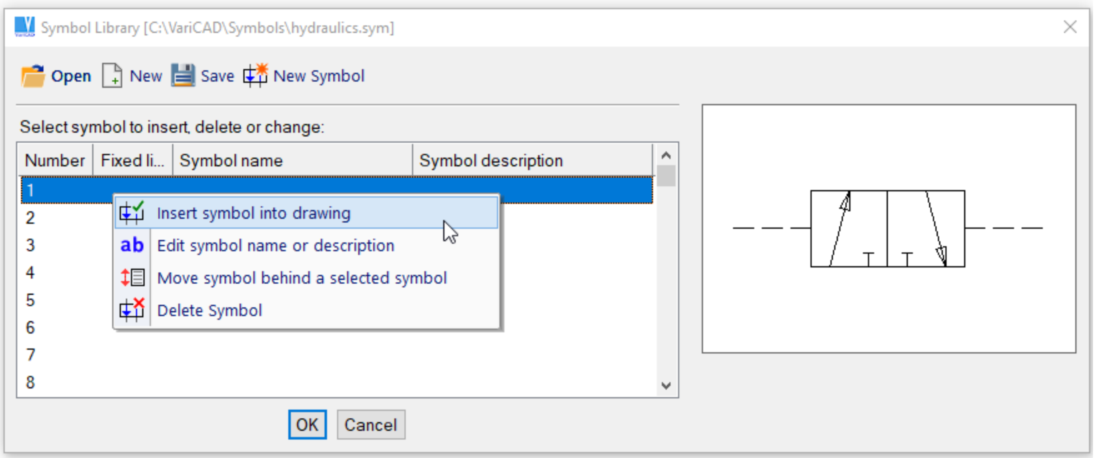
This command creates edits or deletes symbols from the open library. If no library is open, select the existing library first or create a new one. Only one symbol library can be open at a time. Symbols available from VariCAD distribution package (like pneumatic or hydraulic symbols) are inserted into drawing by different method – from menu “Parts”.

Following options are available for working with symbols:

- New symbol – creates a new symbol and inserts it into currently open symbol library. To create a symbol, select 2D objects (lines, arcs, texts…), then define an insertion point, connection points and a name and optionally a description.

Clicking an existing symbol in symbol list, you can:

- Insert the corresponding symbol into 2D drawing.
- Edit symbol name or description.
- Move symbol into different position within symbols list.
- Delete the symbol.



Working with symbols

2D Blocks

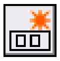
Objects created in 2D can be integrated into blocks. Blocks behave and are selected as single objects. The only time individual objects can be selected is when changing object color or line type. Connection points are used to define snap points for the block. A block is defined by:

- 2D objects, including nested blocks
- Insertion point
- Connection points (up to 32)
- Name and attributes

Blocks can be inserted into the current file, can be stored and inserted into other files.

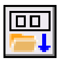
There are libraries of mechanical parts (screws, bearings, threads, etc.) you can insert into your drawings. These are inserted as blocks, with predefined attributes. For more information, see *Libraries of Mechanical Parts*.

Creating and Inserting Blocks



Create Block - BLC

To create a block, define the insertion point, any connection points, 2D objects that comprise the block, and block name and attributes. You can use one of the block attributes as the block name, and you can select whether the attribute will be blanked or unblanked. For visible attributes, the location of attribute text must be defined.



Save Block - BLS

Saves the selected objects to a file. Enter the filename, define the insertion point, and select the block objects to be saved. It is recommended to select only one object.

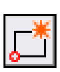
Insert Block – BLI, Ctrl + K

Inserts saved blocks into the drawing area. You can select blocks from a list of saved blocks. Select the location of the insertion point and you can drag the block to new locations. Insert Block is also invoked when you insert a part from a mechanical part library. This type of block is selected from an icon menu and created according to predefined dimensions.

During block insertion, you have the following additional insertion options:

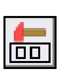
Rotate or Scale - enables you to:

- Rotate the inserted block by a specified angle
- Scale the inserted block by a scale value
- Orient the X axis of the block along or perpendicular to a selected line. The X direction is defined from the insertion point to the right
- Insert the block at the origin
- Scale the block according to the drawing scale (not available for library parts)
- Change units from mm to inches or vice-versa (not available for library parts)



New Insertion Point - changes the block insertion point. Insert the block first into a temporary position and then select the new insertion point.

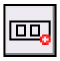
Editing Blocks



Edit Blocks - BLE

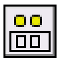
If no block is open for editing and this function is invoked, you can select a block to edit. The selected block is highlighted and you can use any 2D functions to create, edit, or delete block objects.

If a block is currently open for editing and this function is invoked, the block objects are highlighted and inserted into a temporary work set. You can select other objects to add to the block, or select objects to be deleted from the block. See also *Selecting 2D Objects (page 45)*.



Change Block Insertion Point - BIE

Redefine the block insertion point.



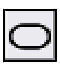
Edit Block Attributes - BAE

Add, edit or delete attributes of the selected block. The attributes list appears after you select the block. You can blank or unblank attributes as well, and for visible attributes you can change text position or text attributes as well.

2D Polylines

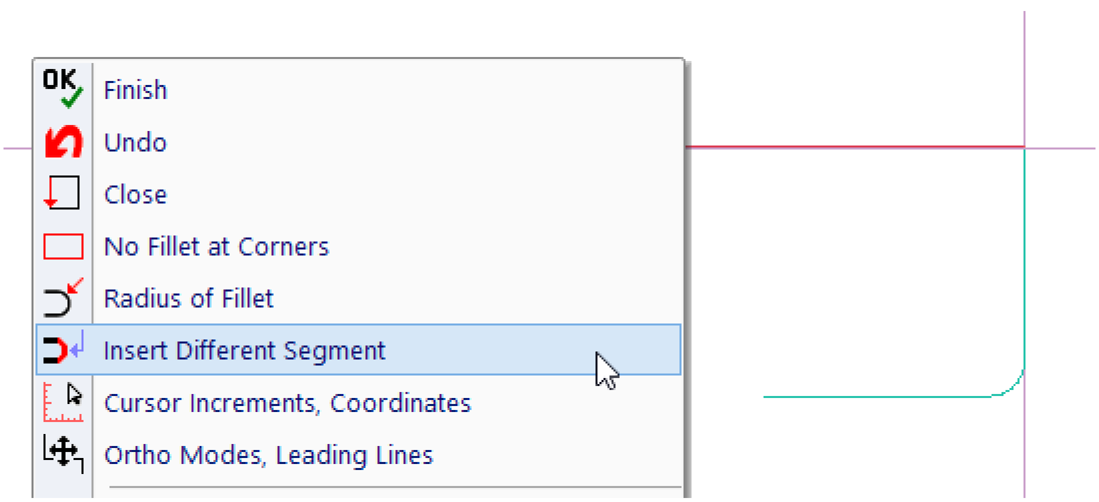
Objects created in 2D can be integrated into polylines, similarly as into blocks. Polylines also behave and are selected as single objects. Unlike blocks, you can integrate only existing lines, arcs or NURBS curves. Moreover, you can draw a new polyline instead integrating it.

Creating Polylines

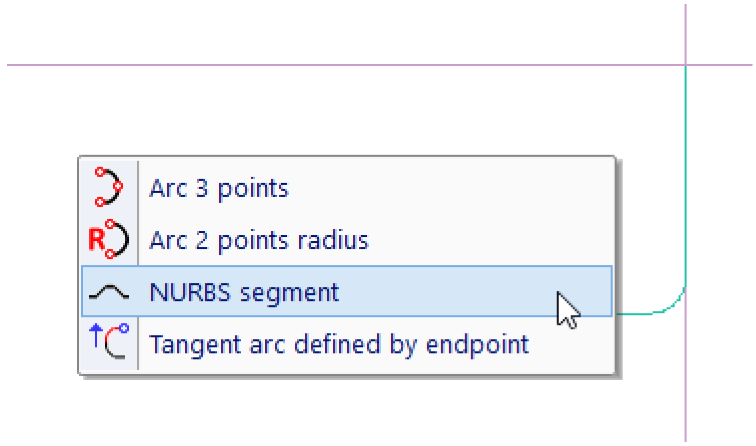


Draw Polyline - CPL

Drawing polyline, you can insert linear segments. Optionally (and by default), each corner is rounded. Also, you can insert a different type of segment. Icons containing polyline options are added into 2D input toolbar, or into pop-up (called by right + left click).



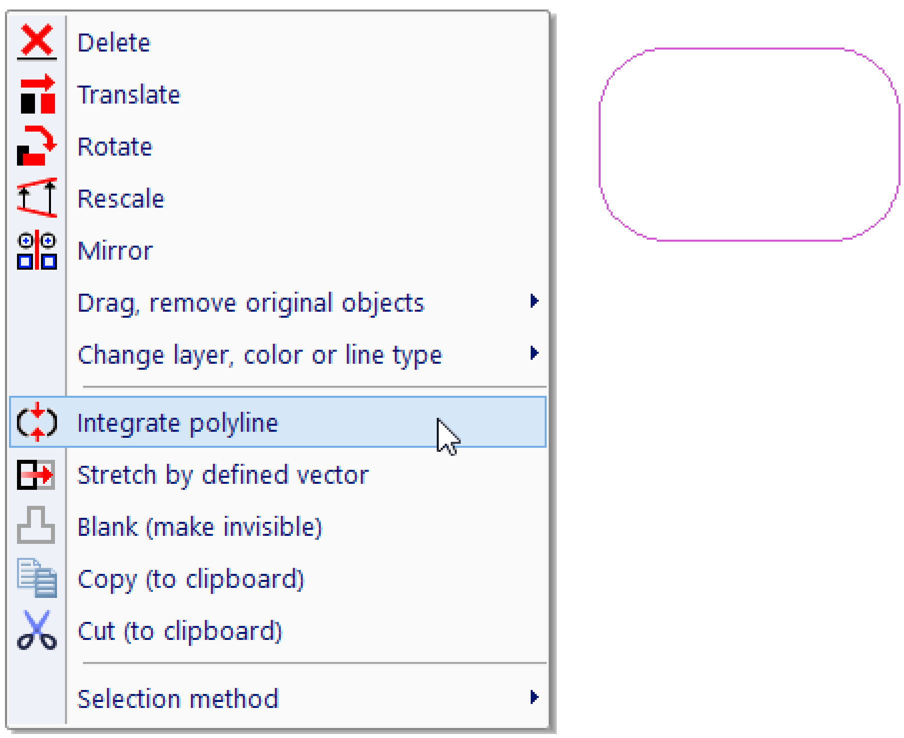
A pop-up containing polyline options. Drawing of polylines uses temporary and transient construction lines.



A pop-up containing options of different types of segments.

 **Join Objects into Polyline - PLL**

You can select existing objects and integrate them into polyline. If you select a set of objects and finish by right-click. If all selected objects can be integrated into polyline, the option is displayed in pop-up menu.



Selected objects, integration into polyline.

Editing Polylines

Polylines are edited the same way as blocks. Either select the command, or right-click a polyline and select editing from menu. See *Editing Blocks (page 108)*. Unlike blocks, polylines do not have defined attributes or insertion point and connection points.

Chapter 8. Automatic Updates of Dimensions, Axes and Hatches after Changes in 3D.

Dimensions, axes and hatches are automatically updated after change in 3D. Connections between these 2D objects and corresponding 3D objects are created within complete 3D view exported into 2D area. VariCAD provides a lot of tools for checking of updatable 2D objects.

Automatic updates of dimensions

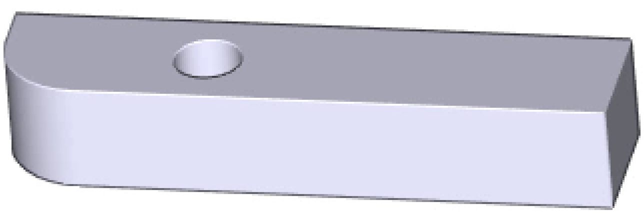
Horizontal, vertical or parallel dimensions are updated automatically, if both dimensioned points are selected within one 3D view, at 2D objects. Detection of updatable location is signaled in cursor tool-tip. In front of coordinates, UPG+ is displayed. If one dimensioned point is not located at 2D object exported from 3D, or if location is not within the same view, the created dimension is not updatable.

Similarly, angular dimensions are updatable, if both detected lines are from same 3D view exported into 2D area. Detection of objects corresponding to updatable dimension is displayed as UPG+ near cursor, too.

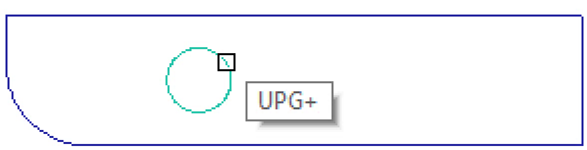
Radius or diameter dimensions are updatable, if dimensioned circle or arc belongs to 3D view exported into 2D. Finish symbols are updatable, if they are located at arc or line from 3D view. Again, detection of objects corresponding to updatable dimension is displayed as UPG+ near cursor.

If a dimensioned point is located at end-point of axis, the dimension is updatable if the axes is updatable, too.

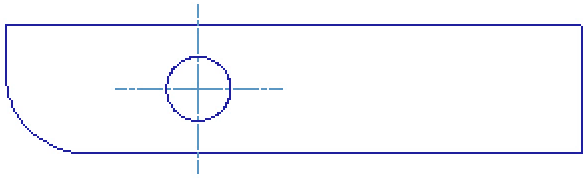
Following images show example of updatable dimension creation.



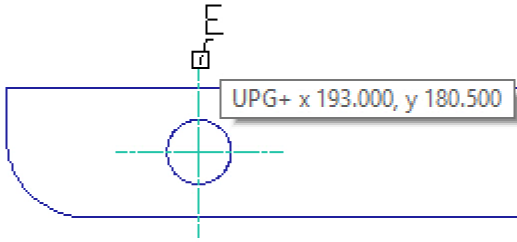
Top view of the solid will be created and dimensioned.



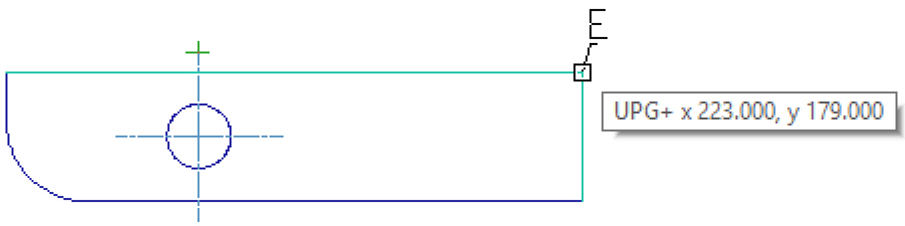
Creation of circle axes. A circle is detected and toolbar near cursor shows connection to 3D (UPG+).



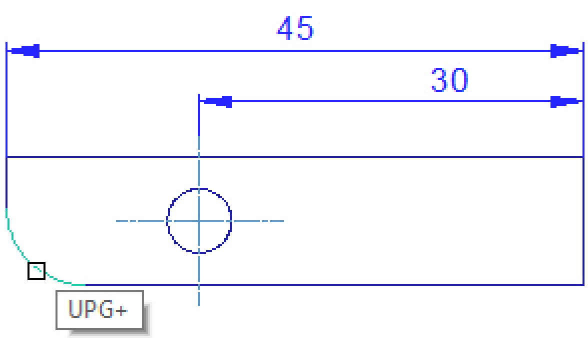
Created circle axes.



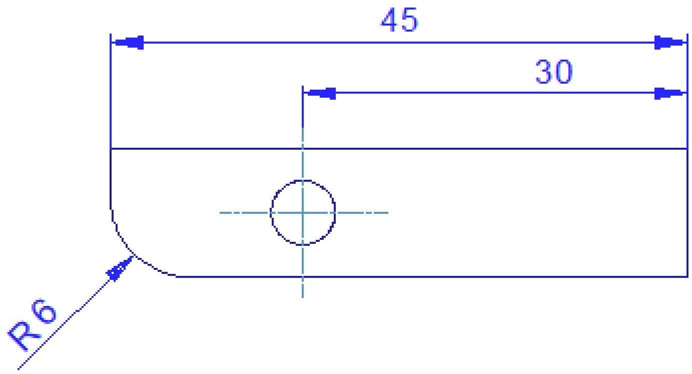
Creation of a horizontal dimension. First location (point) is detected at end of updatable semi-axis. Tool-tip near cursor shows connection to 3D (UPG+).



Second location (point) of horizontal dimension is detected.



Creation of radius dimension. An arc is detected. Tool-tip near cursor shows connection to 3D (UPG+).

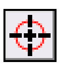


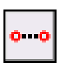
Finished dimensions.

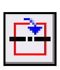
See how these dimensions are updated after change of 3D shape. (page 120)

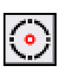
Automatic Updates of Axes

Created axes are updatable, if corresponding 2D objects are updatable, too.

 Axes of circle or arc are updatable, if the corresponding circle or arc are updatable. See previous section – *creation of updatable circle axes (page 112)*.

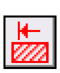
 Axis connecting two points with defined excess is updatable, if both points are located at updatable 2D objects. For instance, you can detect a mid-point, circle quarter etc. In case of axes creation within 3D view, axis by two points is intended as axis of symmetrical objects or row of holes, but not as axis of rotation surface

 Axis of rotation surface. This axis cannot be created, if lines corresponding to rotation surface are not part of 3D view. This type of axes is created as updatable. See *Creation of axes of rotation surface* how to create such type of axes.

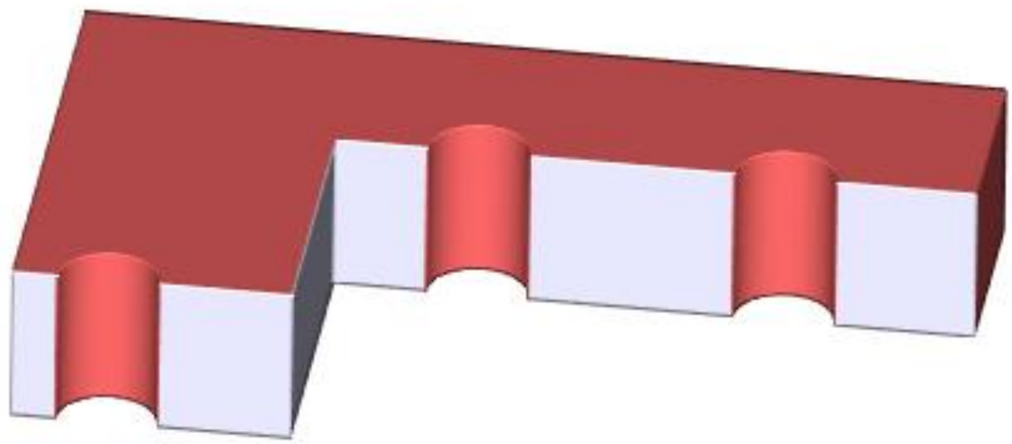
 Pitch circle is updatable, if all automatically detected circles at pitch diameter are updatable. Also, outer diameter surrounding these circles, or inner hole must be updatable. See *Creation of pitch circle (page 95) (page 27)* how to create such type of circles.

Automatic Updates of Hatches

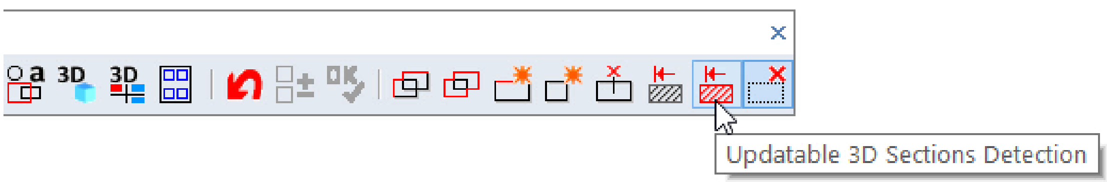
Hatches in 3D view are updated automatically, if their boundaries are selected completely as outline of a 3D section. Also, you can use corresponding option to detect such boundaries automatically:

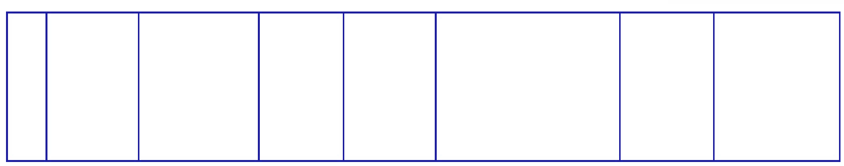
 Updatable 3D sections detection

Example of updatable hatches creation:

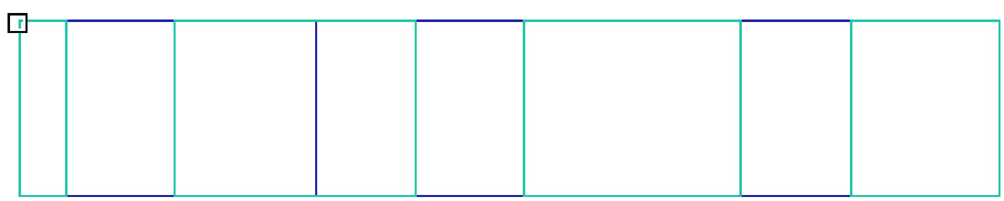


A 3D solid with section turned on. Front view is exported into 2D area.

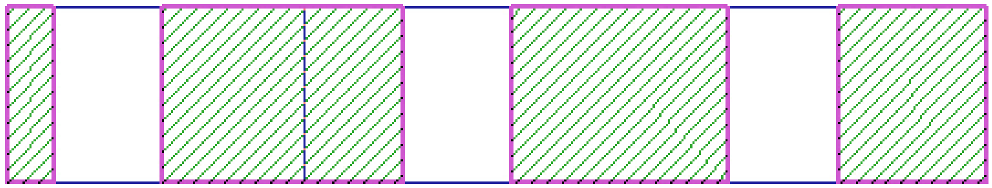




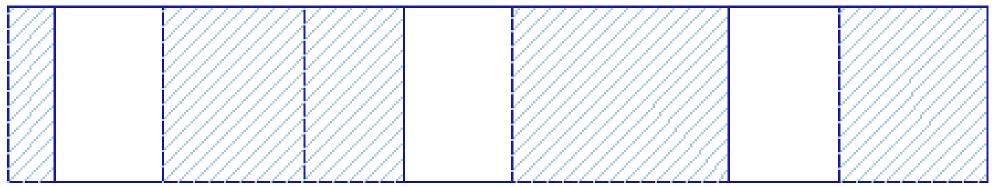
This option allows to detect boundaries of entire section at one step.



Boundaries of 3D section are detected



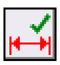
Confirmation of hatches



Created hatches of 3D section. After changes of 3D model, these hatches are rebuilt automatically.

Checking of Updatable 2D Objects and Dimensions

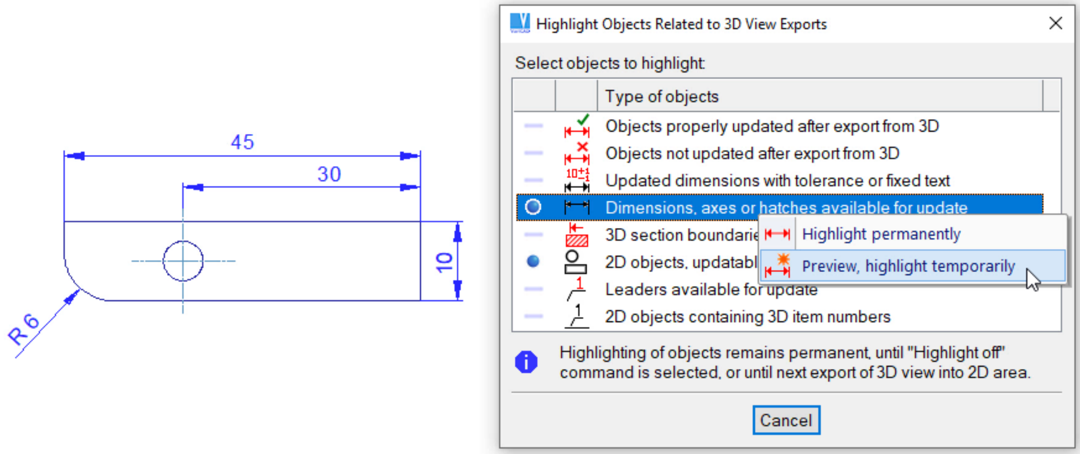
VariCAD provides tools for checking of updatable objects and checking of properly or improperly updated dimensions, axes or hatches. These objects can be temporarily or permanently highlighted.

 **Highlight Objects Related to 3D View Exports HOD**

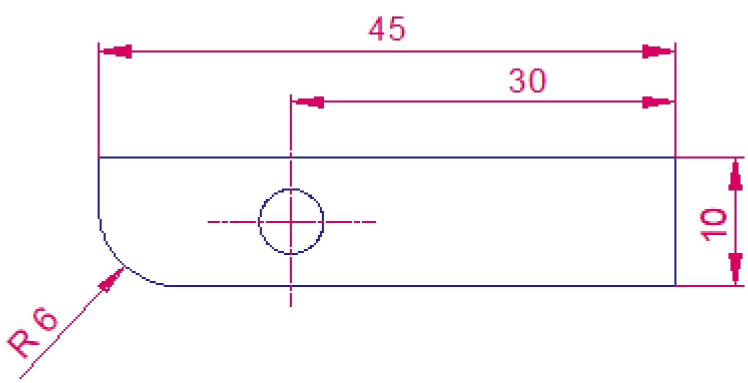
Selected type of objects can be highlighted temporarily or permanently. You can highlight:

- Objects properly updated after export from 3D
- Objects not updated after export from 3D
- Updated dimensions with tolerance or fixed texts
- Dimensions, axes or hatches available for update
- 2D objects, updatable dimensioning
- Leaders available for update
- 2D objects containing 3D item number – updatable leaders are created

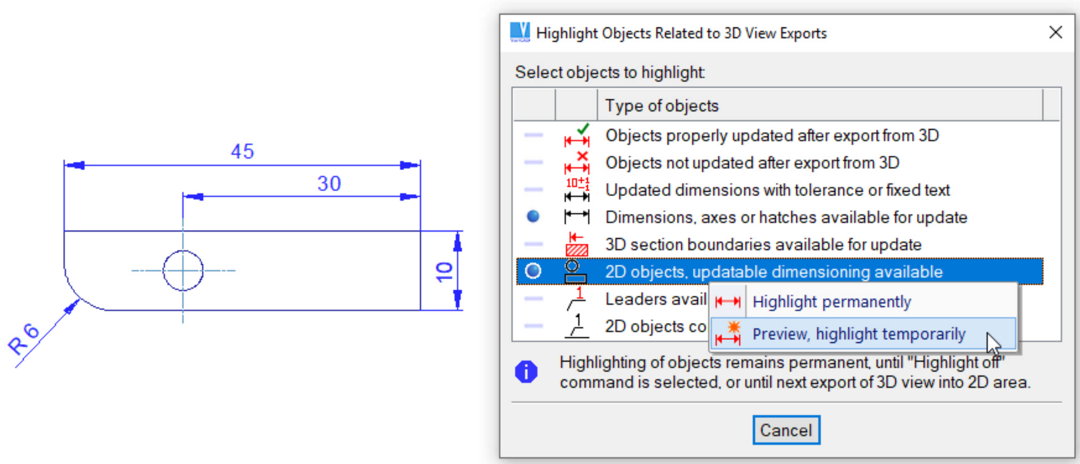
Updated dimensions with tolerances or fixed texts can be checked after automatic changes, so you can highlight them to make checking easier.



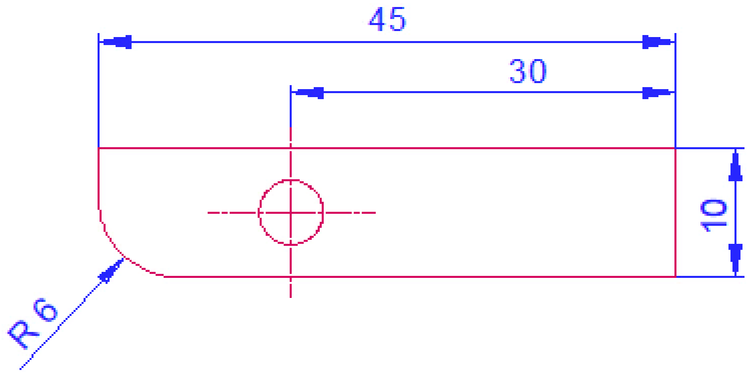
Example of selection of highlighting – automatically updated dimensions and other objects.



Selected type of objects is highlighted.



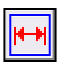
Example of selection of highlighting – 2D lines, curves, circles or arcs created by 3D view export.



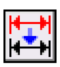
Selected type of objects is highlighted. Dimensions connected to them will be updated automatically.

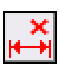
 **Highlight Objects Off - OOD**

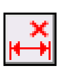
If objects are permanently highlighted – in command described above, or automatically as not-updated dimensions, you can turn highlighting off. Select whether turn off highlighting in entire drawing or in selection window.

 **Zoom in on Highlighted Dimensions - ZOD**

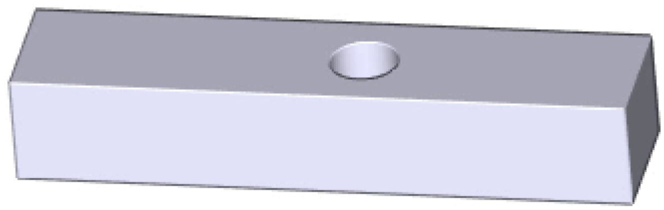
If objects are permanently highlighted – in command described above, or automatically as not-updated dimensions, you can zoom in on each individual object. This option may be useful in case of large drawings containing a lot of dimensions. Some of not updated dimensions can be neglected. Automatic zooming makes easy to handle each unsolved situation.

 Highlight off selected objects. This option appears, if you select 2D objects and some of them are highlighted. Click the option “For highlighted, highlight off” to turn highlighting off

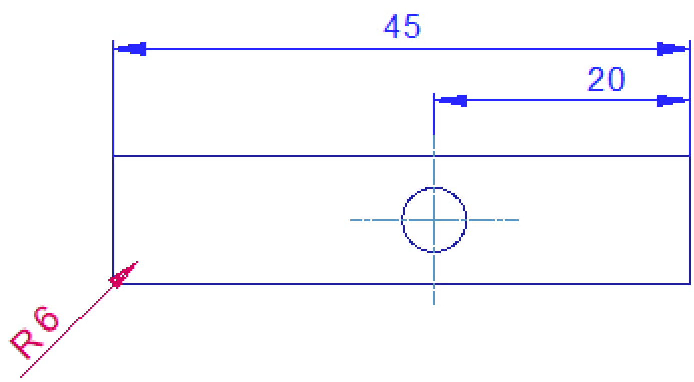
 Remove 3D connections of selected objects. This option appears, if you select 2D objects and some of them cannot be updated. Click the option “For improperly updated, remove connections to 3D” to turn 3D connections off

 **Remove Objects Unable to be Updated - ROD**

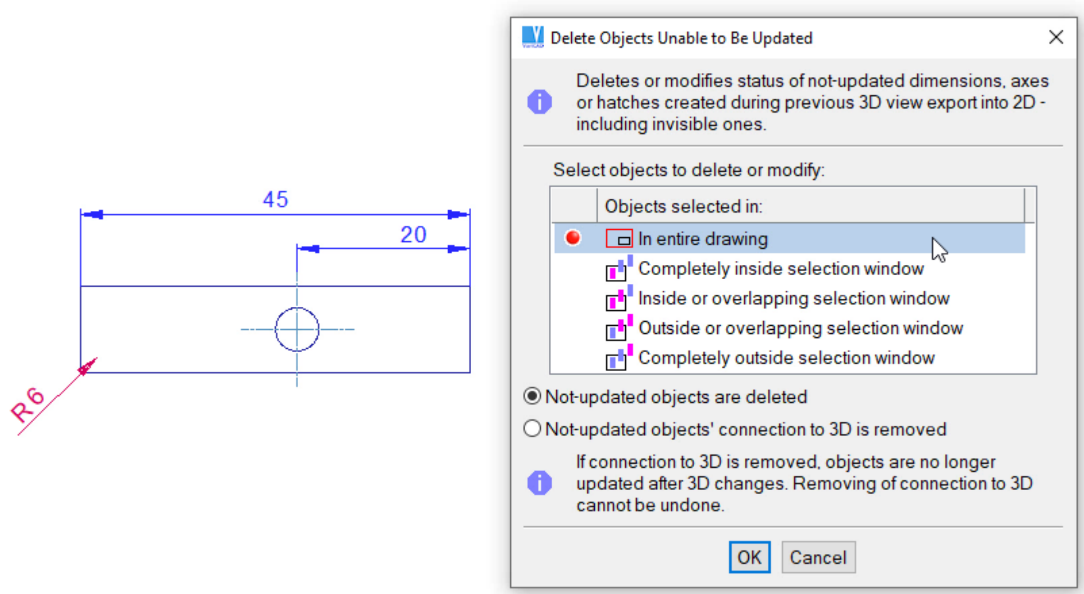
After changes in 3D and switch mode into 2D, some dimensions, axes or hatches cannot be updated. Typically, it is if corresponding 3D objects were removed or replaced. A solid in following example was dimensioned *here*. Then, shape was changed and dimensions were updated.



A changed solid, already dimensioned.



Horizontal dimensions and axes of circle were adjusted properly after the corresponding hole was moved. Dimension of radius is automatically highlighted and not updated, after deleting of corresponding fillet.



Removing of objects unable to be updated. Command options are displayed in dialogue panel.

Chapter 9. Libraries of Mechanical Parts

All mechanical parts can be found on the Parts menu. The following parts are available:

- Screws (bolts)
- Nuts
- Washers
- Pins and splines
- Rings
- SKF bearings
- ZKL bearings
- Rolled profiles (structural steel, beams)
- Flanges
- Spline shafts (only for 2D)
- Threads (only for 2D)

The parts are created according to the following standards:

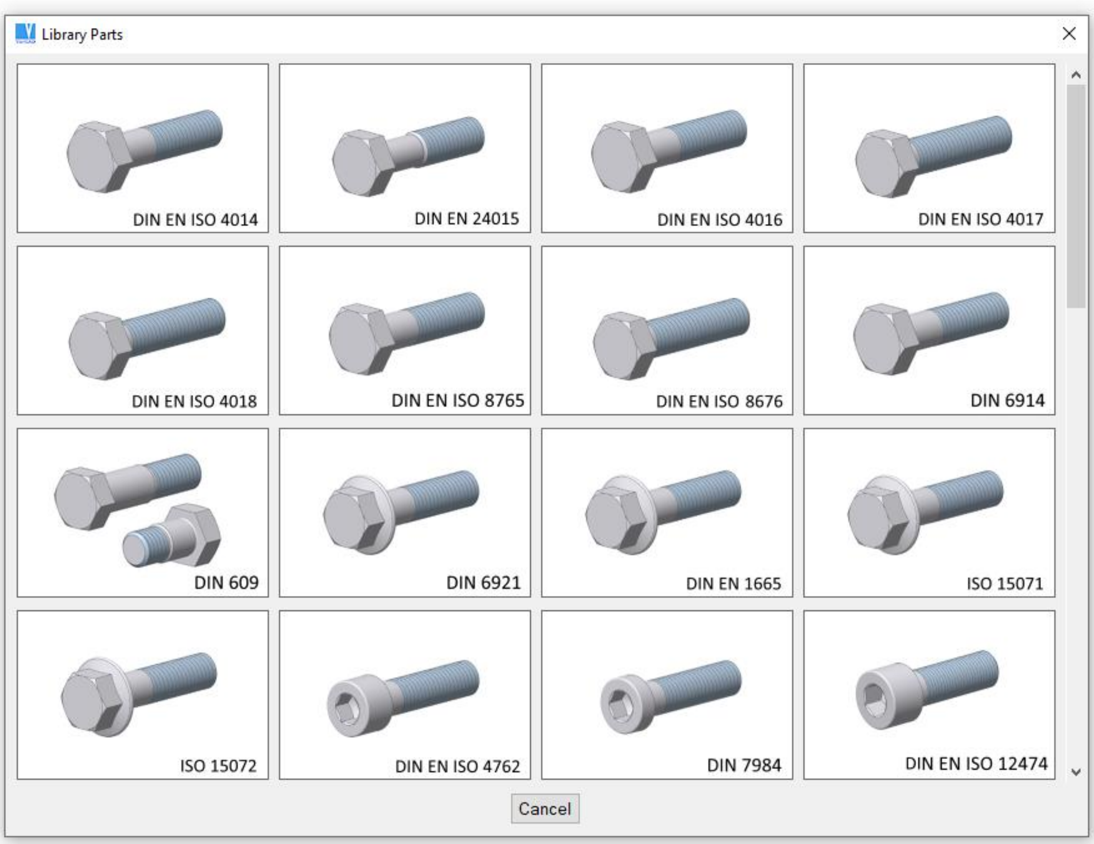
- ANSI
- DIN
- JIS
- CSN EN ISO

All parts except flanges and beams have defined their names and attributes. The names indicate the part definition and basic dimensions according to the chosen standard. These data are used in BOM creation. Flanges and beams are intended for further modification, in contrast to screws, nuts, bearings and others.

Selecting Mechanical Parts

Select the standard and the type of part (screw, washer, etc.) from the Parts menu. The icon menu appears from which you can select a specific part, such as a hex bolt. To define the dimensions, you can select:

- One basic dimension from the list. The other dimensions will depend on this basic dimension. For example, for defining a bearing, select the shaft diameter.
- One basic dimension from the list, then other dimensions from the list. For example, for defining a screw, select the bolt diameter. Then a list of lengths appears, depending on the selected diameter.
- One basic dimension from list, then enter the other dimensions manually. For example, for defining a rolled profile, select the profile dimensions and then enter the desired length of the beam.



Example of selection of DIN screws

Configuring Mechanical Parts Selection Dialog

In “CFG” command, you can select “Library Objects Selection Settings”, and then configure:

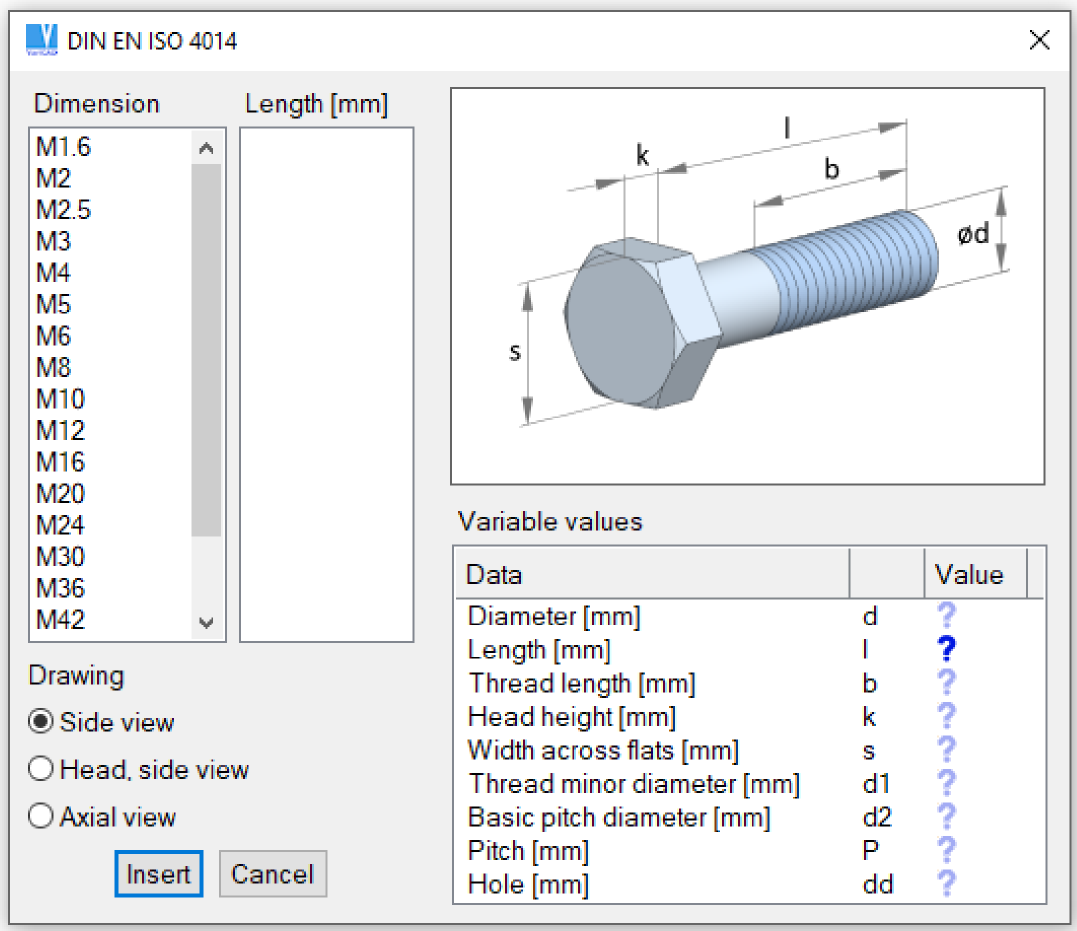
- Image size at panel selection (selects type of screw, type of bearing…)
- Image size at panel defining dimensions (selects size like M10, M12…)
- Whether images at panel defining dimensions are transparent

There are two image size available. For 4k, only the large size is used.

Inserting Mechanical Parts into 2D

When defining part dimensions in 2D, you can also select the method of drawing or which view of the part will be inserted. For example, when a rolled profile is defined, you can choose whether or not to hatch the section. When defining a screw, you can choose to insert the front view, left view (axial view), and head.

Mechanical parts are inserted as blocks, and have predefined connection points. For details on inserting blocks, see *Insert Block (page 108) (page 27)*.

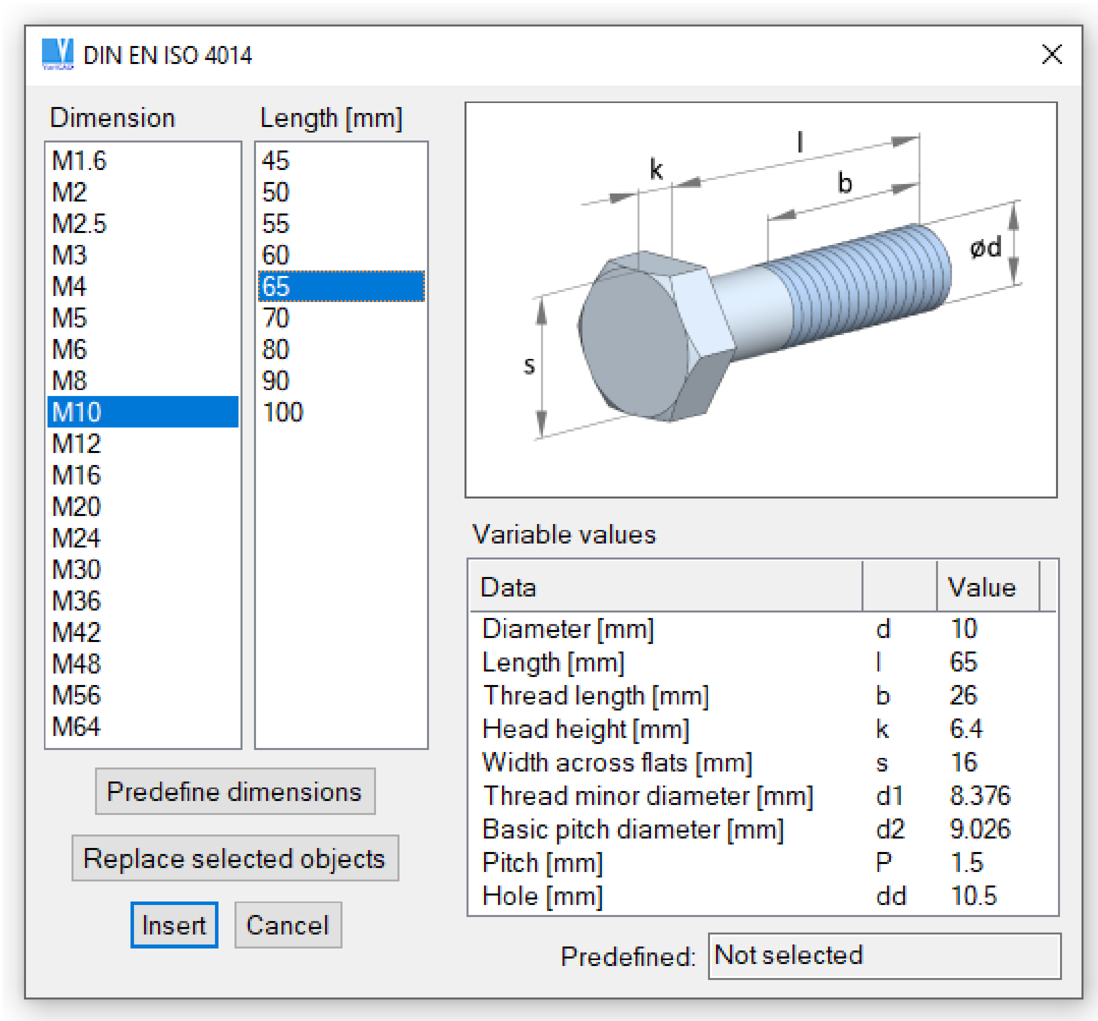


Mechanical part selection - 2D screw (bolt), dimensions are not yet selected.

Inserting Mechanical Parts into 3D

When defining part dimensions in 3D, you can:

- Predefine dimensions (see below).
- Select whether the part is inserted as a new part, or replaces a defined part. An example of replacement would be replacing one type of screw with another. If you define the parts to be replaced by name, you can replace all instances of a part in the entire file. See also *Selecting Solids (page 204) (page 154)*.



Mechanical part selection - 3D screw (bolt), dimensions are already defined

Pre-selecting Dimensions

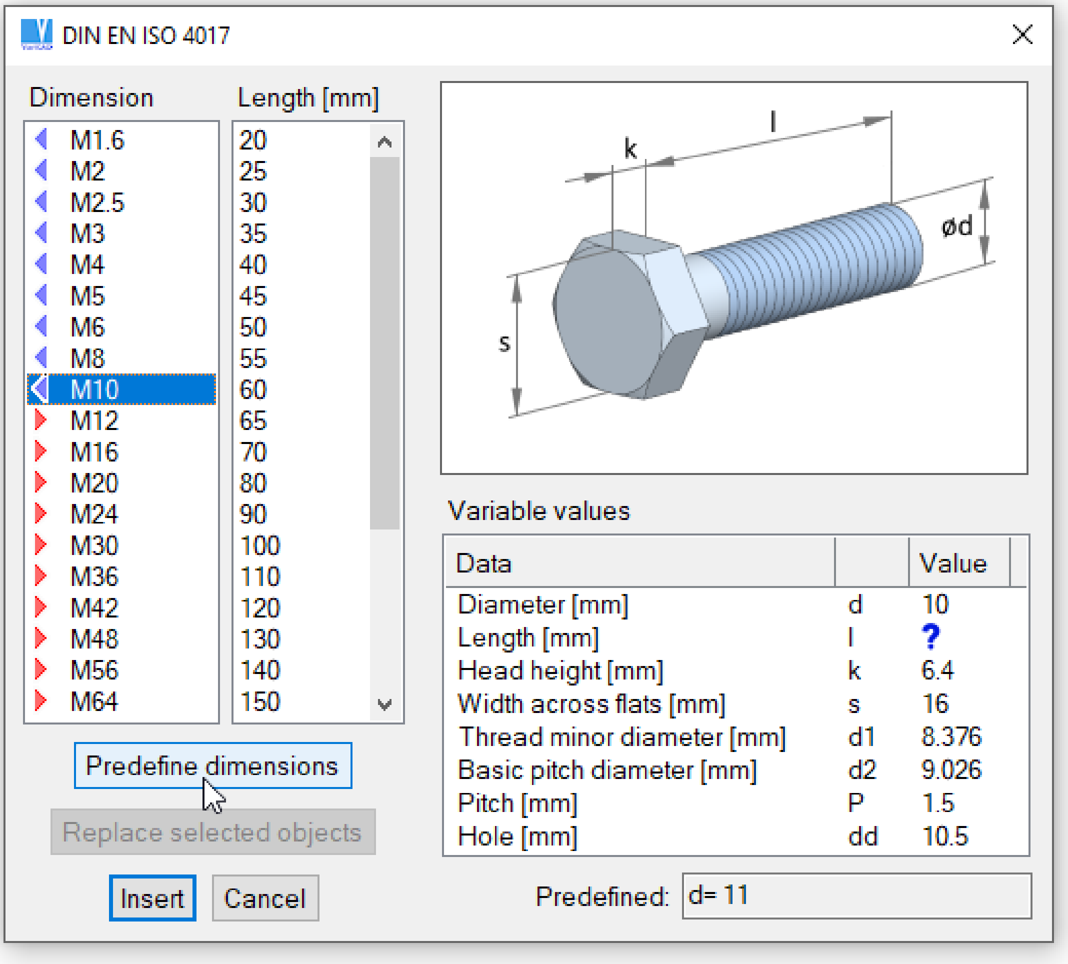
Almost all types of mechanical parts allow you to pre-select one or multiple dimensions. These dimensions can be either measured in 3D space, or copied from existing similar part. For instance – inserting a screw, you can select another screw and its dimensions are then pre-selected in current dialog window.

Pre-selected dimensions are marked by corresponding icons in list of dimensions. Icons distinguish situation when a dimension is smaller than, equal or greater than pre-selected value, or if it complies or not. See images in following examples.

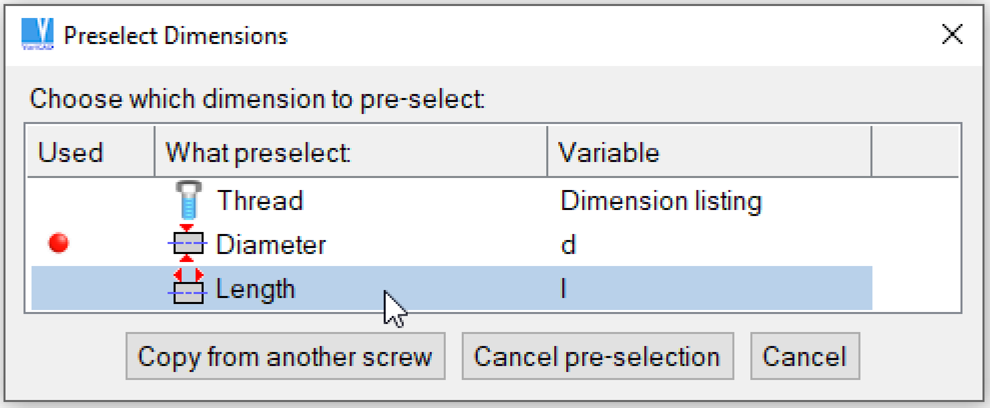
Individually measured values are:

- Thread specification
- A diameter (usually nominal diameter)
- Outer diameter – if it must be distinguished from a nominal diameter

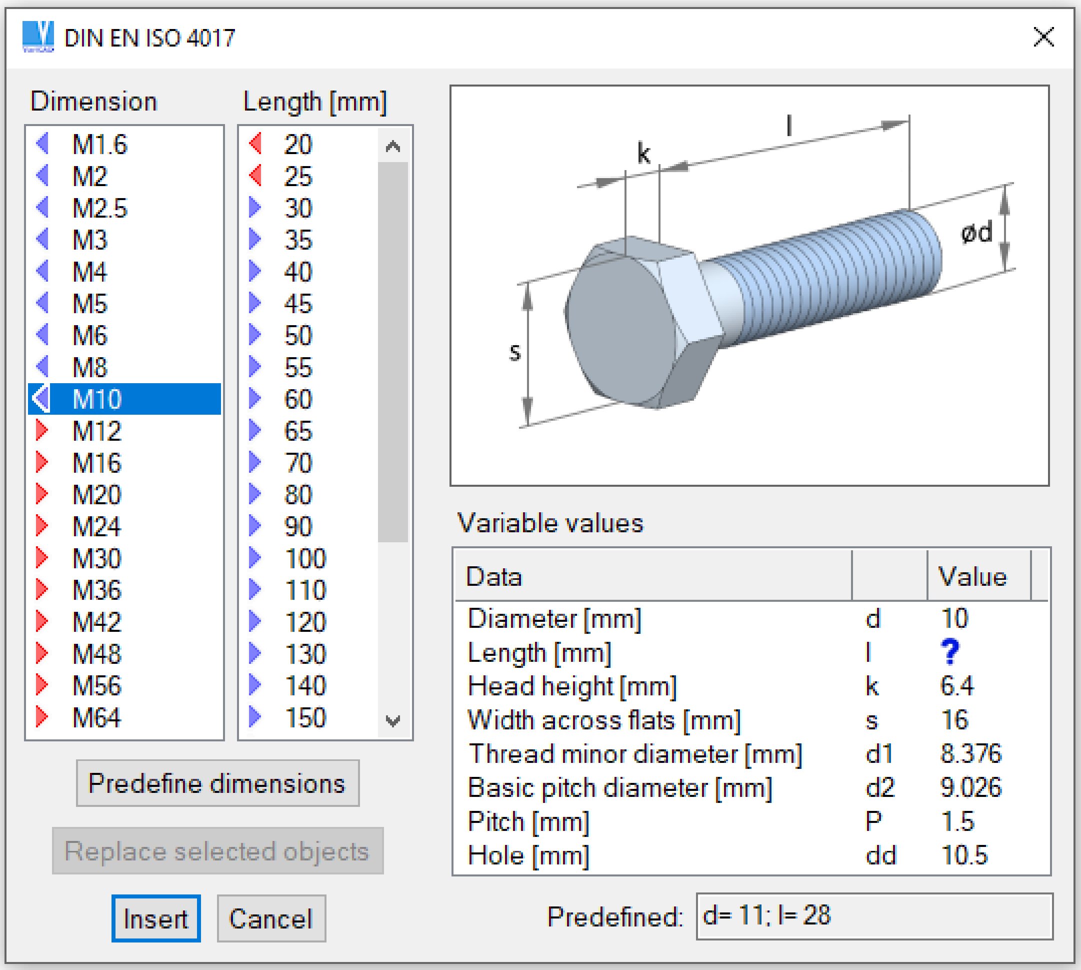
- Inner diameter, similarly as an outer diameter
- Length
- Width
- Height
- Thickness



Select "Predefine dimensions" for length measurement. Thread specification is already predefined.



Select measurement of screw length.



Predefined lengths are marked in list by corresponding icons.

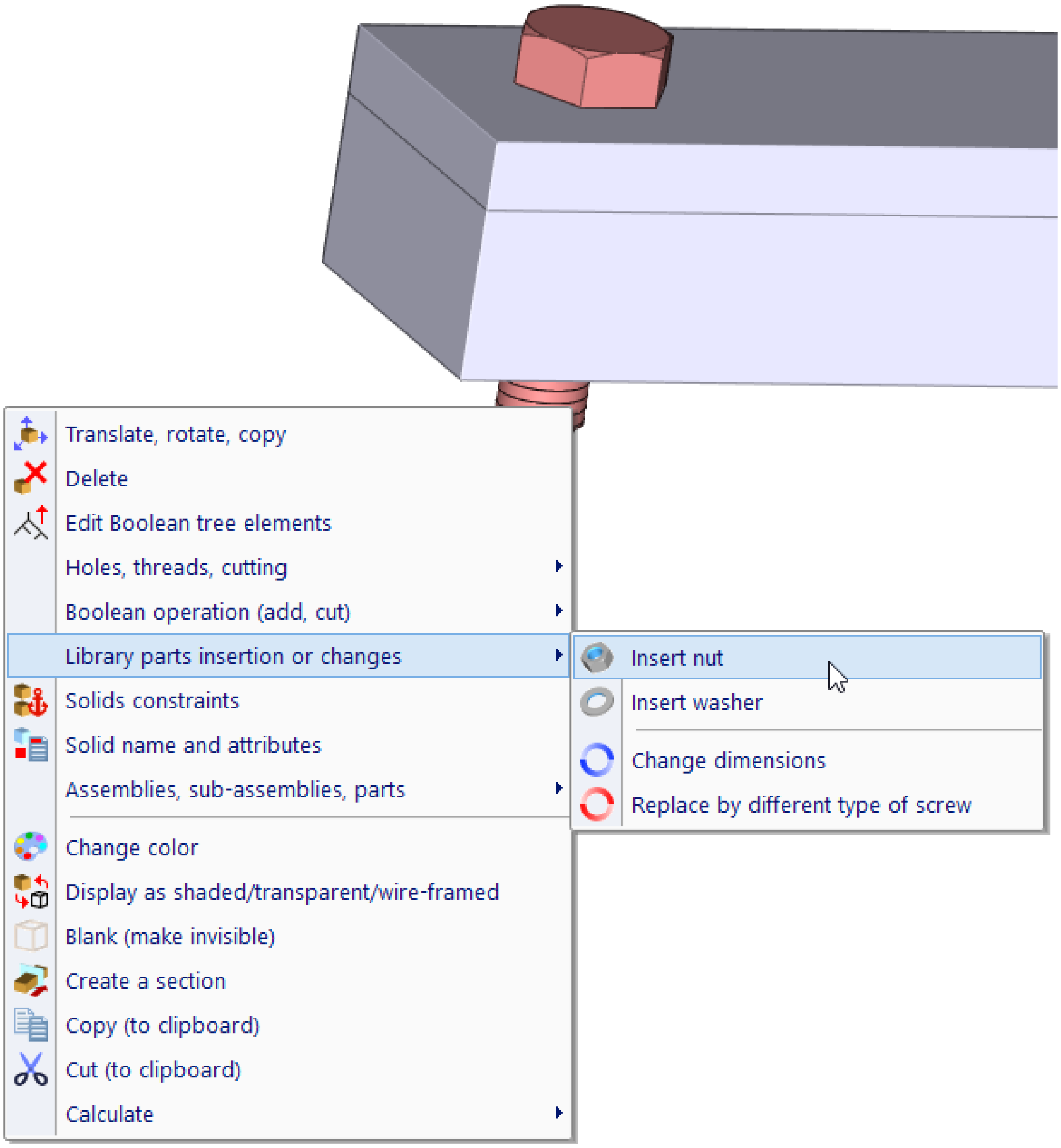
Options Available for Mechanical Parts in 3D

After right-click a mechanical part from library or selection of multiple parts, pop-up menu contains options related to library parts. These may be insertion of counter-parts, dimension change or type (standard) change.

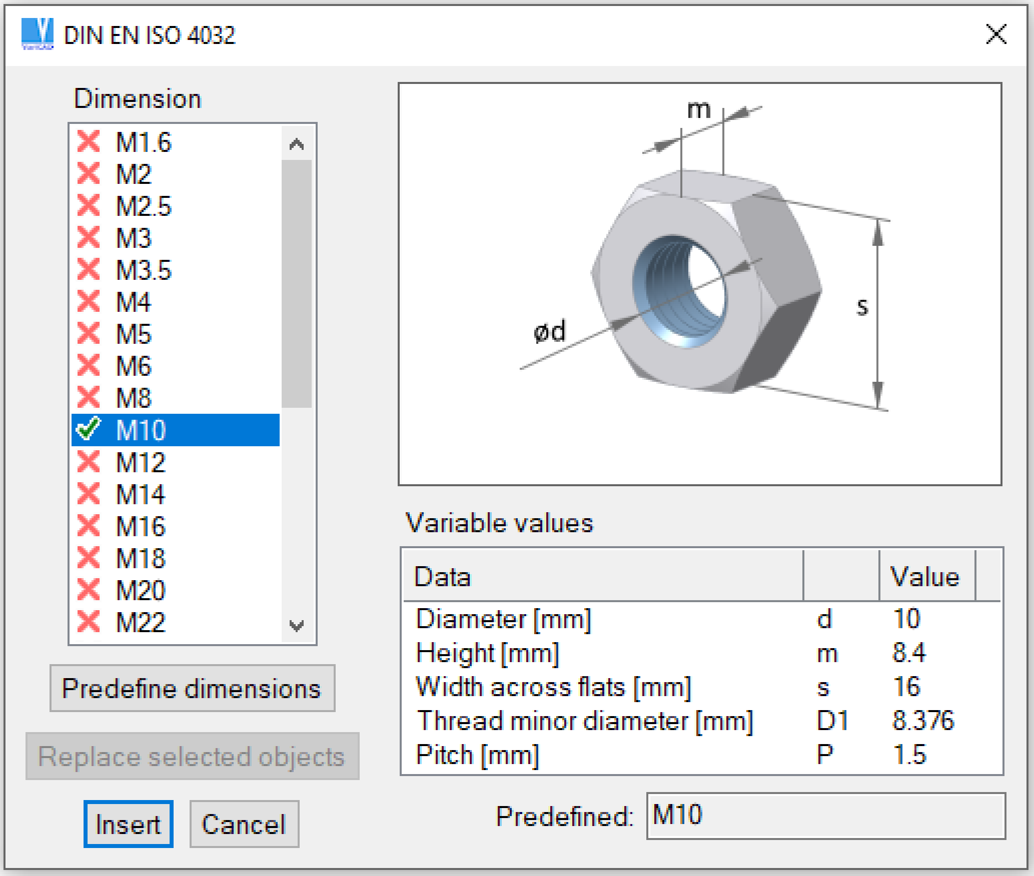
Inserting Counter-parts, Inserting Nuts onto Screws

According to situation, menu offers:

- Insertion of a nut, if you click a screw
- Insertion of a nut, if you click an outer threaded surface
- Insertion of a corresponding washer in both such situations
- Insertion of a screw, if you right-click a surface of a cylindrical hole
- Insertion of a screw, if you right-click a surface of a threaded hole
- Insertion of bearing or ring, if you right-click an outer cylindrical surface
- Insertion of bearing or ring, if you right-click a surface of a cylindrical hole



Right-click a screw, pop-up menu related to mechanical parts is selected



A panel with pre-selected thread of corresponding nut. The thread specification is defined according to detected screw.

Changing Dimensions or Type (Different Standard)

Right-click menu offers also change of dimensions of detected part. For instance, you can change length of an existing screw, diameter or thread etc. Dimensions in dialog window are pre-selected according to dimensions of detected part.

Apart of dimension changes, you can also change type (corresponding standard) of selected object. Again, dimensions in dialog window are pre-selected.

Changing Multiple Library Parts

Options allowing dimension or type change of library parts are available also, if you select multiple objects or if you select objects from assembly structure scheme. These options are active, if all selected objects are mechanical part libraries and if all of them have the same dimensions and are of the same type (the same standard).

Selecting Library Parts – Selection Filters

New features related to mechanical part libraries are completely available for parts inserted in version 2020-1.0 or higher. Obviously, they require additional data which are not present in older files. Some features are available for old library parts, similarly as for any other 3D objects. Right-click a threaded surface offers the same options for old parts, as for new ones.

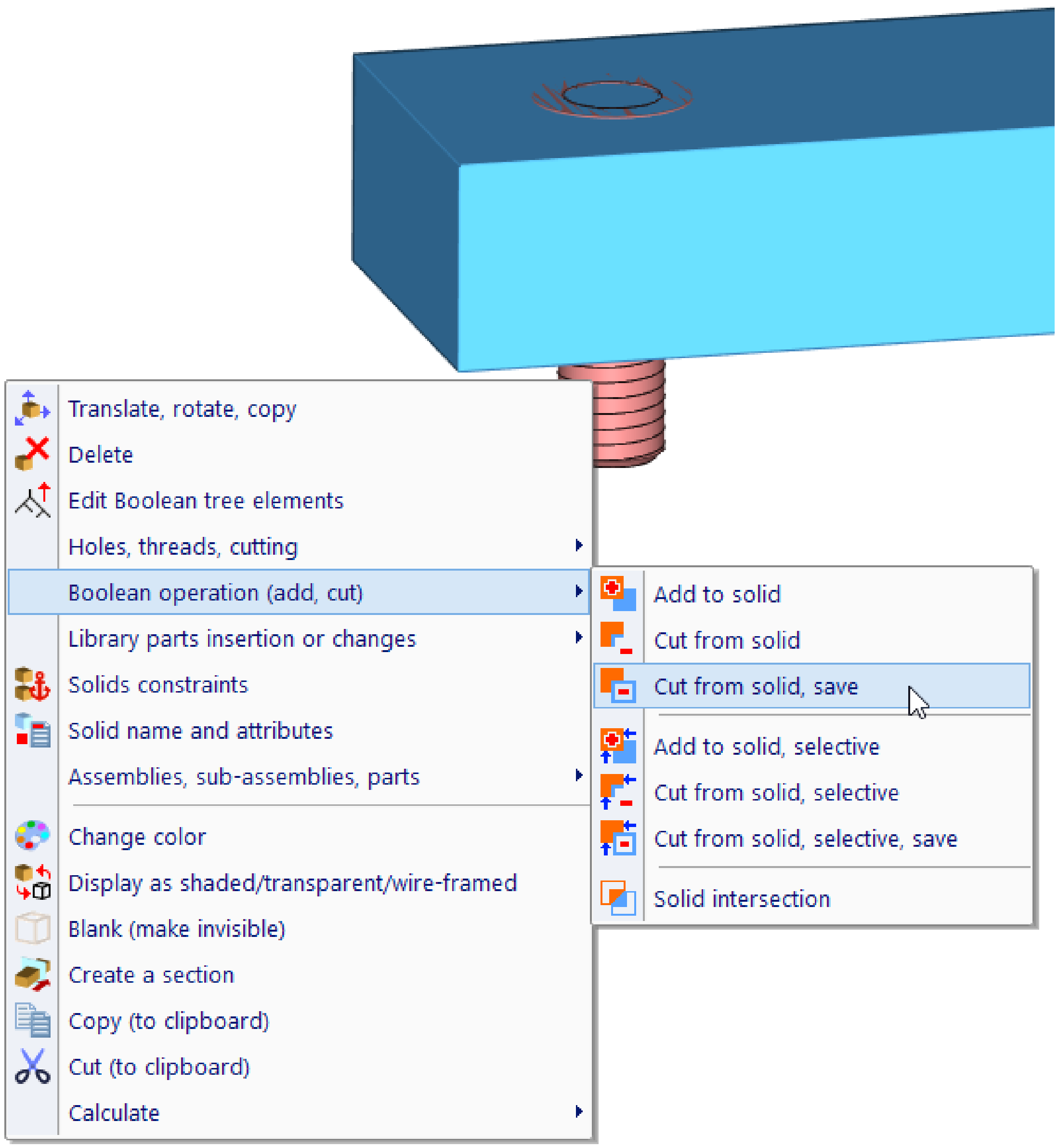
To make sure you select library parts supporting advanced features, you can switch selection mode in 3D space. Right-click an empty area and choose “Selection method” in pop-up menu. Then, select “Detect library parts, advanced features”. For given step, only part libraries containing related data are detected. To end such selection mode, press ESC.

Modifying Mechanical Parts in 3D

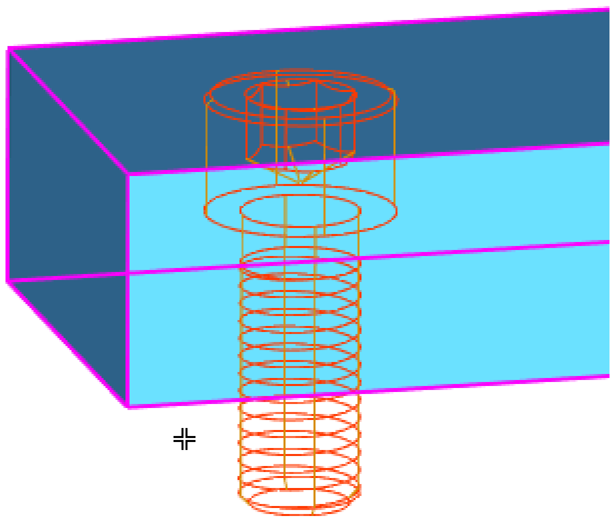
Mechanical parts are created according to respective standards and they should not be modified. If you attempt to edit shape of library mechanical part, warning message is displayed. Rolled profiles (beams) or flanges can be edited without any warning.

Modifying Counter-parts, Drilling Holes for Screws

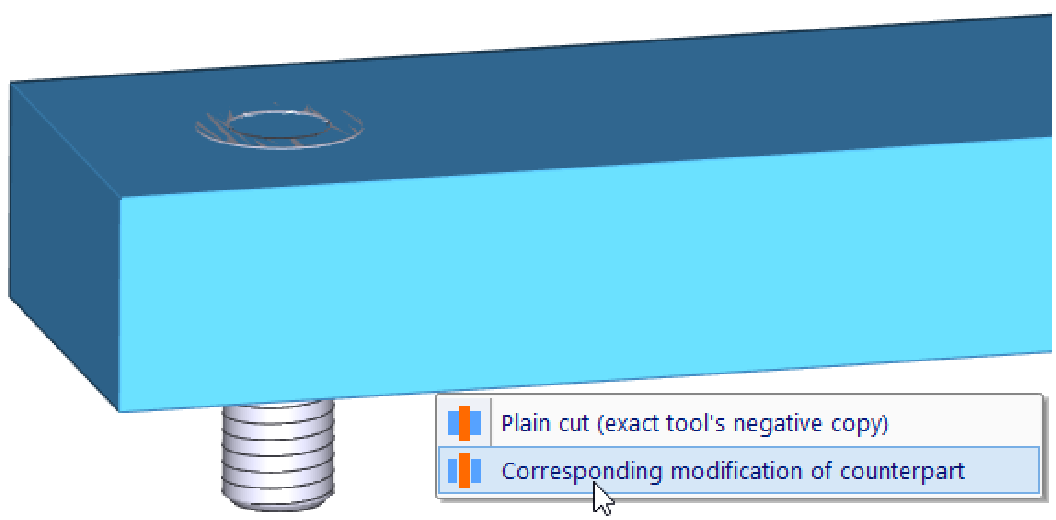
If possible, some mechanical parts have defined method of cutting or trimming other solids. For example, if you want to create a hole for screw, you can use screw as a “cutting tool” and create hole as exact negative copy of cylindrical part of screw. If you select corresponding modification of counterpart, a hole with diameter greater than screw diameter is created. Hole’s diameter is created according to standard, related to a diameter of the screw. For more information related to Boolean operations, see *Boolean Operations (page 208) (page 154)*.



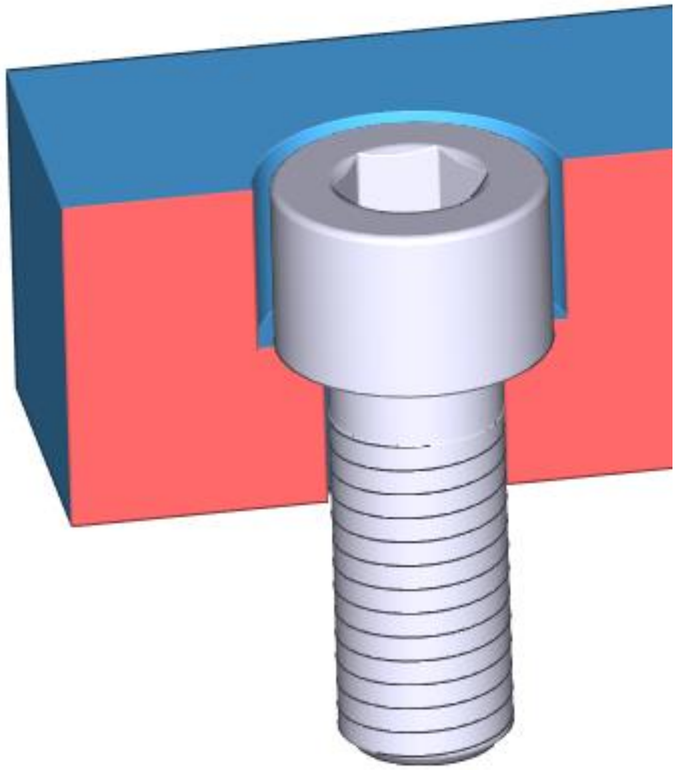
A screw inserted into full material is selected as a tool for Boolean cut.



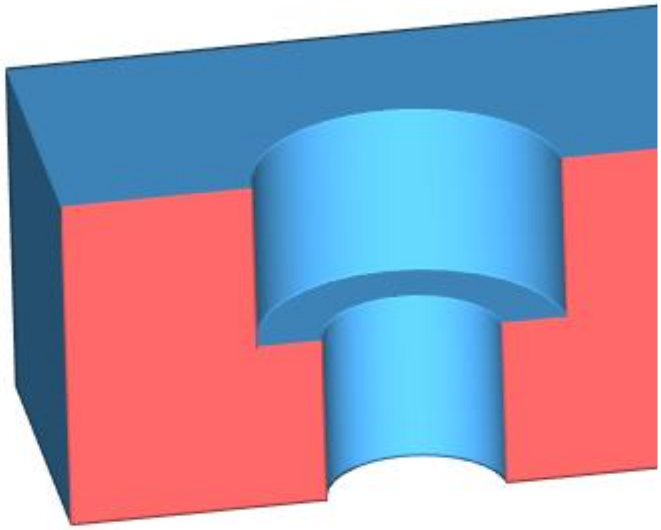
Selection of a solid the tool (screw) will be cut from (this drills a hole).



Menu with option Plain cut vs. Modification



A solid in section, after modification.

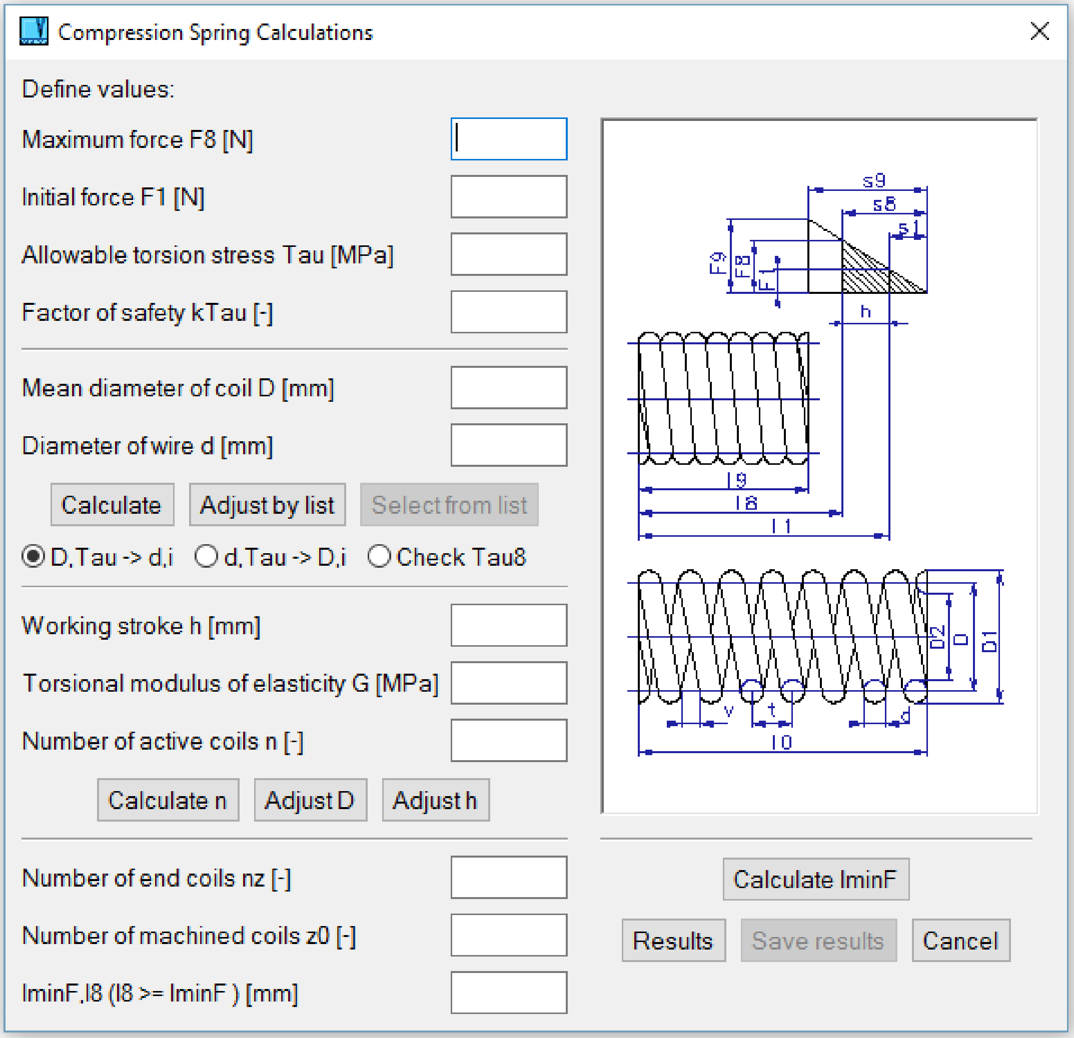


A solid in section after modification, without the screw

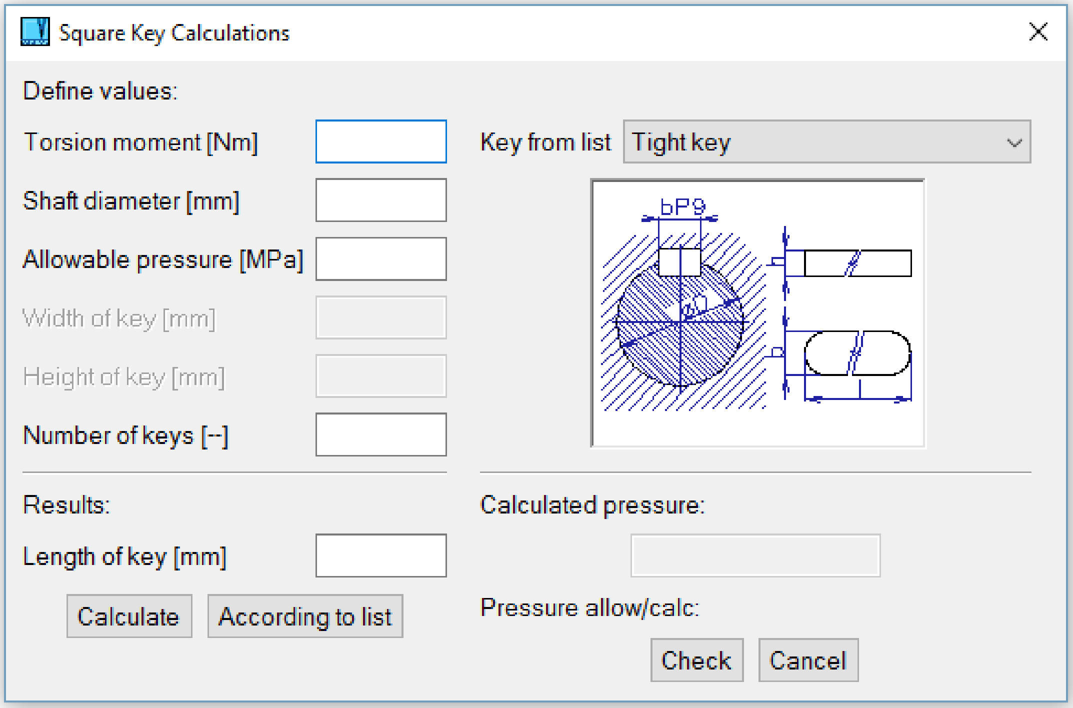
Chapter 10. Mechanical Part Calculations

Part calculations can be accessed from Objects / Calculations. Only ISO units (mm, N, MPa etc.) are used for calculations. Values can be calculated for the following mechanical parts:

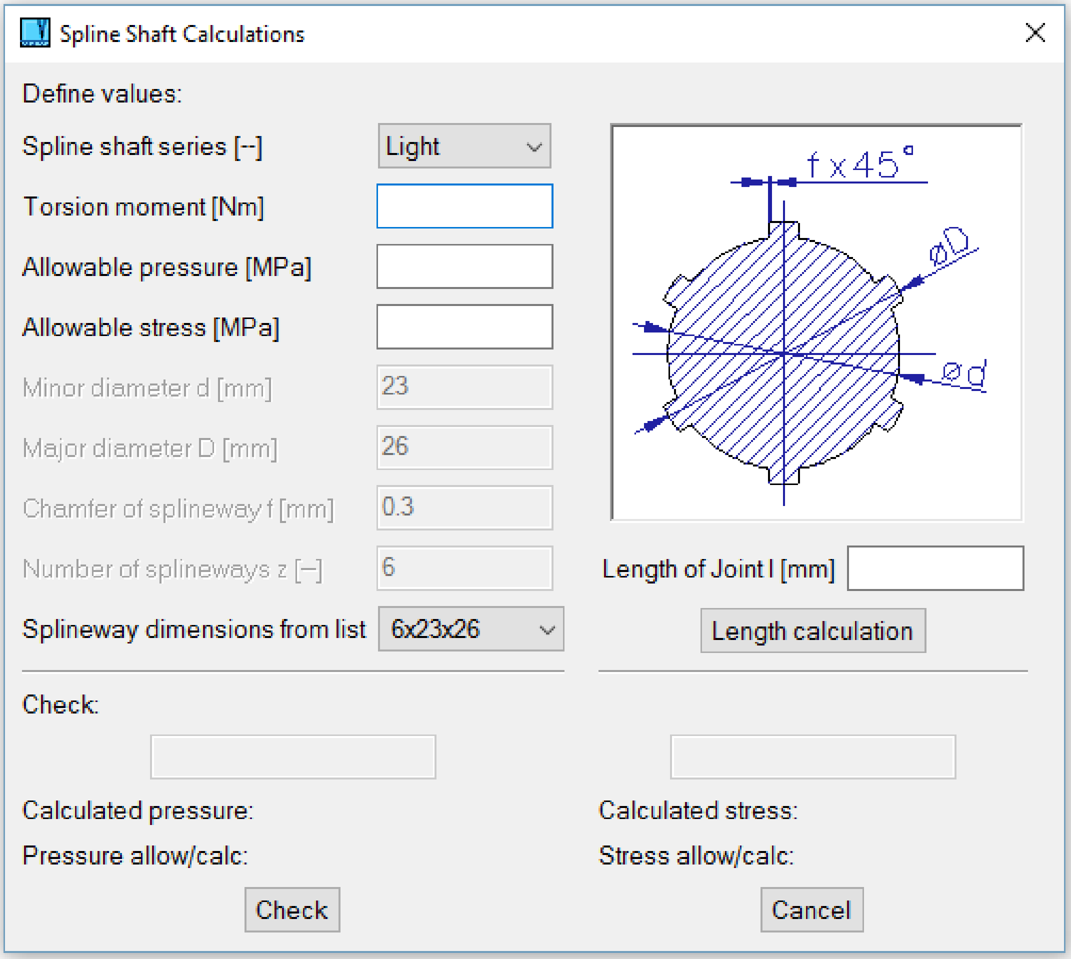
- Compression springs
- Tension springs
- Keys
- Spline shafts
- Screws
- Pins
- Spur gears
- Straight bevel gears
- V-belt drives
- Roller chain drives
- Beams or shafts (for bending and torsion)



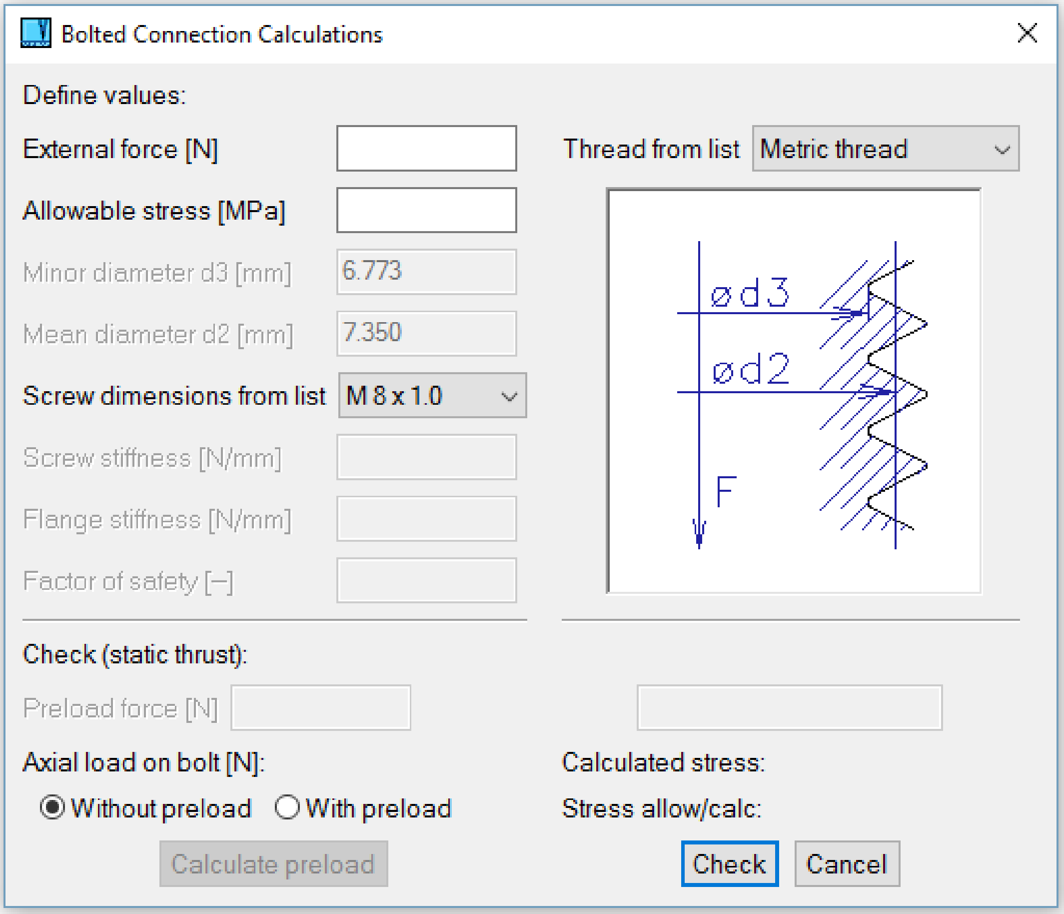
Compression spring


Square Key Calculations
Define values:
Torsion moment [Nm]
Shaft diameter [mm]
Allowable pressure [MPa]
Width of key [mm]
Height of key [mm]
Number of keys [--]
Key from list
Tight key
bP9
Results:
Length of key [mm]
Calculate
According to list
Calculated pressure:
Pressure allow/calc:
Check
Cancel

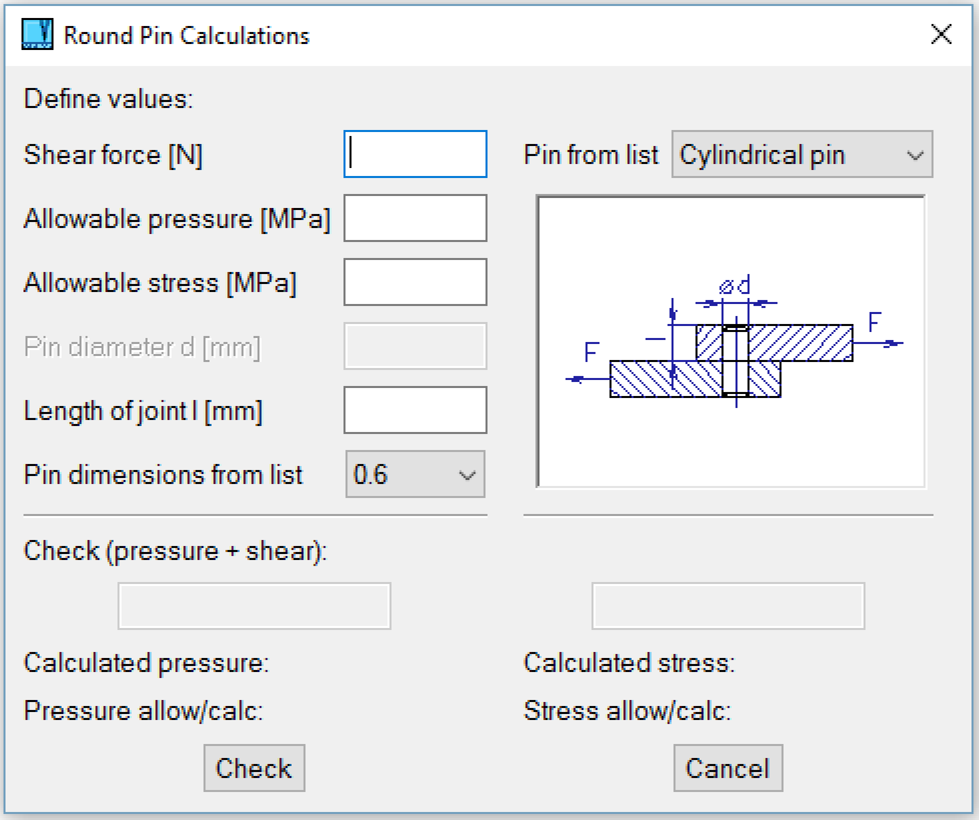
Square key



Spline shaft

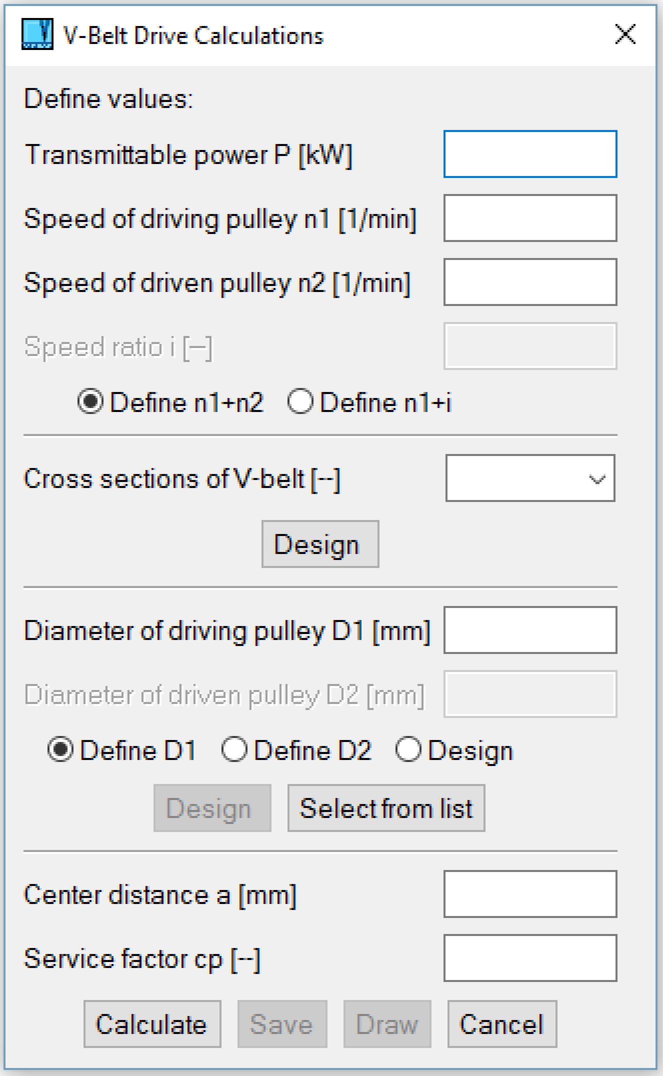


Bolted connection

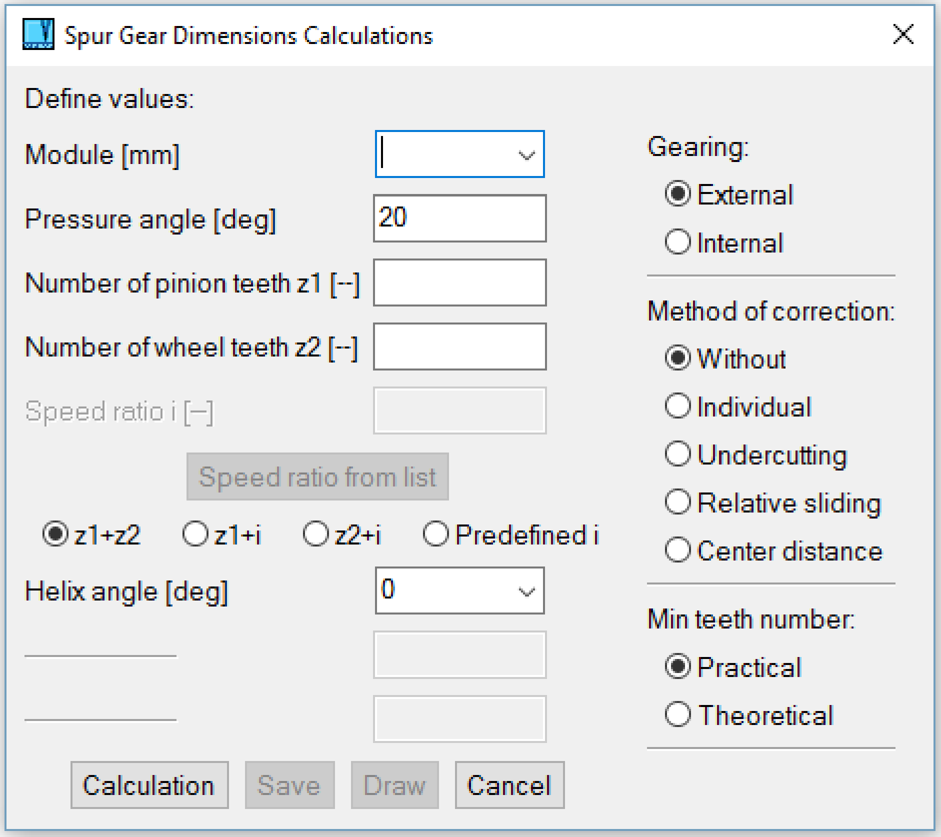


Round Pin Calculations
Define values:
Shear force [N]
Allowable pressure [MPa]
Allowable stress [MPa]
Pin diameter d [mm]
Length of joint l [mm]
Pin dimensions from list
0.6
Pin from list
Cylindrical pin
ød
F
F
Check (pressure + shear):
Calculated pressure:
Pressure allow/calc:
Calculated stress:
Stress allow/calc:
Check
Cancel

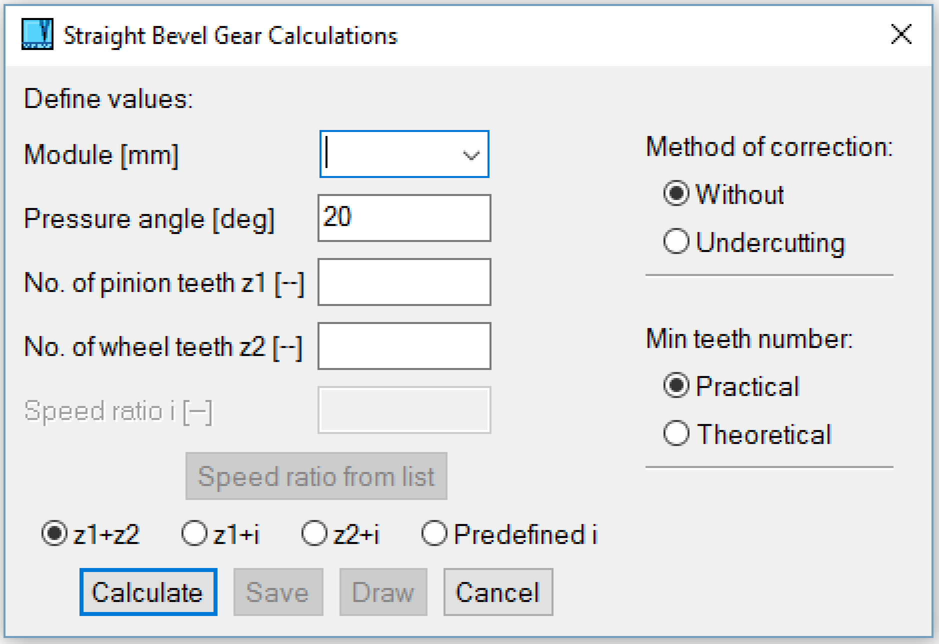
Round pin


V-Belt Drive Calculations
Define values:
Transmittable power P [kW]
Speed of driving pulley n1 [1/min]
Speed of driven pulley n2 [1/min]
Speed ratio i [--]
Define n1+n2
Define n1+i
Cross sections of V-belt [--]
Design
Diameter of driving pulley D1 [mm]
Diameter of driven pulley D2 [mm]
Define D1
Define D2
Design
Design
Select from list
Center distance a [mm]
Service factor cp [--]
Calculate
Save
Draw
Cancel

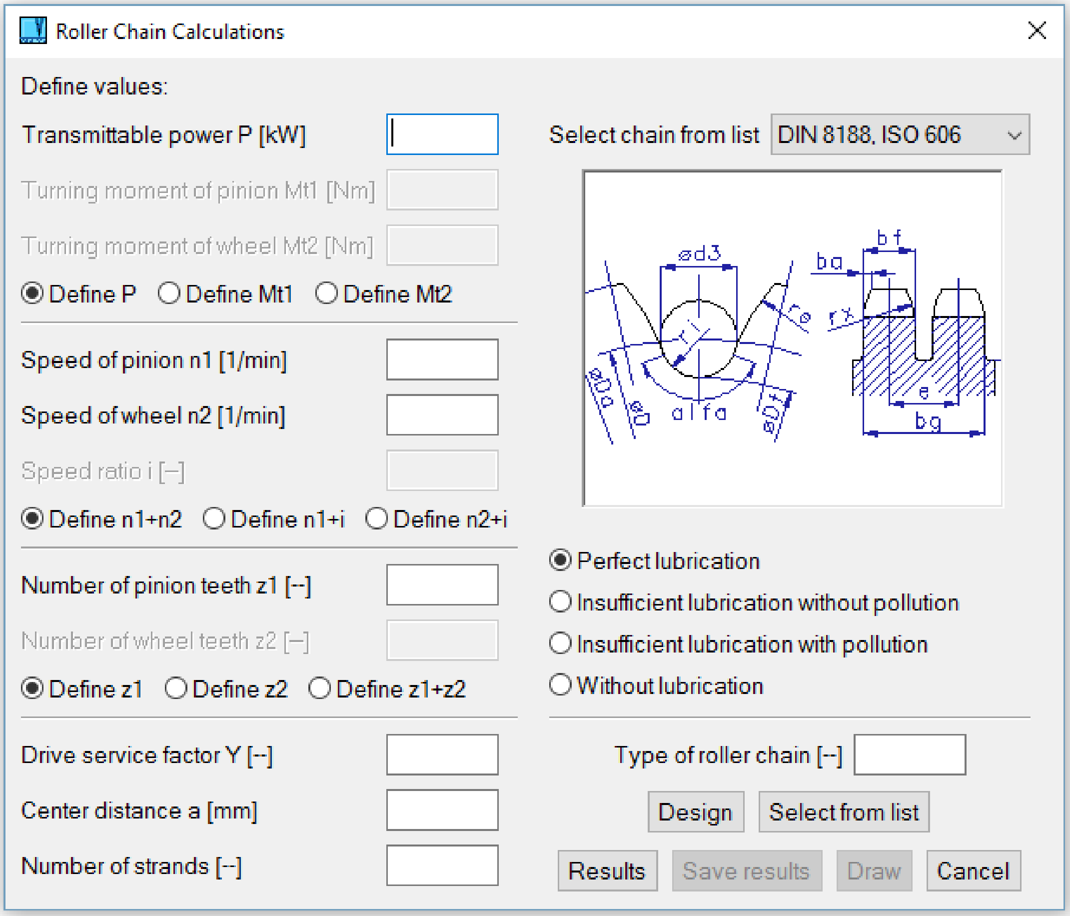
V-Belt drive



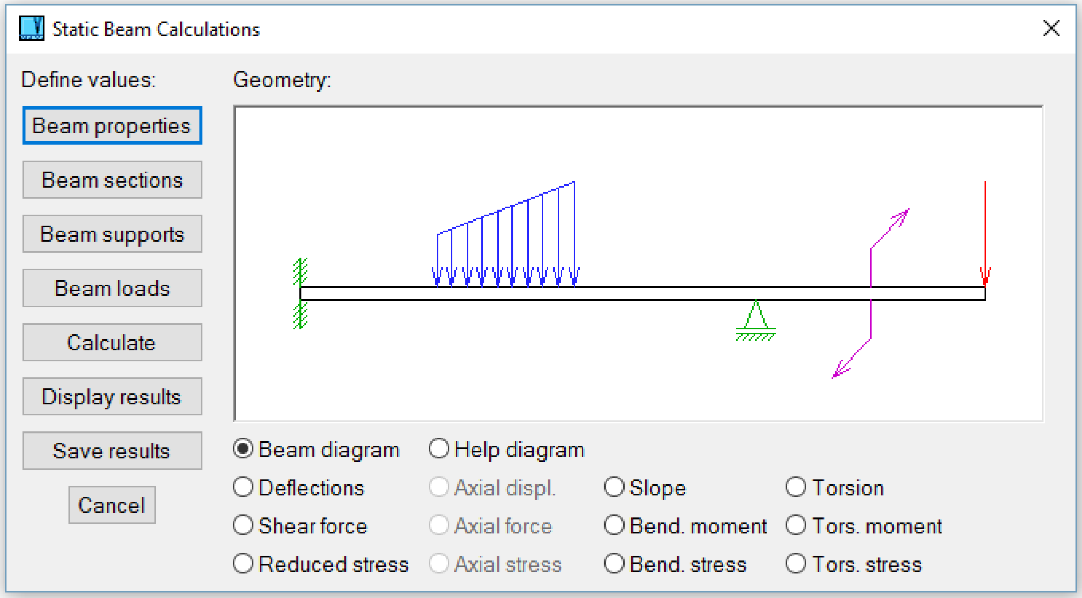
Spur gear dimensions



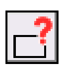
Straight bevel gear



Roller chain



Beam calculations



2D Area - 2DA

Calculates 2D surface area, bending section modulus and center of gravity of a selected 2D area. These values can be used in mechanical part calculations. A closed boundary must be defined, and islands can be included. You can define a boundary automatically or segment by segment. When the results are displayed, you have the option to create construction lines at the center of gravity.

Chapter 11. Printing and Plotting

For Windows users, VariCAD enables you to print to devices that use Windows drivers. Linux users can use Qt print capabilities. All users can always use VariCAD print drivers as well.

Printing Methods

Defining the Printed Area

There are multiple ways to define the printed area:

- Current drawing format - the size of the printed area is the same as the current 2D drawing format (A1, A2, etc.). Only objects within the format boundary are printed.
- Defined rectangular printed area. The printed area is selected by cursor, as a rectangular window.
- Only displayed objects - prints only objects that are visible on screen. You can change the zoom or the view window for the desired print area.
- All visible objects in 2D area.

Changing the Print Orientation

By default, the area is rotated so the longer side of printed area will be parallel to the larger side of the printed sheet. There are following additional rotation options:

- Portrait - the shorter side of printed area is parallel to the longer side of the sheet
- Landscape - same as automatic rotation, can be used if the printer driver gives unexpected results
- Do not rotate - can be used if the printer driver gives unexpected results

Defining the Printed Sheet Size

By default, the sheet size is the same as drawing format, or the maximum size allowed by the printer. There are following additional sheet size options:

- Maximum possible format - the printed sheet size will fit the maximum size allowed by the printer.
- Format according to drawing - resets the default.
- Select format from list - select a standard drawing formats such as A1, A2, etc. The selected size is the sheet size of printed sheet.

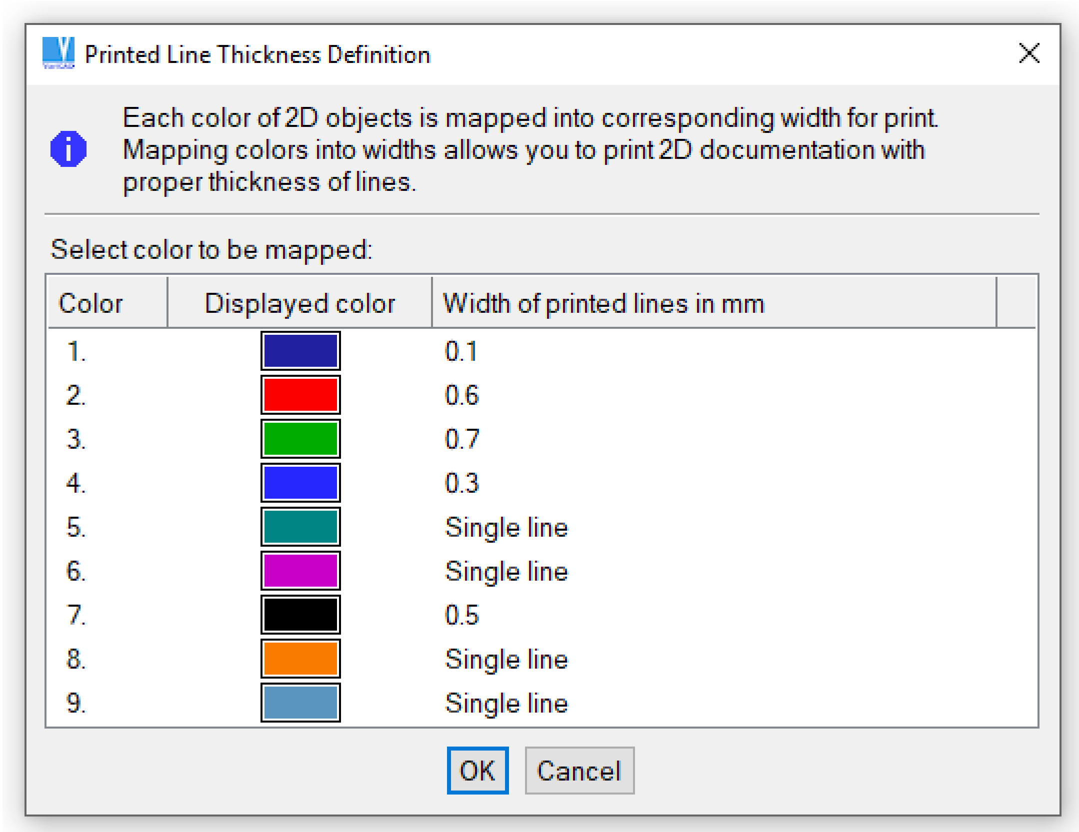
Changing the Print Scale

By default, scaling is disabled, but you can select scaling options:

- No additional scaling
- Fit to sheet size - the printed area is scaled to fit the size of the printed sheet. If the ratios of the sides of the printed area and the sheet dimensions are not consistent, scaling will still be isotropic (circles remain circles).
- Desired area scaled - scaling is defined by a value.

Color and Thickness Mapping

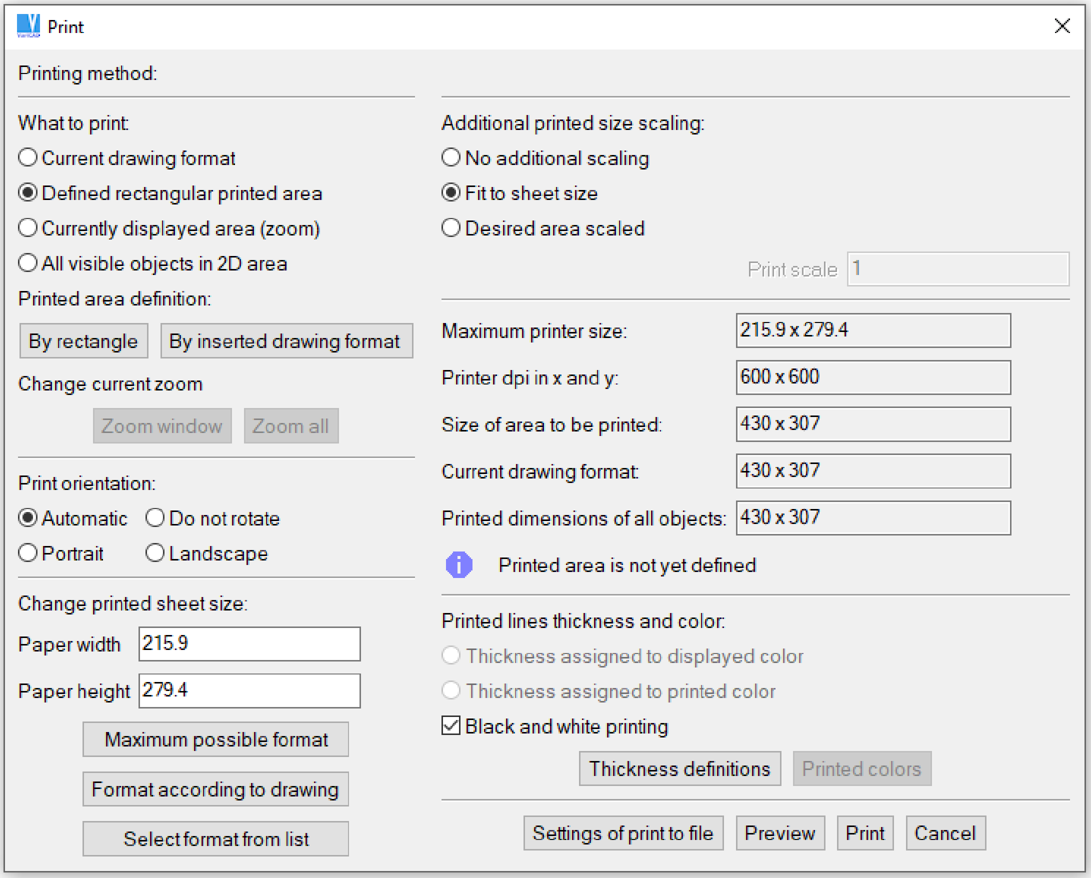
When using a color printer, you can map the display colors to printer colors. For standard printers, you can always map display colors to print line thicknesses. The result is different if the colors are mapped first. Line thickness and color mapping can be done within the Print Settings function, or in command “CFG”, in 2D section.



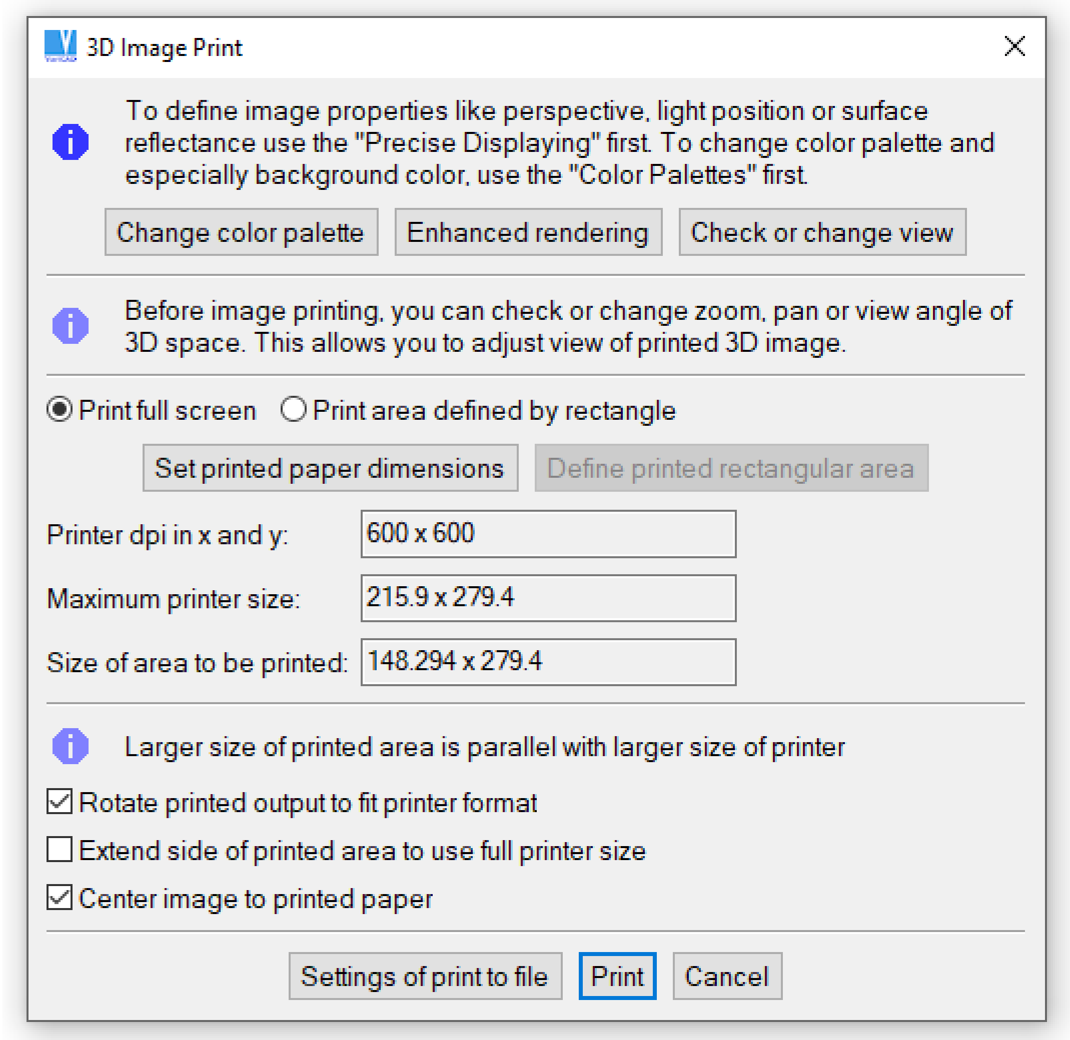
Definition of thickness of printed lines

Printing from 2D or 3D environment

If you are in 2D mode and if you select a printing command, the 2D area is printed according to print output settings. If you are in 3D mode, shaded or wire-framed objects are printed. For 3D printing, consider selection of enhanced rendering and especially, pure white background – similarly as for output of bitmap images – see *Exporting Images as Bitmaps (page 149)*.



Print output selection, 2D environment



Print output selection, 3D environment

Selecting a Printer

 **Print to System Printer - WPS**

 **Print - WPR, Ctrl + P**

Select a printer from the list of printers available in your operating system. The driver of each printer must be installed and set properly.

Print, VariCAD Drivers - PRN

This method is obsolete and no longer developed. It should be used only as a substitution in case the proper print system drivers are not available, or in case of an old hardware.

It creates output to any device using PostScript, HPGL/2, HPGL, PCL5 or Epson formats. You can select a printer listed in the window, or a printer compatible with any listed device. You can save your output to a file, or you can change or define the printing command of your operating system.

During printing, the operating system sends a temporary file to the printing device. In the command, the filename must be replaced by the sequence %s. For example, “copy %s LPT1 /b” is the Windows command for sending data to the parallel port. Default commands are predefined for this method of printing.

Batch Print

Batch printing is used for printing of multiple files at once. First, select the files to print. Then define the method of printing. Print settings can be defined for each file individually or for all files, and settings are the same as for single prints. See *Print Settings (page 145)*.

Once defined, batch print settings can be saved to a configuration file, which lists all printed files and their settings. If you need to print multiple files again, you can use this batch configuration file.

Batch printing is handled by the following functions:

Batch Print, from Predefined List - BPRP

Batch Print, or Define List to Print - BPRW

Batch Print, VariCAD Drivers - BPRV

Exporting Images as Bitmaps

Bitmap File from 3D - BMP

You can create a bitmap file as a content of 3D display. The image is saved into PNG format, or optionally into BMP format in true color mode (24 bits per pixel), into JPEG format or into GIF format (into GIF only under Windows). We recommend to use the PNG format, because this is compressed format without any loss of original quality. This aspect is important especially for any images scanned from computer screens.

Before the file is saved, the following properties can be set:

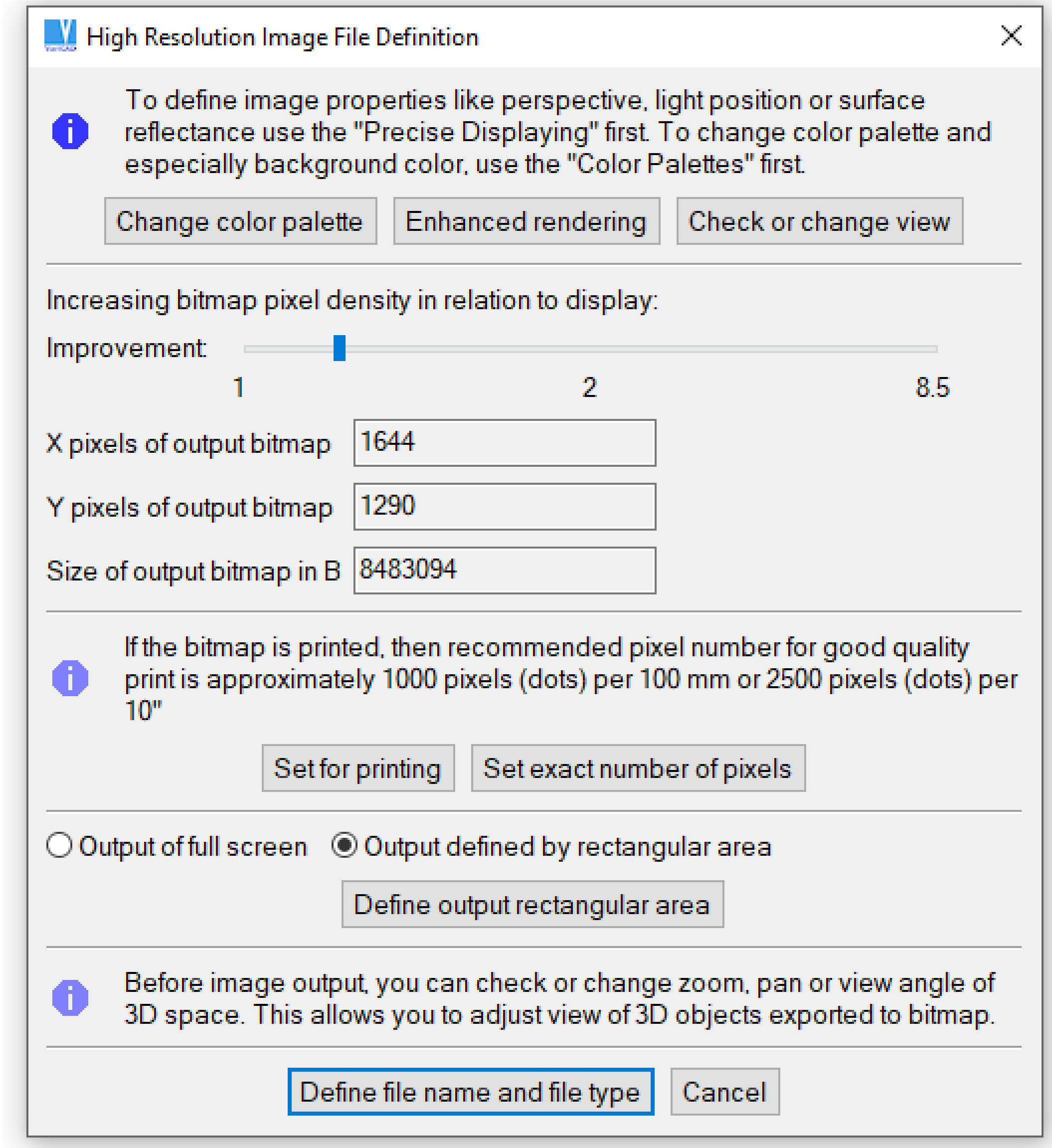
- Output of entire display or output of defined rectangular area.
- Pixel density. The bitmap pixel density can be easily defined in relation to display. If the value equals 1, bitmap quality is the same as obtained from display scanning or “print screen”.

- Number of pixels for printing. The pixel density is defined approximately according to the printed format. You can select paper size and number of pixels per 100 millimeters or per 10 inches, respectively.
- Number of pixels defined exactly. Changing these values, you modify also the output height-to-width ratio. This setting is available only for output of a defined rectangular area.

Before the bitmap image is created, you can check objects displayed inside the output area (full screen or defined rectangle). The display can be finally changed by standard functions like zoom, pan or view angle. If you need to change displaying from standard to enhanced (allowing perspective, light position changes etc.) or if you need to change the color palette, perform these changes before the bitmap output function is called.

If the bitmap file is prepared for printing, then consider color palette changes – especially the background color. Installation package of VariCAD contains a set of color palettes, containing a palette named "3D printing". We recommend to select this palette. The white background should have pure white color (RGB coordinates equal to 255, 255, 255).

The white background is usually the best option for printing of 3D images. Number of pixels (or dots number) for printing should be set according to the paper size, printer settings (dpi) or color vs. grey scale printing. 1000 dots per 100 mm or 2500 dots per 10 inches give you reasonable sufficient quality of the printed document.



Bitmap creation

Chapter 12. VariCAD on the Internet

You can access some of web sites or web pages directly from VariCAD, using your default browser.

Home Page - INH

YouTube VariCAD Channel - YOUTUBE

Facebook VariCAD Page - FACEBOOK

What's New Page - INN

Opens the page describing new versions

Upgrade - INI

Opens the page containing system upgrades and download links. See also *Installing Upgrades (page 3)*.

Feedback - INF

Opens a page you can fill out with your feedback

FAQ - FAQ

Opens the page containing frequently asked questions

Web Browser Settings - INST

Define which web browser will be used to access VariCAD web pages. We recommend using your default browser.

Trial Versions, Online Purchasing

You can download a 30-day trial version from the VariCAD website. After registration, you will be able to download a trial version for free. When you purchase a VariCAD license or upgrade, you receive a license and key-code that will convert your trial version to a fully functioning version.

Also, a trial-version specifically modified is available for users working with full version. This trial version is intended for evaluation of a new VariCAD release. Everyone has a chance to test VariCAD before purchasing it – either a new user never working with it, or someone who has a full license and wants to test a new version.

The following tools are provided for working with downloaded versions:

 **Online Purchase - PCHS**

Purchase licenses or upgrades

 **Registration - TREG**

 **License Code - ELCD**

Enter your license and key-code

Chapter 13. 3D Modeling

3D Display

The 3D display provides the following features:

- Shaded and wireframe views
- Standard views (top, left, etc.)
- Dynamic rotation, pan and zoom
- Defining the center of view rotation
- Rotate using arrow keys
- Save and restore views
- View undo and redo

Dynamic View Manipulation

You can use the following standard methods to change your view dynamically:

- To rotate: press the right mouse button, and move the mouse.
- To pan: press the middle mouse button (mouse wheel) and move the cursor
- To zoom: use the mouse wheel.
- To rotate: press the right mouse button, and then also Ctrl and move the mouse to rotate. The speed of view rotation is dependent on dimensions of a displayed area. This works well especially if large zoom is used.

Other methods are also available – they are usually provided to keep interface compatibility with older versions of VariCAD:

- To rotate: press Ctrl + Shift key, and the left mouse button, and move the mouse to rotate. The speed of view rotation is dependent on dimensions of all visible objects. This works well in most cases except the large zooms.
- To rotate: press Ctrl + Shift key, and the right mouse button, and move the mouse to rotate. The speed of view rotation is dependent on dimensions of a displayed area. This works well especially if large zoom is used.
- To rotate: press the right mouse button, then the left button, and move the mouse. This is the same as Ctrl, Shift and left mouse button.
- To rotate: press the right mouse button, then the left button, Ctrl key and move the mouse. This is the same as Ctrl, Shift and right mouse button.
- To zoom: press Shift and the left mouse button, and move the mouse. Move the mouse up to shrink, down to enlarge.
- To zoom: press the right mouse button, then the middle button, and move the mouse.
- To pan: press Ctrl and the left mouse button, and move the mouse.

- To pan: press the middle mouse button, then the left button, and move the mouse.
- To refresh view, press F6

For dynamic view rotation, zoom or pan all keys and/or mouse buttons must be held simultaneously, of course.

Dynamic view changes can be set in command “CFG”.

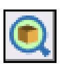
Animated View Changes

If view is not changed dynamically by cursor movement or mouse wheel rotation, the change is animated. For instance – if you select a front view and the current view is under a random angle, the view is not redrawn immediately. Angle of view is changed within a few tenths of second. This gives you better impression of the change.

VariCAD permanently checks time of redrawing. If you have slow graphics and if you want to redraw a large amount of 3D data, animation is automatically turned off. Animation of view changes can be managed in command “CFG”.

Predefined View

You can use a predefined angle of 3D view. Predefined view is created from front view by rotation around X, Y and Z axes of display. It can be set in command “CFG”. By default, the predefined view is used whenever a new file is created.

 **Predefined View - PRV**

This command sets the view according to predefined angles.

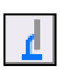
Rotating View Using the Arrow Keys

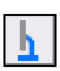
You can use the arrow keys to rotate the view around a specific axis. Press Shift, Ctrl, or both, and then press the following:

- Left or right arrow to rotate around the Y axis
- Up or down arrow to rotate around the X axis

You can use the keys in combination to change the view angle.

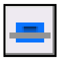
3D View Tools

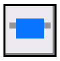
 **Left View - VLE**

 **Right View - VRI**

 **Front View - VFR**

 **Back View - VBA**

 **Top View - VTO**

 **Bottom View - VBO**

 **Isometric View 1**

 **Isometric View 2**

 **View to Plane - RNP**

Sets the view perpendicular to a selected plane. This tool is useful especially when creating a 2D view export.

Rotate View Around X 90 Degrees - X90

Rotate View Around X 180 Degrees - X180

Rotate View Around X 270 Degrees - X270

Rotates the view by the specified angle around the global X axis (horizontal axis)

Rotate View Around Y 90 Degrees - Y90

Rotate View Around Y 180 Degrees - Y180

Rotate View Around Y 270 Degrees - Y270

Rotates the view by the specified angle around the global Y axis (vertical axis)

Rotate View Around Z 90 Degrees - Z90

Rotate View Around Z 180 Degrees - Z180

Rotate View Around Z 270 Degrees - Z270

Rotates the view by the specified angle around the global Z axis (perpendicular to display plane)

Undo View - ZPR

Redo View - ZRD

Zoom Window - ZWI

Defines the zoom by specifying opposite display corners

Zoom All - ZALL

Adjusts the view to display all visible objects

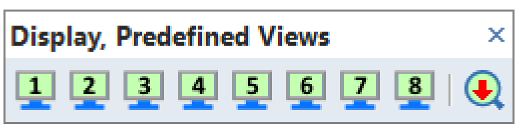
Saving Views

You can save up to eight views that contain rotation, zoom and pan settings. The Predefined Tools toolbar enables you to easily restore these saved views.

Save View - click this icon and select the number of the saved view.

The other icons on this toolbar restore the numbered views. Your views are saved with the file.

Both the 2D and 3D components of a file can each contain eight saved views.



Predefined Views toolbar

Shaded and Wireframe Display

Shade/Wireframe Entire Display - SHW

Switches the display from shaded to wireframe or vice versa.

3D Display Settings

3D Shading and Edges Settings - 3DS

Enables you to change how 3D objects are displayed. You can define:

- Displaying edges. You can define shininess of edges, edges darkness and whether the tangent connections of patches are displayed.
- Surface reflectance. Enables to set light attributes and defines method of surface shading. See *Surface Shading (page 160)*.

 **Colors and Wires of Solids - SCO**

This function defines:

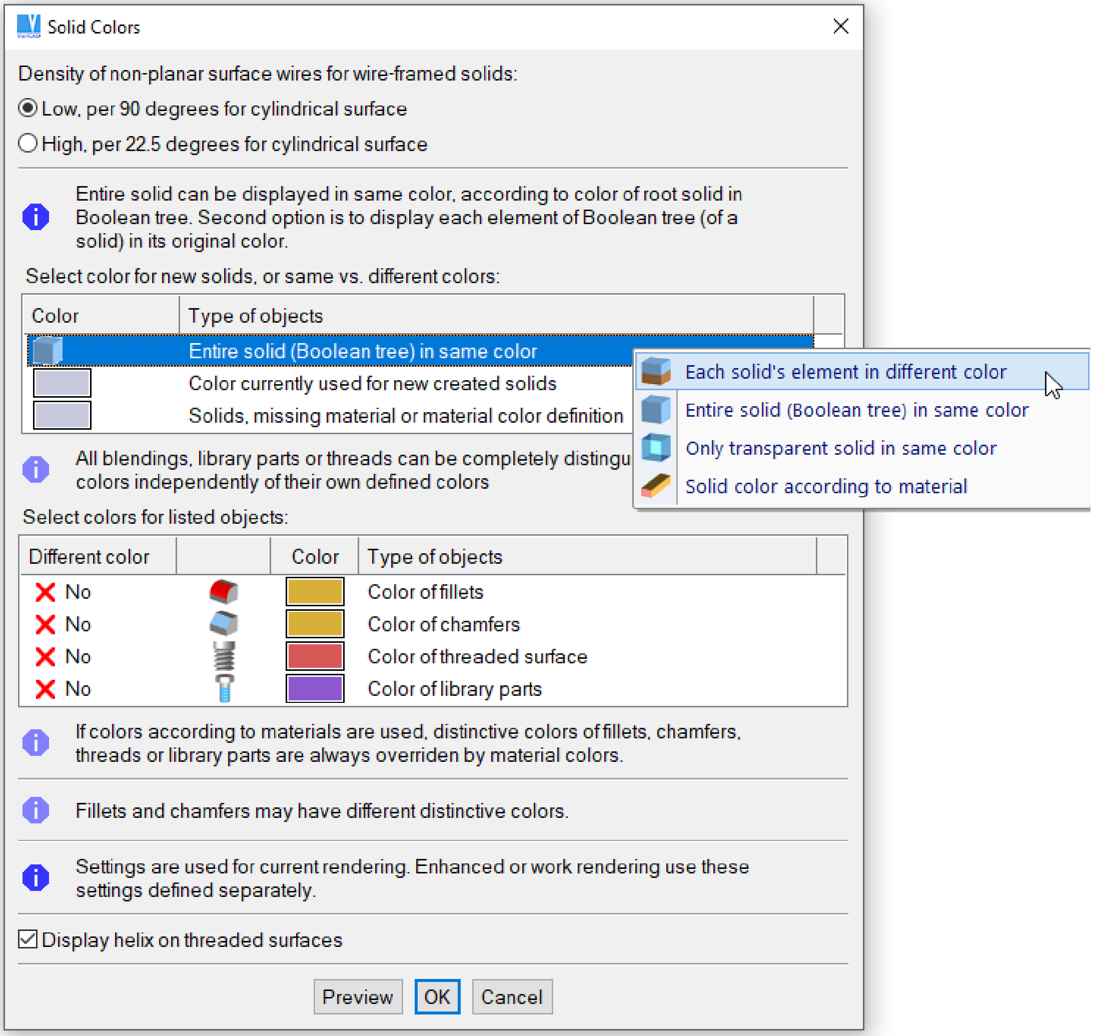
- Density of non-planar surface wires. Density setting affects solids displaying, if all space is displayed as wire-framed or if a particular solid is displayed as wire-framed.
- If the entire solid is displayed in the same color. Otherwise each component added to the solid during past Boolean operations is displayed in the original color.
- If the solid is displayed in color assigned to material (like a particular steel grade, for instance). Then, solids are distinguished by material colors. If a color for given material is not defined, or if a solid has no material defined, then a common color is used. See *List of Materials (page 313) (page 298)*.
- Colors of new solids. New 3D objects like a solid, hole, filleting etc. are created in the selected color.

You may also select different colors for entire groups of objects. Thus, the objects are distinguished easily from the rest. This option overrides objects' own colors, if turned on. Turned off, the objects are displayed again in their own colors.

You may select a distinct color for:

- Filleting
- Chamfering
- Threaded surfaces
- Mechanical parts inserted from libraries

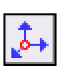
These options are turned off, if a color is defined by solid’s material.



Solid display settings window

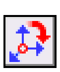
View Rotation Center

Whenever you rotate 3D view, the view rotates around a defined point. You have several methods to re-define the view rotation center. The best option is to right-click an empty area and to select the change from pop-up menu.



Auto View Rotation Center - VCN

Sets the view rotation center to the center of geometry of all visible solids.



Define View Rotation Center - VCNI

Sets the view rotation center to a specified location, selected by cursor.

View Rotation Center to Display Center

Sets the view rotation center to center of viewport. This option is especially recommended for enhanced rendering.

Enhanced Rendering

Enhanced Rendering - SRD

Renders 3D objects more precisely. Smoothly rendered images are realistic and can be used in product presentation materials such as brochures, printed PDF files, folders etc. Once selected, enhanced rendering persists until any 3D edit function is called. It means that you can use all functions working with display, like standard views, view rotation etc. and enhanced rendering is still present. Display settings are the same as for standard mode; however, set values are different for each display mode.

Enhanced rendering is recommended if you want to create a bitmap output of images, or for printing of 3D display.

For enhanced rendering, you can set the following attributes:

- Surface reflectance. Enables to set light attributes and defines a method of surface shading. See *Surface Shading (page 160)*.
- Display perspective. You can turn perspective on or off. If the perspective is on, you can smoothly change relative eye distance. This distance is defined as eye distance from nearest 3D space location divided by 3D space dimension.
- Light position. Defines the light position by the cursor movement.
- Reset light position. Sets the light position above the center of display.
- Round surface smoothness. Each non-planar surface is displayed as a certain amount of tiny planar facets. If the number of facets is increased, displaying is more precise and slower. Facets are obvious, for instance, on a cylinder in large zoom, if the sight line is parallel to the cylinder axis.
- Edges displaying. You can define shininess of edges, edges darkness and whether the tangent connections of patches are displayed. These settings are similar to standard display settings.
- Threads displaying. We recommend to turn off the different color of threads, if you use such option for standard rendering.

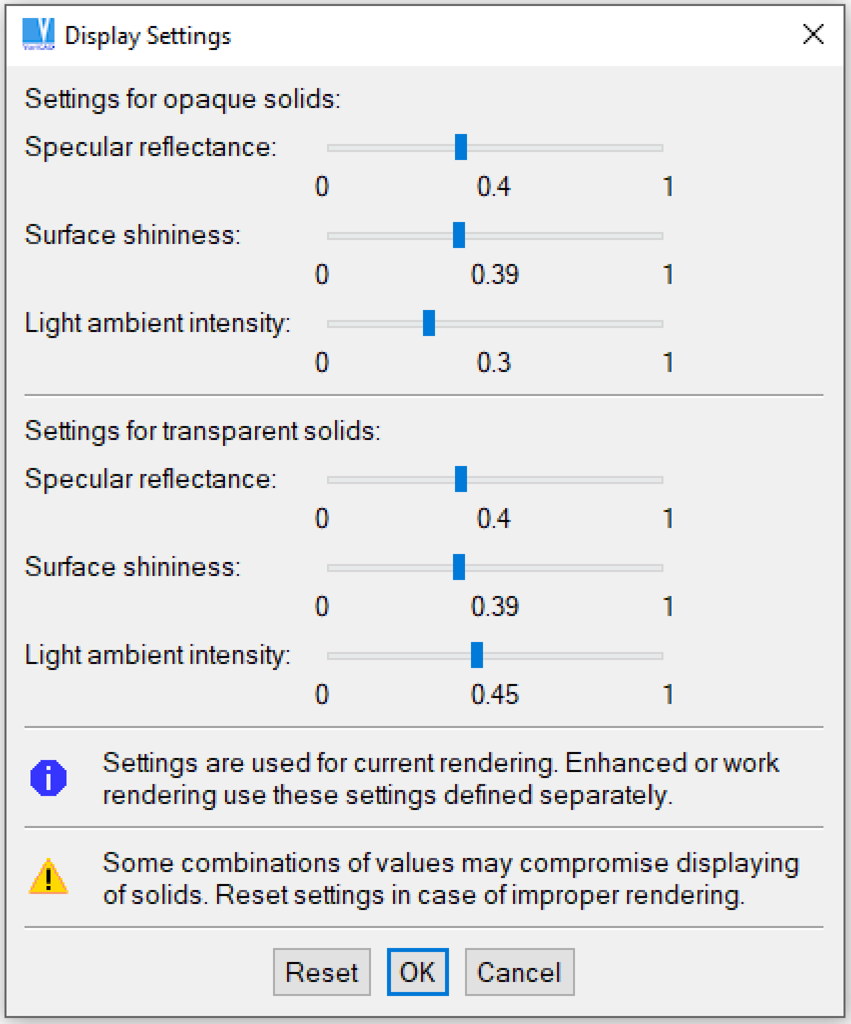
Surface Shading

For surface shading, you can set the following attributes:

- Specular reflectance. Defines lightness of surface lighted under zero angle. If set to 0, no light spot under light source is displayed and surface shininess setting is ineffective.
- Surface shininess. Defines contrast of the light spot under the light source.
- Light ambient intensity. Defines intensity of scattered light. This value can be set only for precise displaying.

It is recommended to combine enhanced rendering with selection of a proper color palette. All values can be changed by cursor movement, effect is seen immediately and values can be reset whenever.

The settings are different for shaded solids and for transparent solids.



Shading surface settings window

Setting 3D Display Performance

During start, VariCAD detects supported OpenGL version. If available, OpenGL 4.3 or at least 4.0 is used. For older hardware or insufficient OpenGL support, VariCAD works with OpenGL 1.1. In such situation, some features may be limited.

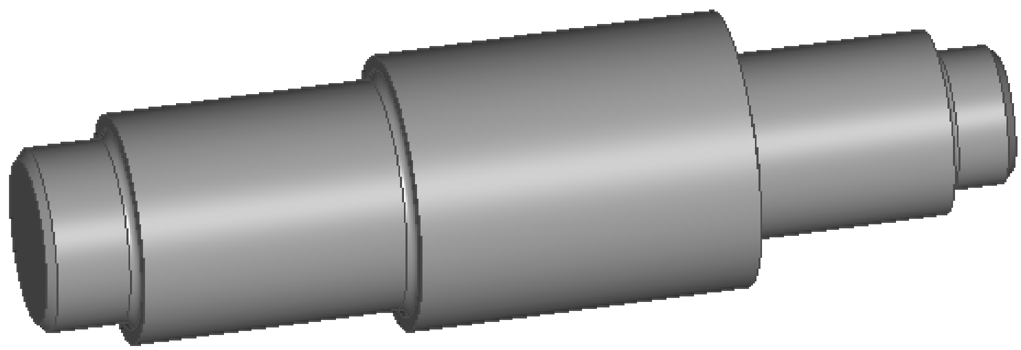
OpenGL 4.0 and higher allows you to fully exploit power of graphic adapter (card). Rendering time of large 3D data is significantly shorter than under old OpenGL 1.1.

If necessary, you may redefine which OpenGL version is used – although we recommend to let the decision at VariCAD system. To change the OpenGL version, run command “CFG” and then select “OpenGL Version and Initialization Settings”.

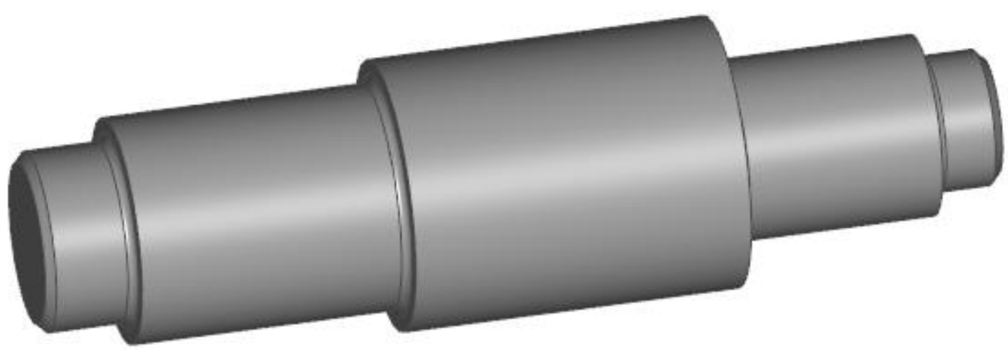
To change functionality of OpenGL, you may run command “CFG” and select “Performance, 3D Graphics, OpenGL”. For OpenGL 4.0 and higher, you may select:

- Usage of render buffer for detection (by default on, change only in case of problems).
- Image quality – anti-aliasing. For most of hardware, we recommend to work with compromise between speed and quality – Multisampling 4x.

If hardware does not support anti-aliasing sufficiently, VariCAD automatically turns it off. You may override such blocking.



Example of the aliased display method



Example of the anti-aliased display method

For old OpenGL 1.1, you may select:

- Method of display of 3D objects (working with display lists)

- Method of detection of 3D objects
- Highlighting of 3D objects

Again, we do not recommend to change the behavior, if there are no problems detected. Changing these settings may be useful only if hardware or display driver performance is erroneous and you are looking for a method how to circumvent problems.

Hardware accelerated OpenGL

VariCAD is an OpenGL application. It means that for displaying of 3D objects, VariCAD uses the OpenGL functionalities. The OpenGL standard is widely supported by hardware manufacturers.

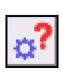
We recommend to work with hardware accelerated OpenGL. It means that final displaying of 3D objects is calculated directly in the graphic adapter (or, in other words, in the graphic card). Even in case the computer is fast enough, displaying is slower if calculated by CPU. Moreover, software emulation of OpenGL may not be always fully compatible with the standard. This may be a problem especially for older hardware.

If you are working under Windows operating system and if your computer has a graphic adapter like NVIDIA or ATI, OpenGL is likely hardware accelerated. Only some cheap solutions working with graphic adapters integrated on the board may be slow.

Under Linux operating systems, OpenGL may not be fully supported. To solve this problem, we recommend you the following:

- Make sure if you have the latest proprietary driver of the graphic adapter. You can download it from corresponding web sites (for instance, from http://www.nvidia.com for NVIDIA graphic cards). You can also find installation instructions there.
- The installation of the hardware accelerated driver may be necessary after upgrading Linux. Sometimes, Linux may offer the installation of accelerated drivers automatically.
- If the proprietary driver is installed and VariCAD still displays a warning message, or 3D graphic remains slow, look at the file “errors.txt”. This file is in your working directory. It may contain clues explaining why the graphic adapter does not work properly.
- Also, the system may not access related device files, because they have not correct permissions. This can be solved if you run the “User managements” and add membership in a group “video” for each user. The related device files can be accessed by the user group “video” and if the user is not a member of the group, applications launched by him cannot access corresponding files.

Test of Hardware Performance



Test of Hardware - HWTEST

This function is useful if you want to compare performance of different VariCAD settings or different hardware, like graphic adapters, main boards, processors etc.

The best option is to run the test after start of VariCAD, with no open files. VariCAD creates its own benchmark file. Then, hardware performance is measured. Results are displayed and can be saved into a text file. For the benchmark file, results are comparable. Optionally, you may run the test with any currently open file.

The test measures and displays:

- Hardware and system configuration
- Time of display preparation (tessellation). Result depends on CPU and GPU (Processor and graphics).
- Time of rendering of all objects. Result depends on GPU (only graphics).
- Time of detection preparation. Result depends only on GPU.
- Optionally, time of rebuilding of 3D solids. Result depends on CPU (Computer processor)

Results depend also at other hardware components.

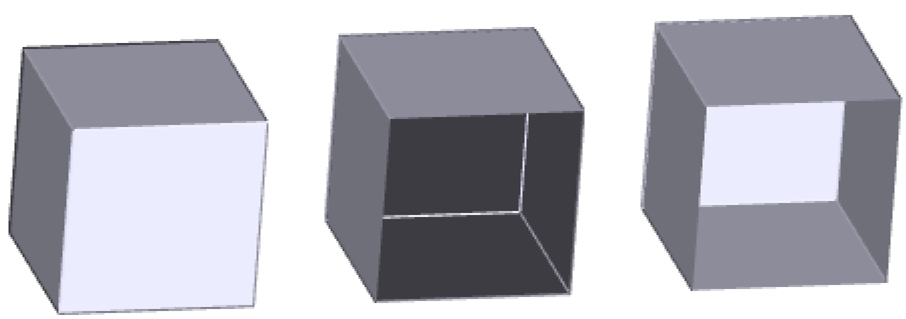
3D Objects Shape Representation

3D objects created in VariCAD are described as closed surfaces (closed shells). Objects loaded from STEP may be also open shells, or in case of detected error, surfaces originally closed - with some missing patches. Solids with missing patches, open shells or set of patches may be also created by conversions from other types – by VariCAD commands.

All types of objects can contain solid attributes, so they appear in BOM or other lists of parts. All types can be exported from 3D into 2D, as a part of 2D drawing. All types can be saved into STEP, IGES or STL. You can detect locations or measure distances or coordinates of points at all types. You can also define constraints among solids for any of these types.

On the other side, you can calculate volume, mass or moment of inertia (related to rotation axis) for only closed surfaces – for ordinary solids.

Types of Shape Representation



Example of a closed surface, missing patch and open shell

Closed Surfaces (Closed Shells)

Closed surface means that an object – solid is completely surrounded by patches. Each patch has outline created by 3D curves. Each such curve connects two adjacent patches. All operations available for 3D objects in VariCAD are supported.

Solids with Missing Patches

This type of object was originally created as a solid, described by closed surfaces. During STEP import, due to detected problems, one or several patches were deleted. Also, patches may be deleted in corresponding VariCAD command – see below.

Objects with missing patches support most of VariCAD 3D operations, except volume and moment of inertia calculations. Boolean operations and edges blending may be limited, though. These functions cannot work, if intersection curves created, for instance, by Boolean cut intersect an edge of a missing patch.

Solids with missing patches are displayed as “objects with holes”. Thru missing patch, you can see the solid from inside – patches are inside out there.

Open Shells

Open shells are also described similarly as closed surfaces by patches and 3D curves with two adjacent patches. However, open shells contain also curves surrounding only one patch.

Open shells do not support any volume or moment of inertia calculations, and do not support any Boolean operations or blending. If necessary, you can convert open shells to objects with missing patches and perform these operations at converted objects. Also, you can select one or more or all patches and create a shell with defined thickness, by offsetting of selected patches. See *Offset Patches – Thick Shells (page 274)*.

Open shells are displayed as zero-thickness patches. The surface looks the same from both sides.

Set of Patches

Set of patches are described by set of single patches. Each patch is surrounded by its own outline. No neighboring patches are defined. This shape representation supports only basic operations (see above). If you convert a selected object into this representation, you cannot convert it back (except undo of editing command, if you do not leave VariCAD session).

If STEP contains shape representation by triangular facets (similar to STL data), VariCAD can load these triangles as a set of patches. However, such representation is only approximate. Different surface types than planes are replaced by certain number of planar facets. Moreover, this representation requires a lot of RAM.

Converting Shape Representations into Different Types

Although different shape representation than closed surfaces (solids) is used among objects loaded from STEP, you can also convert objects into other types by VariCAD commands.

There may be several reasons for such conversions:

- Objects must be saved into STEP for other software, which demands open shells or set of patches, but not closed solids.
- If you need to perform Boolean operation or blending at open shell, you can convert it into object with missing patches

For conversion, you can right-click an object and select a conversion type from a pop-up menu. However, this selection is not possible for the most standard type – for closed surfaces (solids).

Otherwise, use following commands:

Converts a Solid to Open Shell - OSHELL

Select patches which will be removed during conversion. Also, you can optionally select patches which will create the open shell (as an opposite possibility of selection of patches).

Converts a Solid to Object with Missing Patches - PTCHM

Selection of patches is similar as for creation of open shell

Converts a Solid to Set of Patches - PTCHS

Selection of patches is also similar as for creation of open shell

For the commands above, selection of patches offers several options:

Select All except Detected – all patches of entire solid are selected, except a patch which was clicked. This option may be convenient if you need to remove only one patch, or a few patches – others are selected in next steps.

Select One Side of Shell – click a shell, created in VariCAD as offset of patches. Entire surface is selected. Option may be used, for instance, if you need to export a shell with defined thickness into a system demanding only open shell with zero thickness.

Entire Solid from Sheet Metal – click a solid. If the solid is created as a sheet metal object, entire corresponding side (surface) is selected.

Selected Patches Are Saved – if checked, all selected patches are used for solid shape conversion.

Selected Patches Are Removed – if checked, all selected patches are removed before solid shape conversion. This is opposite possibility to the previous option.

Highlighting Objects with Open Surfaces

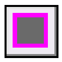
As written above, some operations performed at open solids are limited or not supported. VariCAD provides possibilities to highlight such objects. You can also highlight boundaries of continuous patches of open shells or solids with missing patches. Consequently, you can easily see which objects from entire space do not support all features available for standard closed solids.

To display boundaries of solids with missing patches or boundaries of open shell, right-click an object and select the feature from menu. Also, open solids can be displayed in command *3D Space Information (page 277)*.

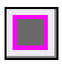
Following possibilities of open shells checking are available:

 Displays All Open Solids - DOS

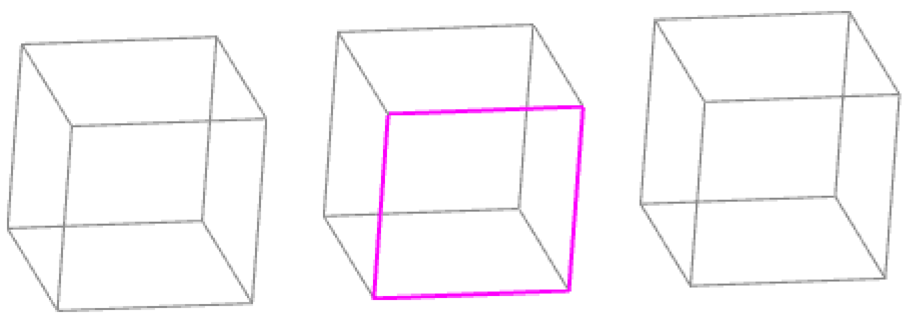
This command optionally highlights all open solids or all boundaries of continuous patches.

 Displays All Holes in Open Solid - DHOS

Command displays boundaries of continuous patches of selected object. Boolean operations or blendings are not supported, if intersections calculated during operation further intersect these boundaries.

 Displays All Holes in Open Solid Around Deleted Patches - DHEOS

Command displays boundaries of continuous patches of selected object, similarly as the previous command. However, only boundaries of deleted patches are highlighted. Deleted patches are removed during input of STEP files, if an error is detected and the patch cannot be processed. This command is useful as detection of incompletely loaded objects from STEP.



Example of highlighted boundaries of an open solid, here a solid with missing patch

Solving Problems in 3D

VariCAD provides tools for solving problems or for partial reparations of corrupted internal data, files or objects loaded from STEP.

Tools Rebuilding 3D Data Structures

 Regenerate All 3D Objects - REGALL

This command rebuilds all 3D objects in space. In case of internal data errors, corresponding solids are exploded into single elements. A new 3D group is created for each erroneous solid and the elements are added into this group.

The regeneration of all objects may detect possible problems during solid shape editing.

 Regenerate All 3D Objects and Repair Transformations - REGTRAN

The command rebuilds objects similarly as command REGALL. Moreover, this command repairs data related to transformation (location) of each object. The reparation of transformations may help in case when solids cannot be edited.

Both commands REGALL and REGTRAN cause end of history of edit changes. A new history (Undo – Redo of commands) starts again from scratch.

 Converts a Solid to Imported Object - TOIMP

The command converts a selected solid to imported object. After conversion, all history of solid’s creation is lost. The new object’s geometry can be changed the same way as the geometry of solids imported from STEP. Only new Boolean operations and new blendings can be performed.

If you encounter a problem when an existing solid cannot be edited and the edit change causes explosion into single elements, do following:

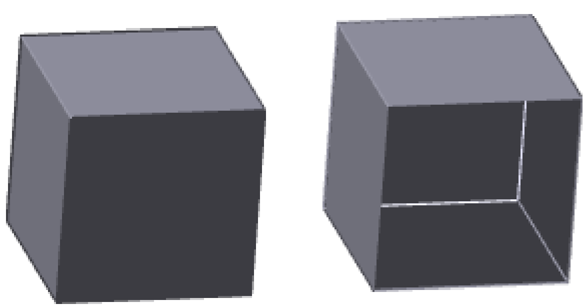
- Close the file without saving.
- Open the file again.
- Convert the erroneous solid (before it is exploded again) into imported object.

Although the existing shape can be modified only by cut, add or blend operations, further editing should not cause errors.

Tools Repairing Erroneous Solids Loaded from STEP

Rarely, some objects loaded from STEP may have reverted normals – either all normals on entire solid, or one or a few separate normals. This is caused by corrupted data recorded into STEP by other software. Reverted normals can be described as “inside out” patch or solid.

Patches with reverted normals may resemble solids with missing patches. But there is a difference – you can detect a patch with reverted normal. In case of missing patch, you see other patches from back side, through a hole.



A solid containing a patch with reverted normal and a solid with missing patch

Use following commands to manage reverted normals at a selected solid:

Reverse All Normals - INSIDEOUT

Reverse Selected Normals - INSOUTSEL

Tools Repairing Erroneous File

If a file created in VariCAD cannot be open again, recovery commands may help to solve the problem:

File Recovery - RECOVERY

Select a file from file dialogue. Optionally, you can select opening of file without active sections. The command rebuilds some internal data structures, and the file may be opened again.

If a VariCAD native format cannot be open, it is caused rather by power failure during file saving or shortly after the “save” is finished by VariCAD. In such case, the file recovery does not help.

Another method how a file can be recovered is to insert a corrupted file into an existing 3D space, or 2D area. This method can insert only 3D part or 2D part. If the 3D part is damaged, you can recover at least the 2D part, for instance.

Sketching - 2D Drawing in 3D Planes, Drawing Methods

If you create a new profile further used for rotation or extrusion into 3D or if you edit an existing profile of solid, VariCAD uses 2D drawing functions at a sketching plane in 3D space. 2D drawing in 3D uses the same functions as the standard VariCAD 2D editor or 3D modeler, but they are limited only to drawing, editing or display control.

You must select the sketching plane first, if you create a new profile (see *Creating 3D Solids from 2D Profiles (page 171)*). Otherwise, the sketching is started whenever you want to edit the shape of a 3D basic solid or solid element (a part of Boolean tree).

Displaying Objects

The 2D profile is drawn in one color. You can switch display of objects using the following functions:

Toggle between thick and thin outlines display in 2D

Switch on/off the auxiliary grid in the drawing plane

Toggle all 3D solids display between shaded and wire-framed

Work with zoom, pan or view rotation or undo-redo of view changes are the same as in 3D. There are additional methods related to the 2D drawing plane:

 Define zoom by two window corners in the drawing plane

 Drawing plane perpendicular to a view

 Zoom all in the projected drawing plane

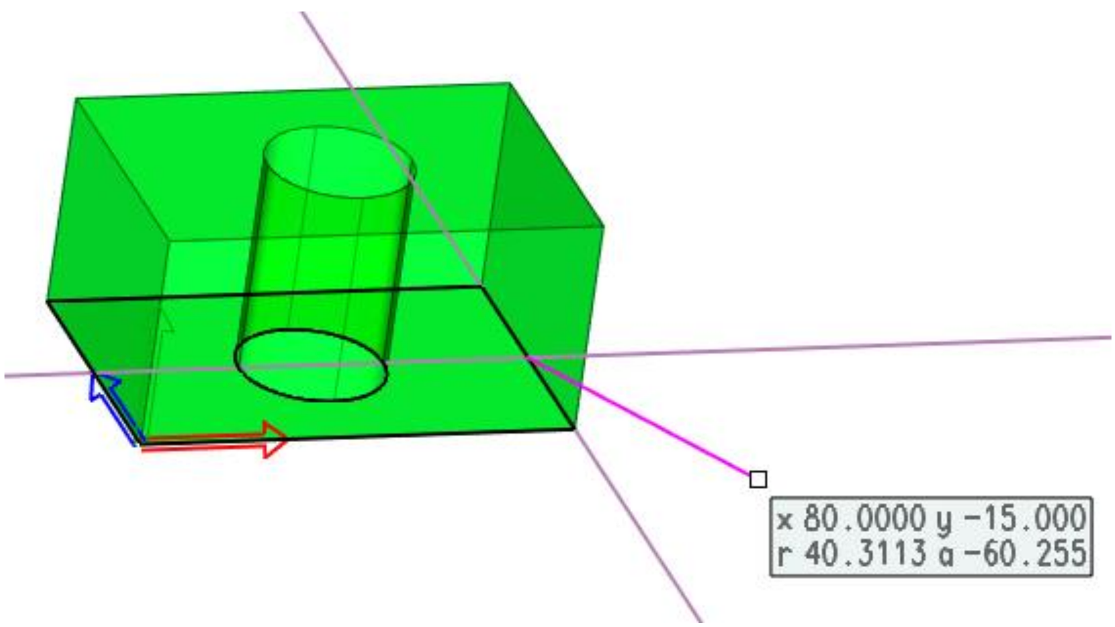
 Select a center of 3D view rotation in the drawing plane

2D Drawing Features

The 2D functions are limited to drawing or editing lines, arcs, curves or points. Sketching does not support any commands working with texts, dimensions, blocks or hatches. You can use Copy and Paste; insert an existing drawing into the drawing plane or save selected objects to a file. If unsupported objects (like text objects) are inserted from another drawing, they are automatically deleted.

Construction Lines, Temporary Construction Lines

Construction lines can be used the same way as in 2D mode. In sketching, it is very convenient to use temporary construction lines (leading lines) for 2D drawing of lines, multi-lines or other objects – see *Temporary Construction Lines (page 41) (page 27)*. If you create or edit a profile used for rotation, the rotation axis is displayed and is, in fact, another construction line. Standard construction lines and temporary construction lines or rotation axes are distinguished by different color.



Example of temporary construction lines in sketching. Temporary lines are at the last location entered during line drawing.

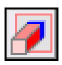
Working with 3D Objects

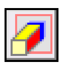
During the definition of a location in the 2D drawing plane, you can select the location in 3D space. The result is the nearest perpendicularly projected point in the drawing plane. Click the following icon in the 2D location toolbar to allow this:

 3D location projected into the drawing plane

You can also press the key ‘q’ or ‘!’ to allow the 3D location projection into the sketching plane.

You can also create new 2D objects (lines, arcs or curves) as an intersection of a selected 3D solid and plane:

 **2D Object as Solid X Sketching Plane - SXDP**

 **2D Object as Patch X Sketching Plane - PXDP**

Creating Solids

Many 3D solids can be created by extruding, rotating or lofting 2D profiles. Other basic solids such as cylinders, boxes, cones or pyramids can be defined by entering dimensions. Nearly all mechanical parts contain basic solids, which can be joined and/or trimmed. Combining and subtracting solids are called Boolean operations, and resulting solids are called “Boolean Trees.” VariCAD provides tools to add solids and to use one solid to cut another, either keeping or deleting the cutting solid. Commonly-used Boolean operations such as drilling holes, creating grooves, and cutting by a large box are also available. Blending functions are provided for rounding and chamfering solid edges.

Following two chapters describe how to create basic 3D solids, which can be later combined into Boolean trees.

Sketching of a 2D Solid Profile

When using a 3D solid creation method that requires a 2D profile (or multiple profiles in multiple planes) as an input, the profile is created in sketching plane in 3D space. Optionally, you may select a profile in 2D area; however, such method is limited only for solids created by single profile in one plane. In sketching plane, the profile is created by standard 2D drawing features. See *Sketching - 2D Drawing in 3D, Drawing Methods (page 169)*. If a solid shape is edited, the profile is always modified in sketching plane.

Sketching Plane Definition

Before you start sketching of a profile, you must define the sketching plane. It may be selected as:

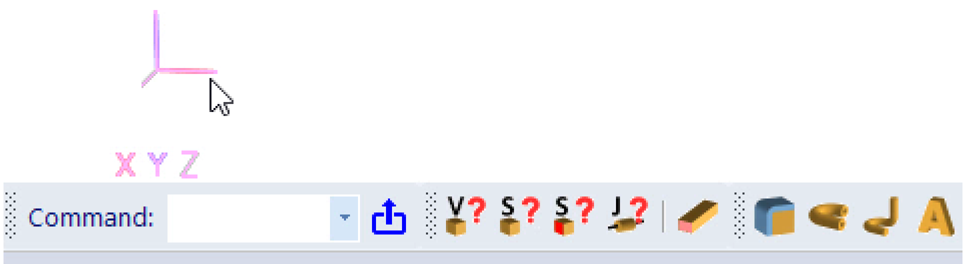
- As an existing plane at a solid
- Plane created by XY, YZ or XZ axes of a solid
- Plane defined by 3 points
- Plane created by XY, YZ or XZ axes of 3D space

An existing plane at a solid can be conveniently selected, if you press and hold Ctrl + Shift and move cursor over a planar solid's surface. Plane's outlines are highlighted and the plane is selected by left click. Outlines are pre-defined as an existing profiles 2D objects.

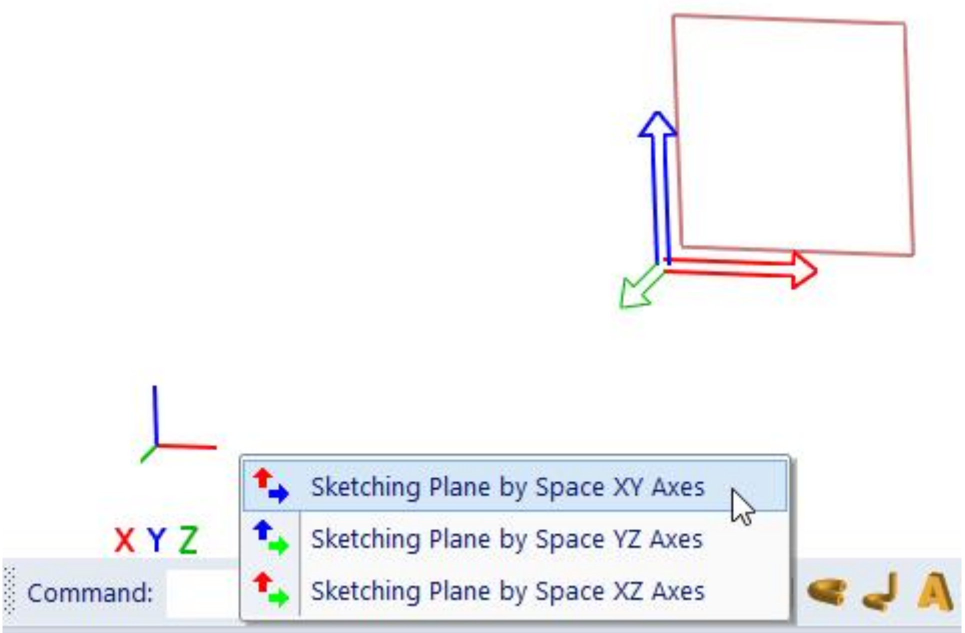
 Sketching (2D Drawing in planes in 3D Space) – DRP, Alt + S

To start sketching, click the spatial axes in lower left corner of VariCAD drawing window. Select sketching plane definition from menu.

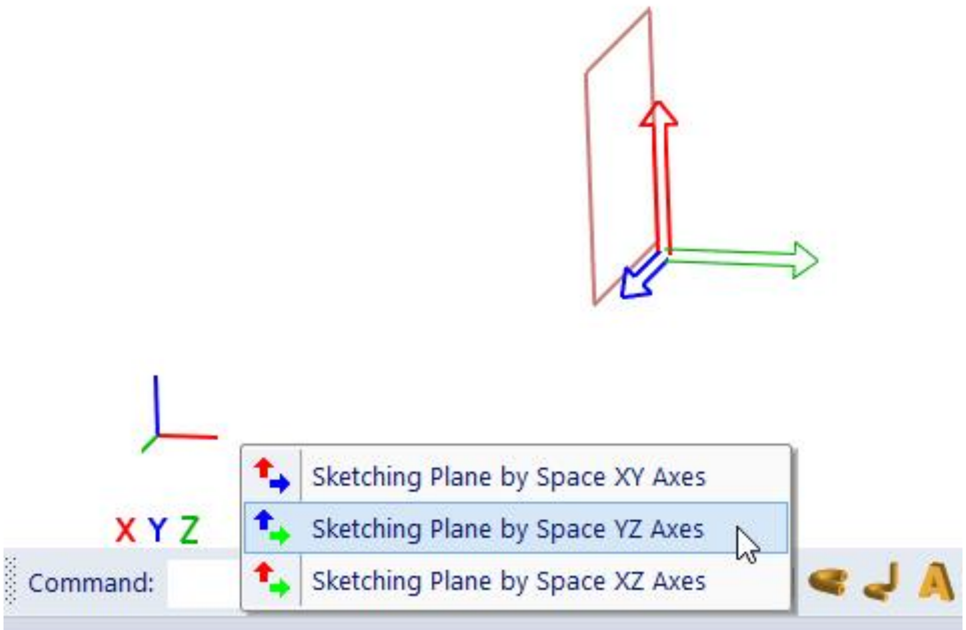
If you want to start sketching at an existing plane (planar surface of a solid), press and hold Ctrl + Shift and move cursor over the plane. Left-click when plane's outline is highlighted.



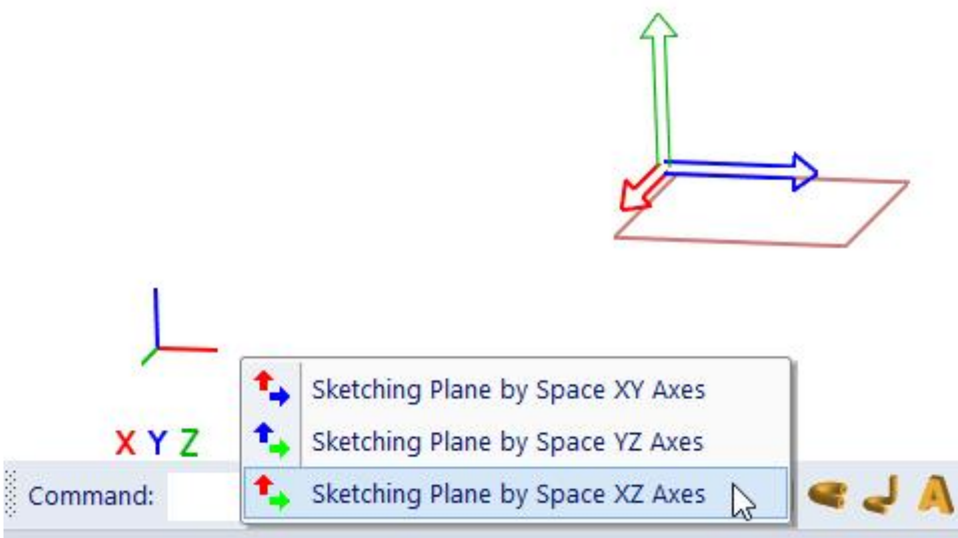
Detection of spatial axes, menu is displayed after click



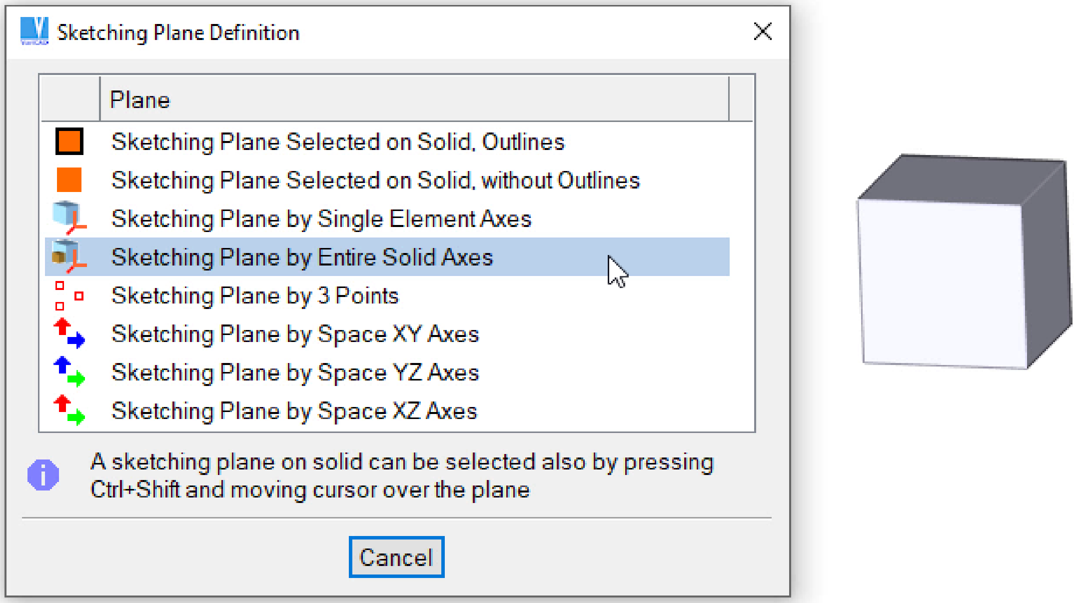
Selection of axes. This pop-up menu is displayed if 3D space is empty. Sketching plane corresponding to menu item is highlighted. Plane axes are displayed as axes of sketching plane, not as world axes.



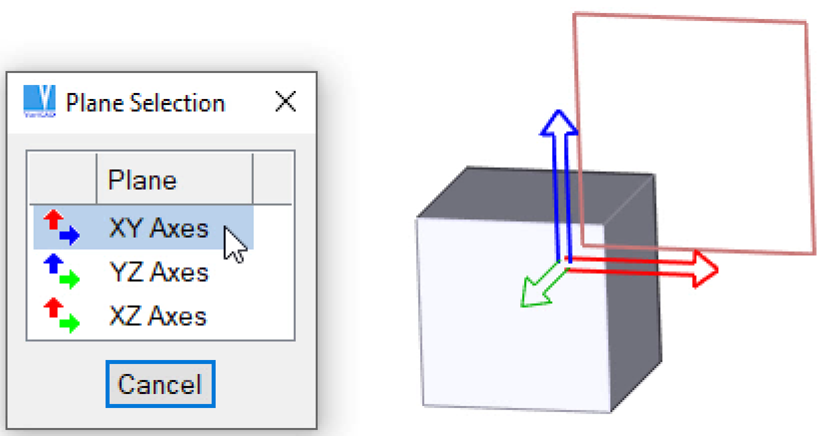
Next option, selection from world axes.



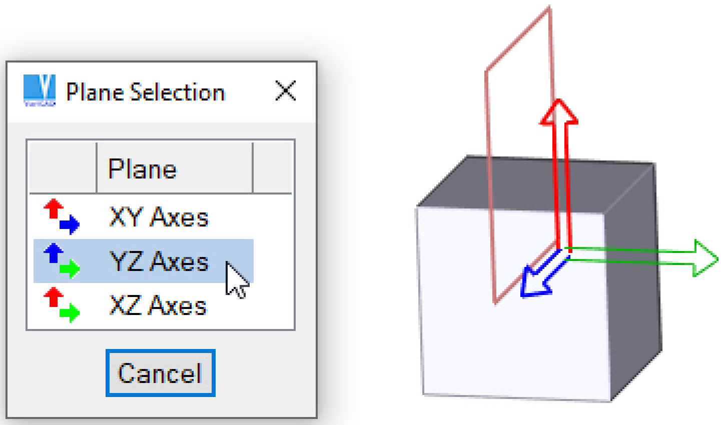
Final option, selection from world axes.



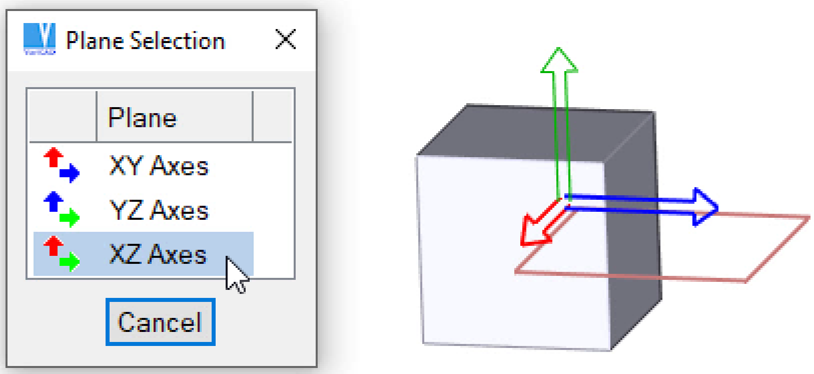
This dialog panel is displayed, if 3D space contains solids. Sketching plane can be selected also at solid planar surfaces or solid axes.



Selection of axes. Sketching plane corresponding to menu item is highlighted. Plane axes are displayed as axes of sketching plane, not as solid axes.



Next option, selection from solid axes.



Final option, selection from solid axes.

If you start creation of a solid defined by profiles extrusion, rotation or lofting, a pop-up menu appears and you can select sketching plane. If you select an existing solid for editing, the sketching plane is defined automatically according to solid location.

Sketching Environment, Finish Sketching

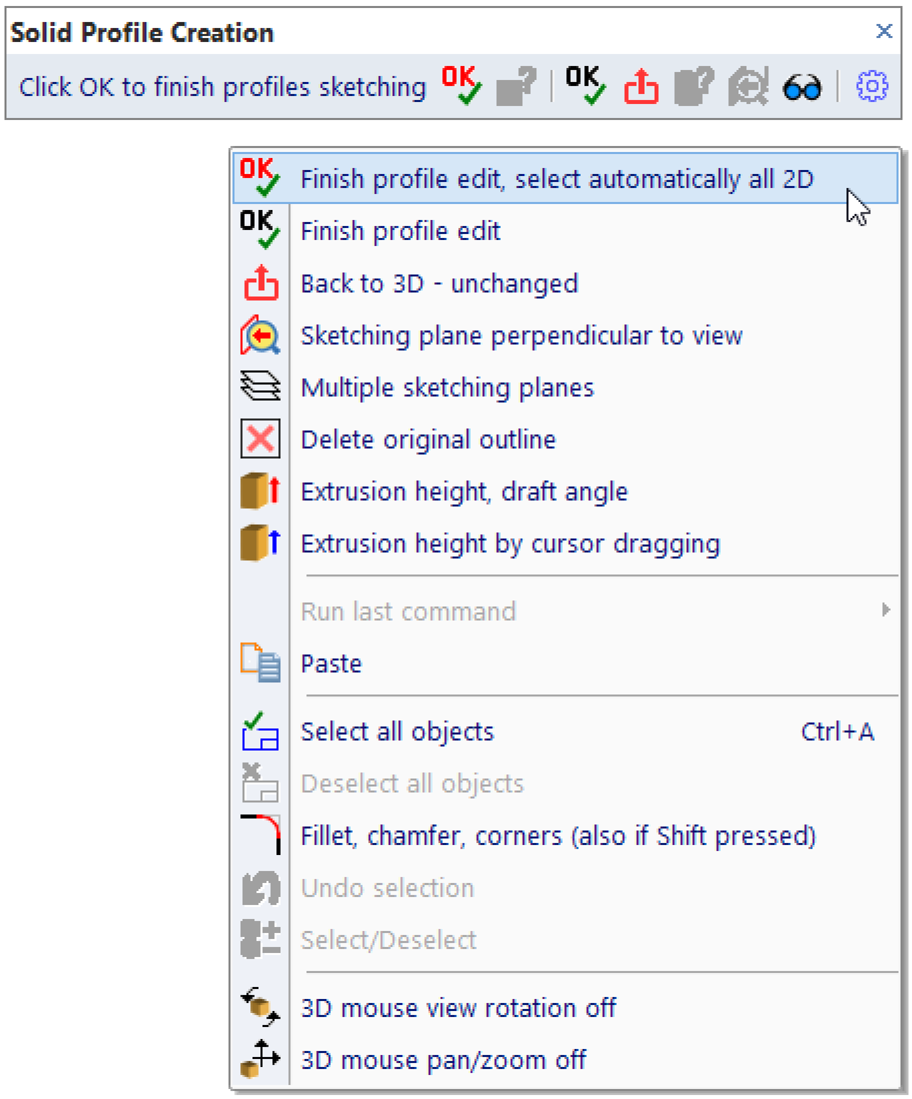
After sketching plane is selected or a solid is selected for profile editing, VariCAD switches environment into sketching mode. Toolbars are changed. You can use standard 2D drawing commands, with certain limitations – see *Sketching - 2D Drawing in 3D, Drawing Methods (page 169)*. A solid can be created from multiple profiles in multiple sketching planes. Undo or Redo of 2D commands is available for each sketching plane separately.

Sketching can be finished, if you:

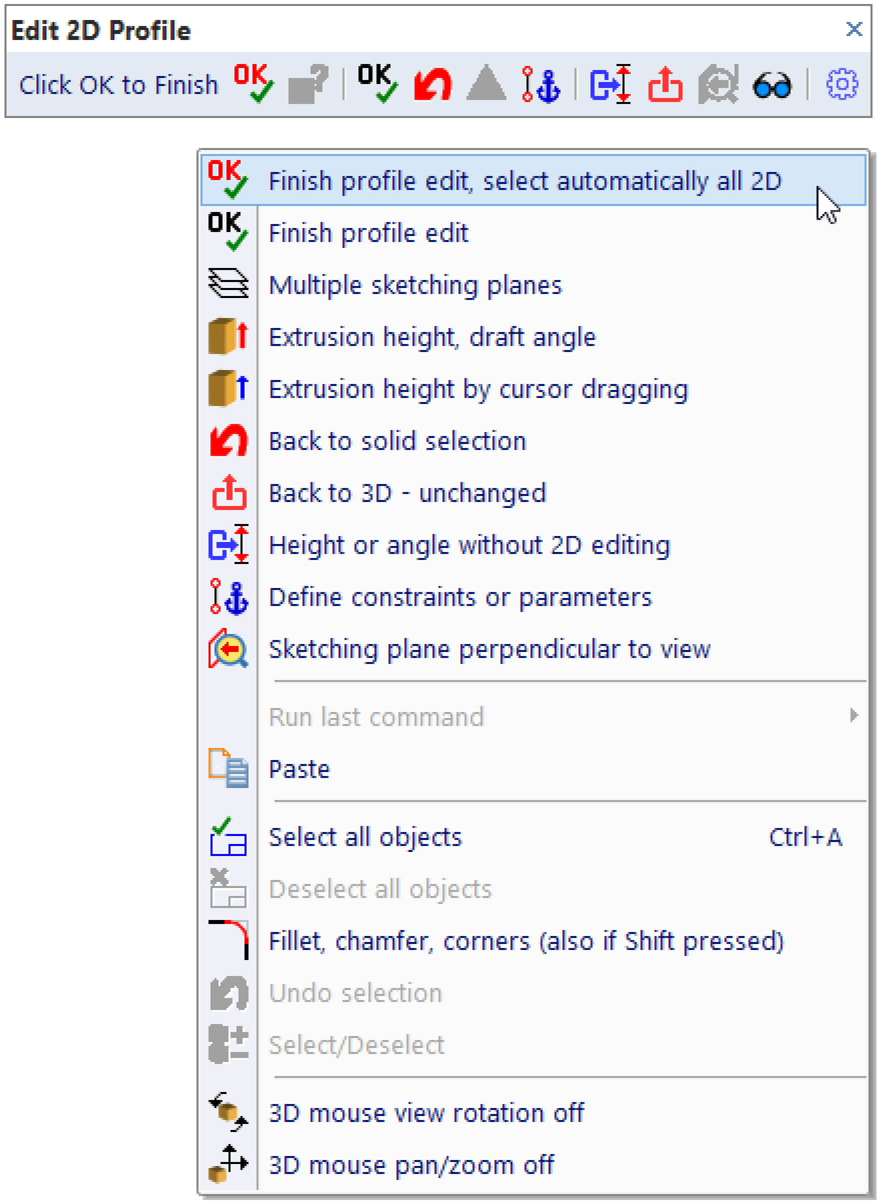
- Select “Profiles sketching finish” from toolbar or pop-up menu after right-click an empty area

- Perform Undo (Ctrl + Z) at a state where no prior 2D commands are available - at the beginning of sketching
- Click Close icon at menu bar, or call command closing the current document.

Commands specific to sketching are available from sketching toolbar or from pop-up menu, which appears if you right-click an empty location.



Toolbar and pop-up menu, solid profile creation



Toolbar and pop-up menu, solid profile editing

Sketching of Profile of New Solid

If you want to create a new solid and sketching has already started, you can create 2D objects of a new solid's profile. You have to select a method of solid creation. You can do this whenever during sketching, or at last when you finish the profile. According to creation method, you must also define solid's extrusion height or rotation angle, draft angle etc. Again, such values may be defined during sketching, or at last at the end of sketching.

To see how the new solid looks like, extrusion height or rotation angle is predefined. You must define the final value whenever during sketching.

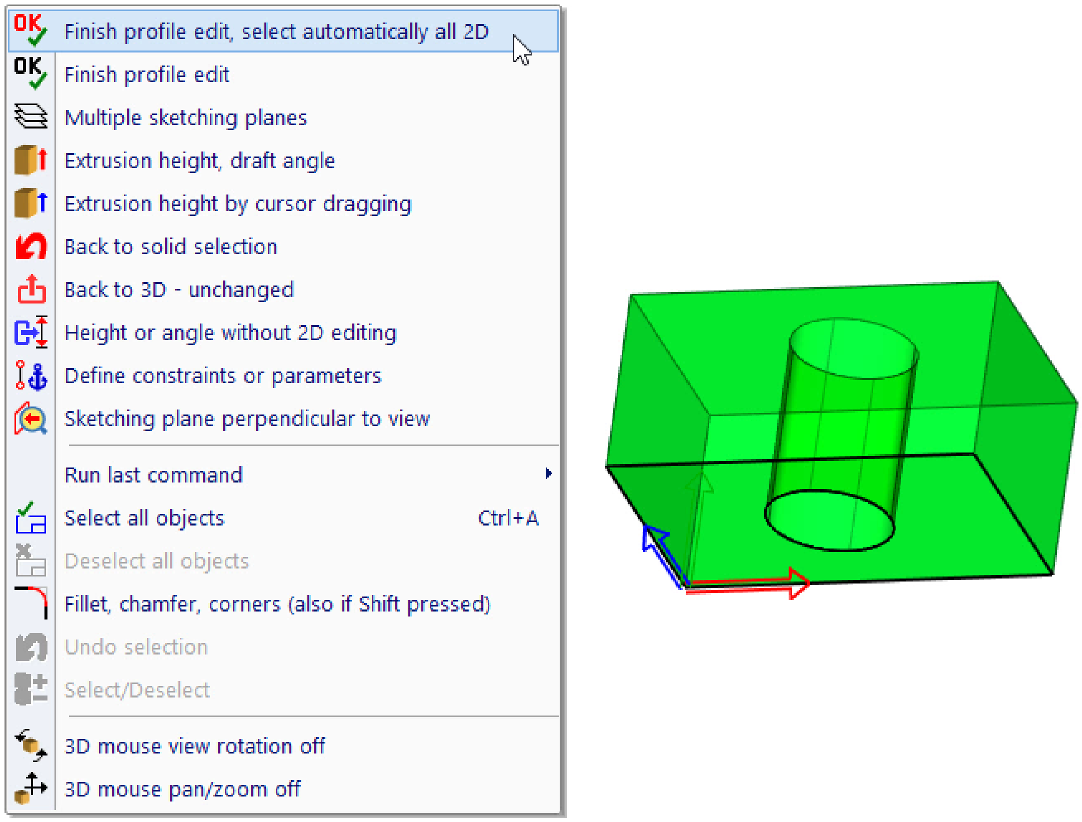
Available methods of solid creation from 2D profile or profiles:

- Full rotation
- Partial rotation
- Extrusion
- Sweeping along defined path
- Helix (extrusion combined with rotation)
- Lofting, including lofting of multiple profiles or lofting from rectangle to circle
- Lofting combined with rotation

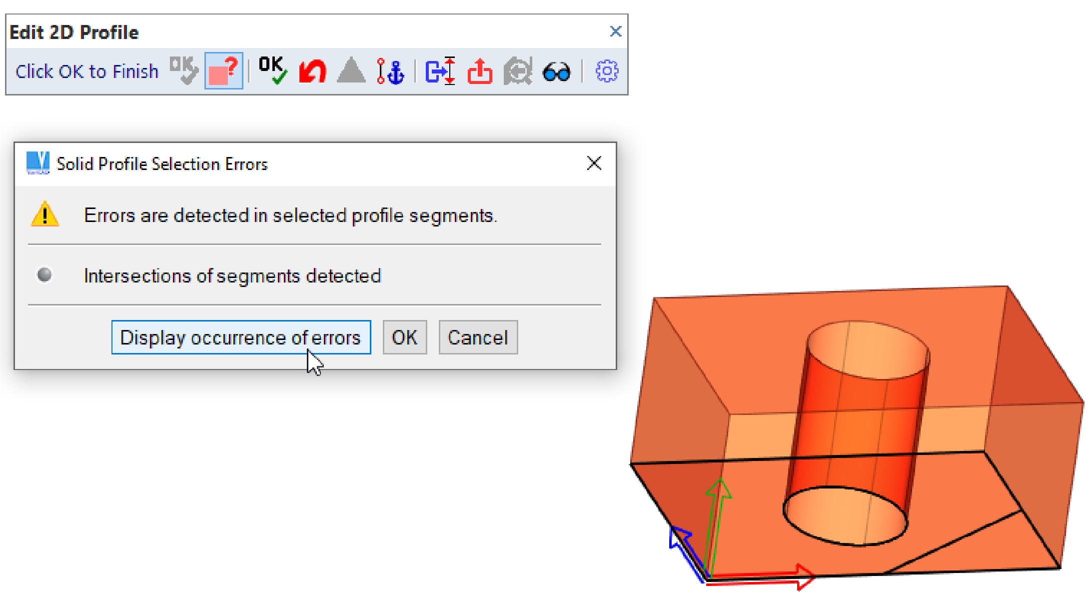
Displaying of Created or Edited Solids

Created or edited solid is displayed as a transparent object. By default, the shape is permanently updated after each change of the profile's shape. If a solid can be created from all 2D profile objects, it is displayed in green color. If the created or edited profile contains an error (like intersections of lines or gaps between lines' endpoints), the solid is displayed in red color at the last correct position.

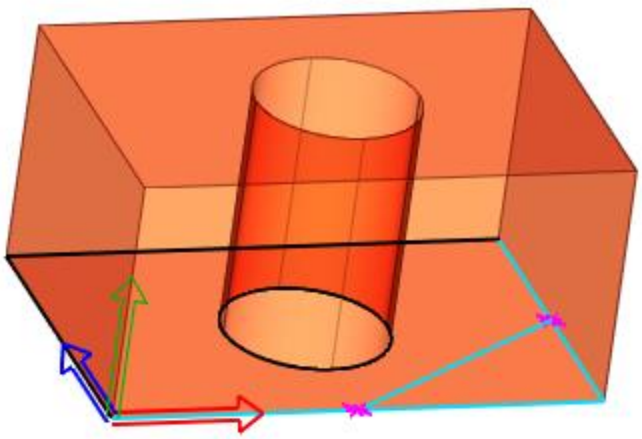
In case of erroneous profile, or whenever a solid cannot be rebuilt, you can click a corresponding icon and display what caused the problem. On the other side, if the profile is correct and the solid can be created, you can finish profile creation and skip selection of profile's segments – all 2D objects are selected automatically.



Example of correct profile. All 2D objects can be selected automatically and then the solid is created.

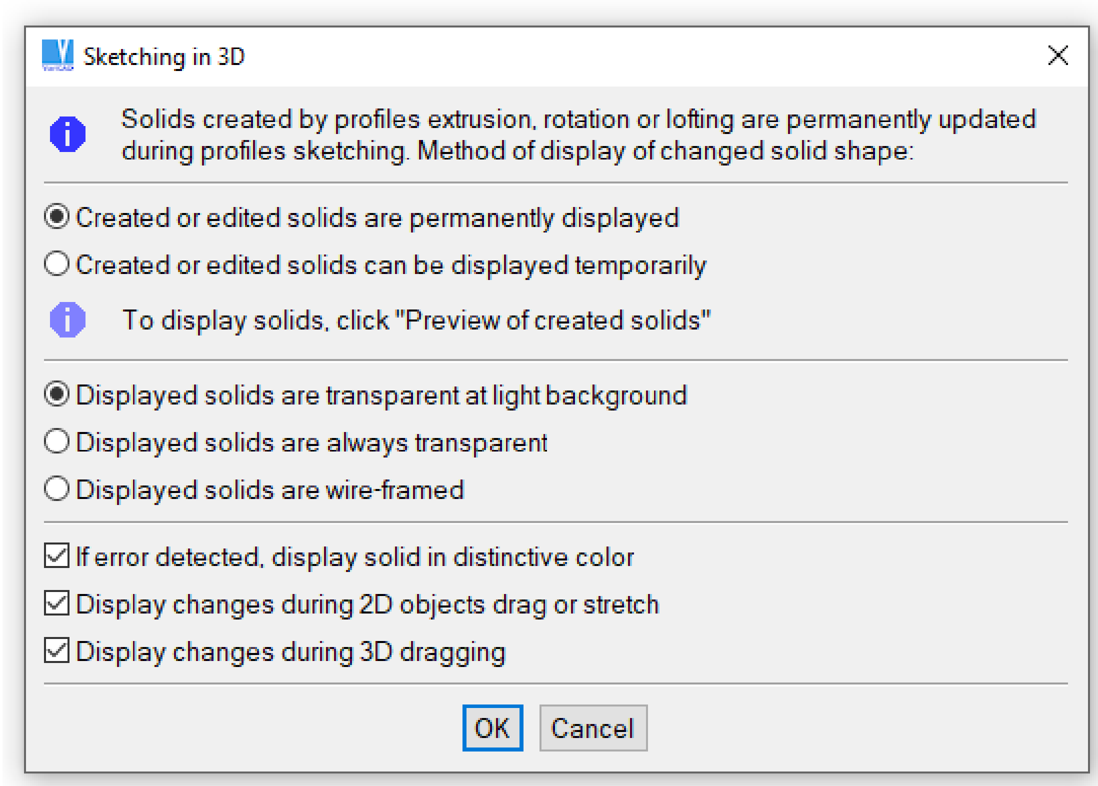


Example of incorrect profile. Click the button to display occurrence of error.



Errors in contour are displayed.

It is possible to manage how the edited solids are displayed. Wire-framed display can be used instead of transparency, or a solid may not be updated permanently.



Settings of display of 2D profile sketching

Multiple Sketching Planes

A solid may be created from multiple profiles. Then, each profile is created at a separate sketching plane. To select features related to multiple sketching planes, click XYZ axes of the current sketching plane and select action from menu.

In case of multiple sketching planes, only one plane is active. 2D objects are created or edited in active plane. Sketching plane is activated, if you click its contours.

First sketching plane is so called a base plane. This plane becomes active automatically, whenever the solid is edited. XYZ axes of the base plane are axes of the created or edited solid. If planes are copied or created as new planes, all planes must be above or under the base plane.

 **Create, Copy or Transform Drawing Planes**

This command manages sketching planes. It may be especially useful if a sketching plane does not contain any 2D objects. Because then, you cannot click its contours.

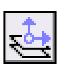
Following features are available at pop-up menu, after you click contours of inactive sketching plane or XYZ axes of active sketching plane:

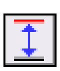
 Activate a sketching plane.

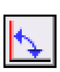
 Copy a sketching plane together with all 2D objects. A new sketching plane is transformed and located the same way as a new or transformed solid.

 Copy a sketching plane without 2D objects.

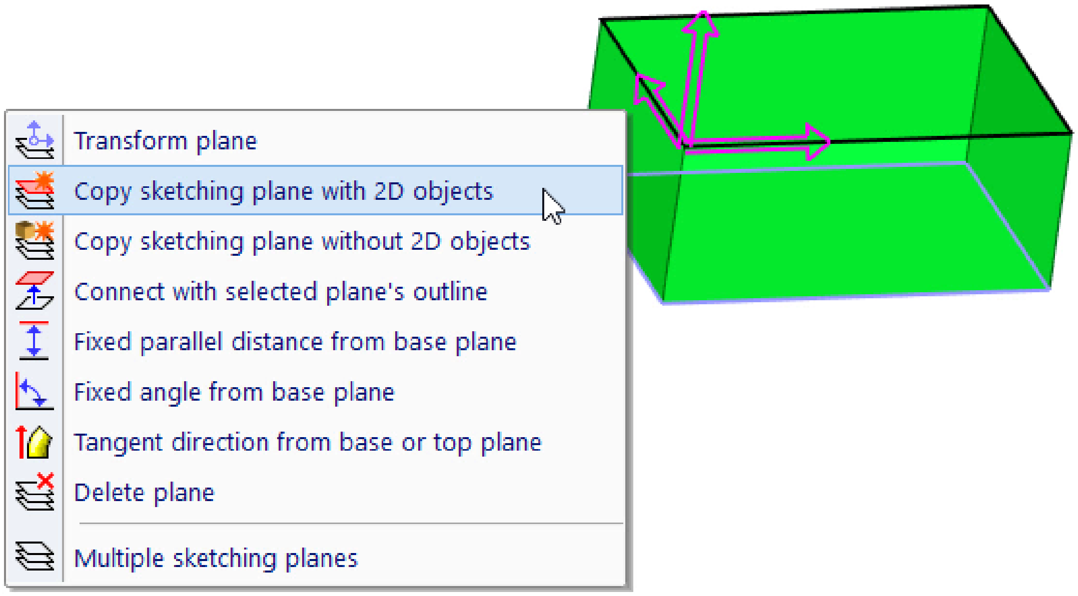
 Delete a sketching plane. Base sketching plane cannot be deleted.

 Transform a sketching plane. If transformation is selected for a base plane, the rest of sketching planes and consequently, entire solid is transformed. Otherwise, the transformation changes position of the sketching plane relative to other planes.

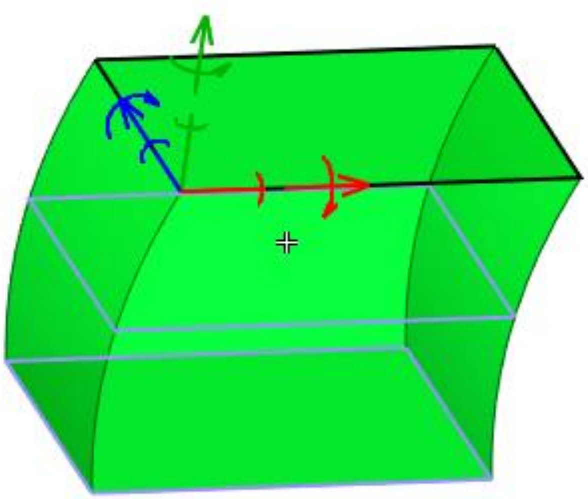
 Defines distance from the base plane. The selected plane is then parallel with the base plane and its coordinate center is exactly above the center of the base plane. Distance can be defined also as a mathematic expression containing parameters.

 Defines angle from the base plane. The selected plane is then rotated around X axis and its coordinate center is at a location of the base plane center. Angle can be defined also as a mathematic expression containing parameters. Angle between planes must not be greater than 60 degrees.

When a plane with 2D objects is moved or copied, created solid is recalculated permanently and if cannot be rebuilt, it is displayed in red color at the last correct position. Planes are sorted automatically, if you move a plane.



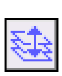
Sketching plane’s options, copying is selected



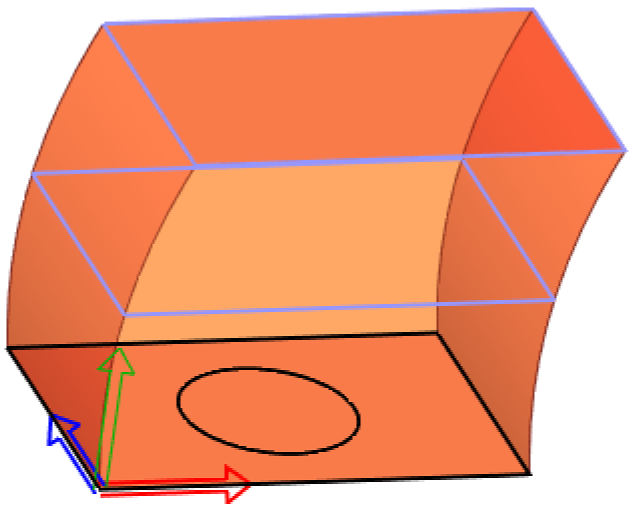
Sketching plane is copied together with 2D objects

Copying 2D Objects among Sketching Planes

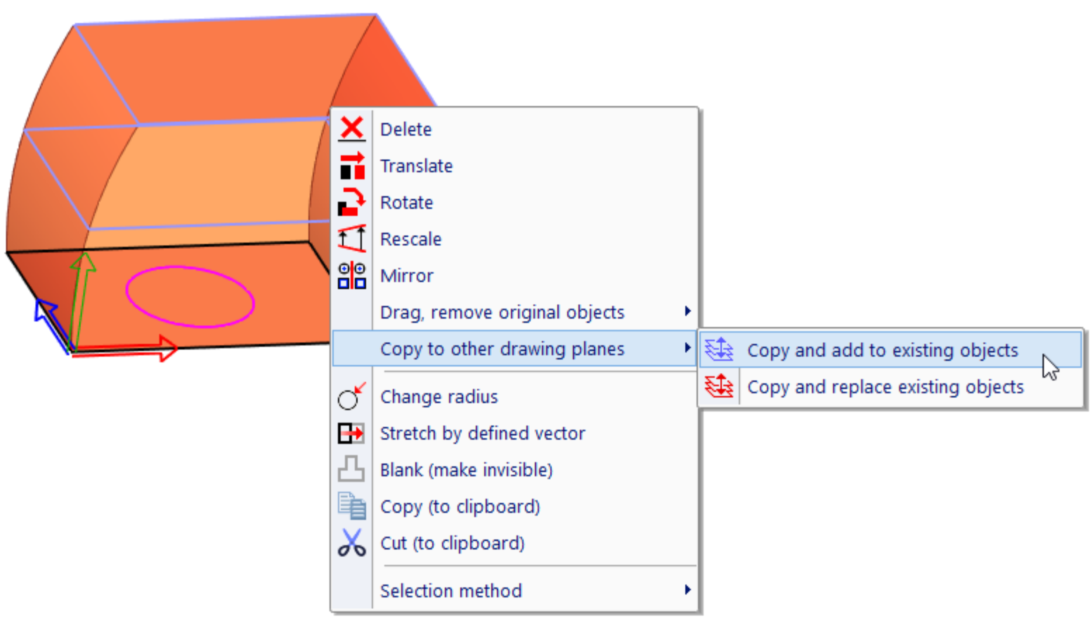
To copy 2D objects among the rest of sketching planes, select the objects. Then, right-click and from related sub-menu, select method of copying:

 Copy and add to rest of drawing planes

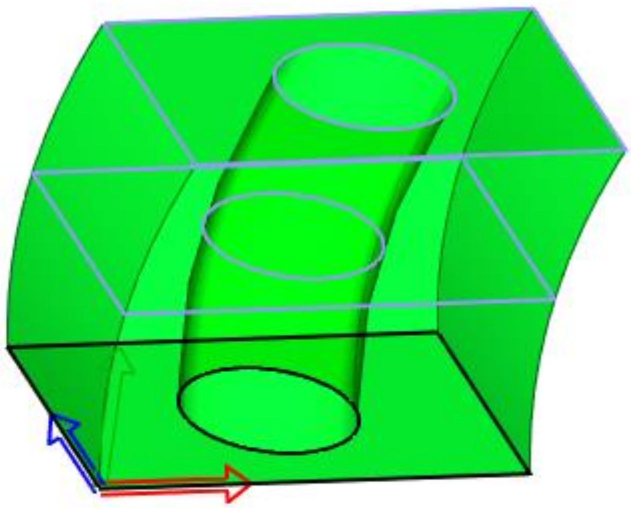
Copy and replace objects at each drawing plane. This feature is useful if all planes contain the same profile and if you need to change profile's shape.



A circle is created at base plane. Contours in other sketching planes do not correspond – a solid cannot be created.



Copying 2D objects among sketching planes, selection of 2D objects.



Solid was created after the circle was copied into all planes.

Generally, a set of 2D objects can be whenever and wherever copied, using Copy and Paste (Ctrl + C, Ctrl + V).

Common Sketching Features

These features can be selected from toolbar, or if you right-click an empty location.

Finish profile edit, select automatically all 2D objects. This command finishes edit. If you already defined values like extrusion height or rotation angle, entering of them is skipped. If you already transformed the base sketching plane or if a plane was selected as a planar surface of a solid, transformation of new solid is also skipped.


Display errors in created profiles. This option is active if a solid cannot be rebuilt. Otherwise, it is inactive and active is the previous option.


Quit sketching without any changes of edited solid, or without creation of a new solid.


If active, tilt of current sketching plane is too large. By other words, you would draw 2D objects to plane almost parallel or parallel with angle of view. Although you may continue, it is better to rotate view properly.

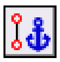

Preview of created solid – can be used only, if permanent changes of a created solid are turned off.


Settings of display of created solid.

Finish profile edit. This command finishes edit. It is not available for solids created in multiple sketching planes. In following steps, you must select profile's segments by cursor or by selection window. Then, define solid location, if a new solid is created.

Returns back to selection of solid to be edited. It is available only for editing of an existing solid.

If active, edited object is affected by constraints. Any change may cause additional transformations of other solids.

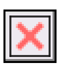


Define parameters and constraints within the current profile. This is available only for editing. If selected, sketching mode is finished and you can define constraints and parameters among profile vertexes.

Skip profile editing and set extrusion height or rotation angle of edited solid.

Additional Sketching Features

These options are available according to specific situations. They are available from pop-up menu after clicking an empty area. Dimensions of cutting tools can be selected from pop-up menu, if you click sketching plane axes.



Delete original outlines. This option can be conveniently used, if you select the sketching plane at a solid (by holding Ctrl + Shift and moving cursor over the plane). Then, the sketching plane contains 2D objects corresponding to detected plane's outlines. You may use these objects, but if you need them only for measurement or as auxiliary objects, you can delete them at one step, by this option.

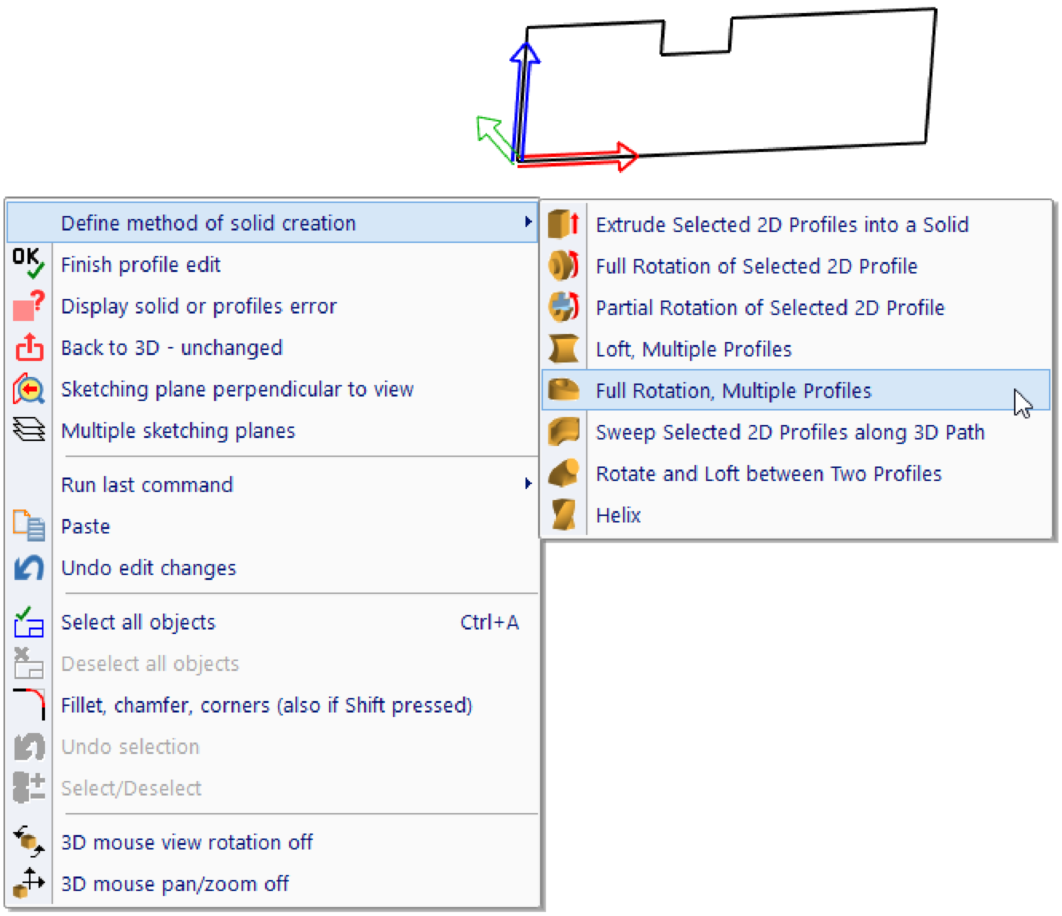
Cutting tool height and location. This option can be used, if you create a tool for cutting of a selected solid. It defines height of contours extrusion and location of sketching plane automatically, according to solid dimensions.

Section tool height and location. A similar option, used for sketching of contours of a section tool (section planes geometry).

The example of working with these options is at: *Cut by an Extruded Solid (page 218)*.

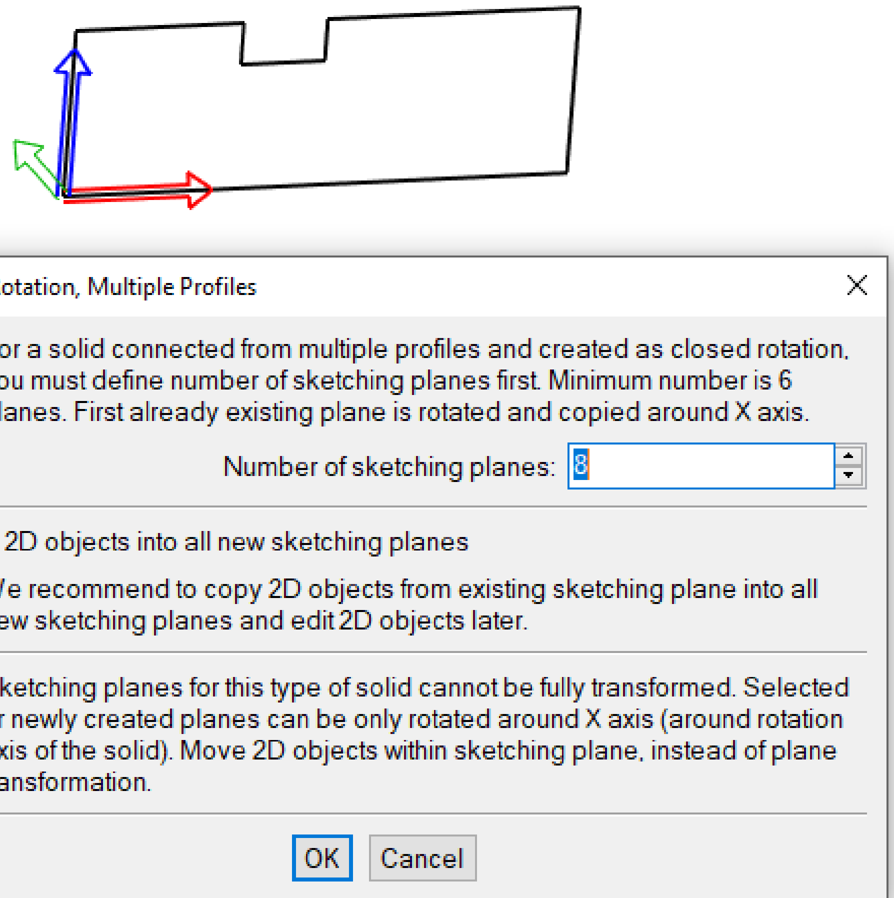
Closed Rotation of Multiple Profiles

You can create solids defined by multiple sections rotated around X axis. First, create a profile you want to rotate. Right-click empty location and select a method of solid creation.



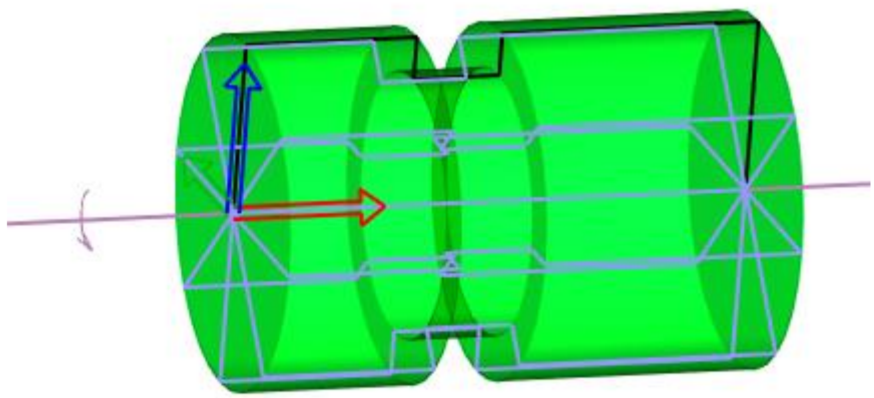
Selecting full rotation of multiple profiles

Then, select number of planes. Minimum are 6, recommended are at least 8 planes.

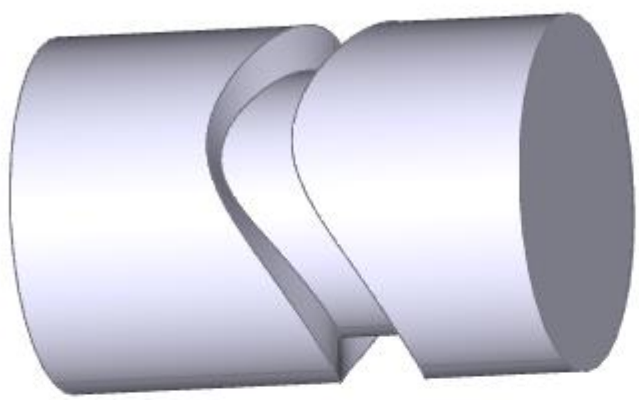


Full rotation of multiple profiles properties

A solid is created. Now, you can switch among sketching planes and modify each individual profile.



First solid created from rotated profiles



Groove location was shifted for each section individually

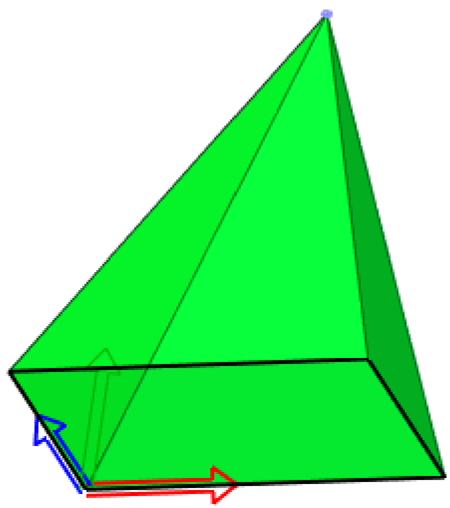
Convergence of Lofted Profiles into One Point

If the last sketching plane, listed in table as the Top Plane, contains only one 2D point instead of closed contours, all patches converge into this point. See examples below.

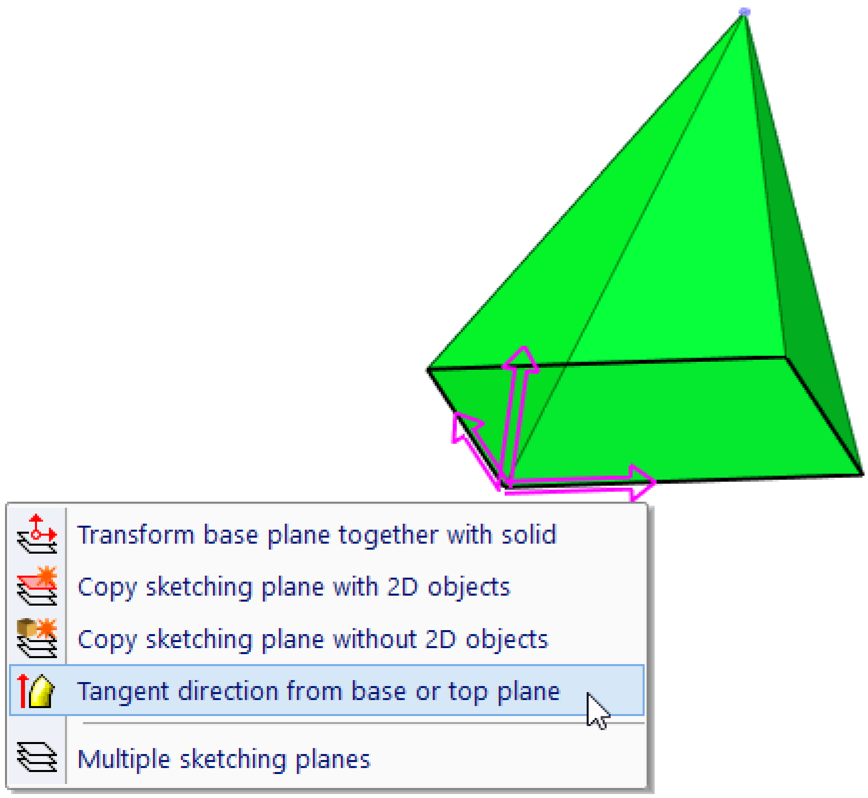
Tangent Direction of Lofting

Tangent direction can be selected for lofting from base plane or top plane. This causes the patches edges at base or top plane have normals perpendicular to plane normal. Tangent direction can be selected independently for both base and top plane.

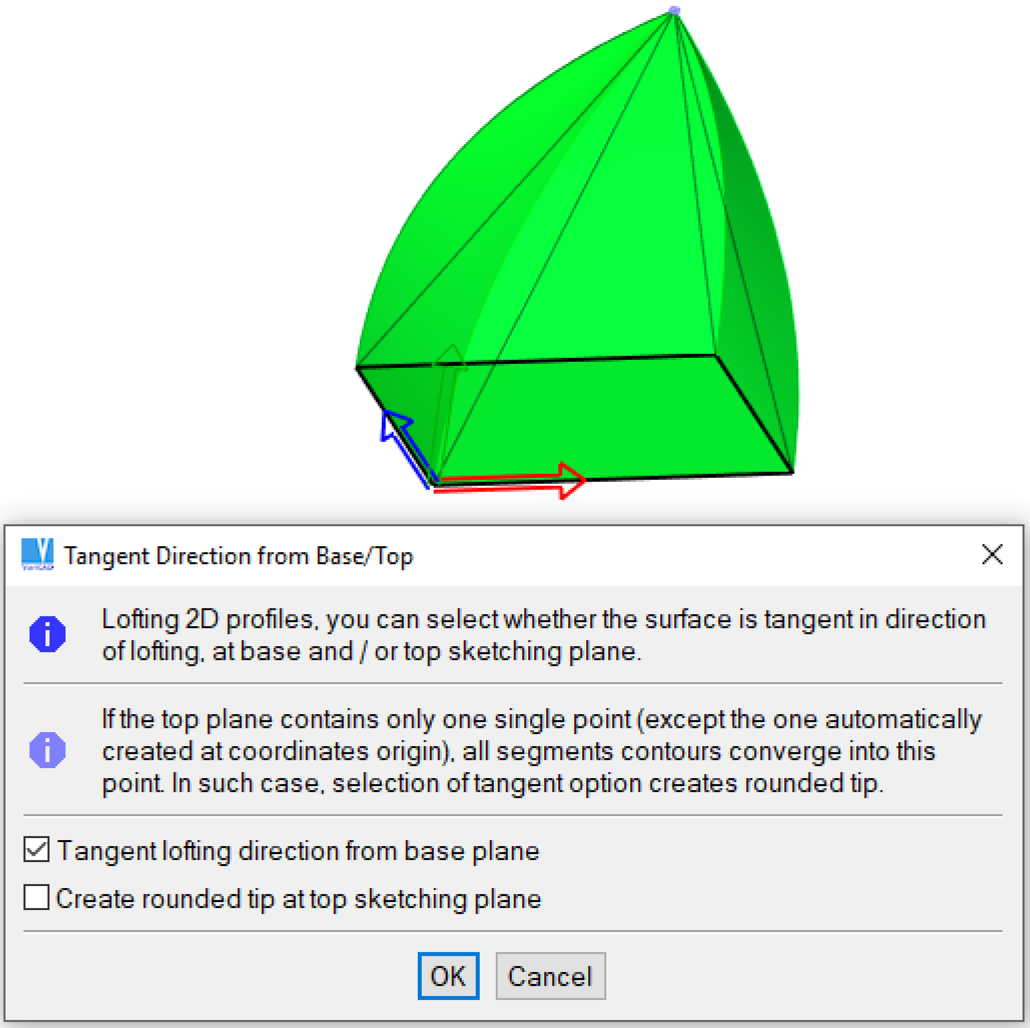
If the top plane contains only one point and patches converge into this point, then normal of patches edges at the point location is parallel to normal of the top plane. Following examples show definition of tangent direction. Here, a rectangle is connected with a single point at the top sketching plane.



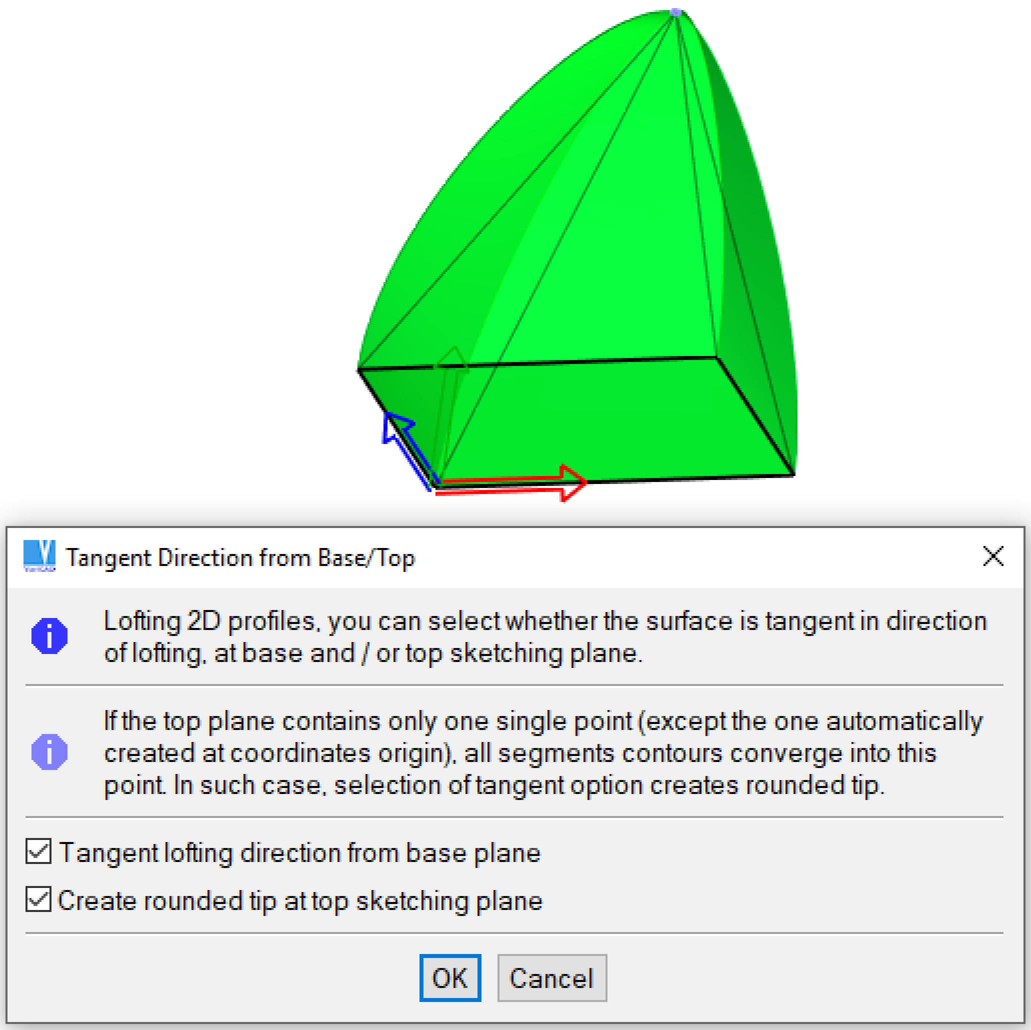
A rectangle lofted into a single point



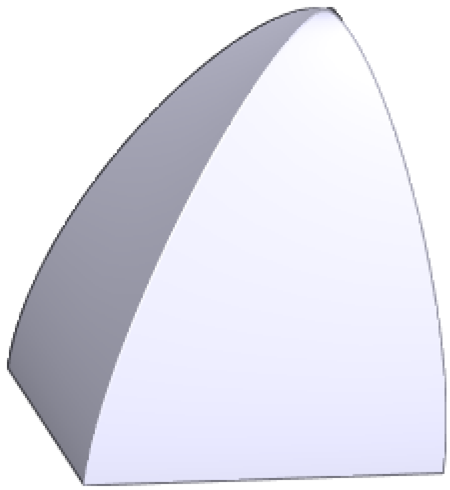
Selection of tangent direction of lofting



Tangent direction selected for base plane



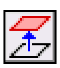
Tangent direction selected for both planes



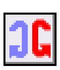
The final solid

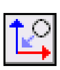
Features Related to Solid Creation Method

These features can be selected from pop-up menu, if you click XYZ axes of the active sketching plane, or if you right-click an empty location. These features are specific for selected solid creation method (or for method the edited solid is created according to).

 Connects the active plane with a plane selected at a solid. This is another method of sketching plane creation, used for multiple planes lofting. This feature is also available as a separate command.

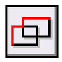
 Redefine rotation axis. Rotation axis is the X axis of solid axes and simultaneously, base sketching plane axes. If you need to change the solid profile relative to rotation axis, you can move 2D objects. Sometimes, it is necessary to redefine the axis in space. To do so, define first and second point of axis. These points are defined in active sketching plane.

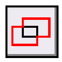
 Change direction of the rotation axis. The profile of rotation solids rotates around X axis counterclockwise. It means that if the X axis is directed toward you, the rotation is counterclockwise. This command reverses direction of axis and consequently, direction of profile rotation.

 Move all 2D objects to axes origin. Center of all 2D objects is moved into origin of solid axes. This feature is important for creation of helix. Here, the radius of helix is measured from axes origin. Consequently, it is necessary that the profile has its center equal to coordinates center.

Manual Selection of Profile's 2D Objects

Profiles are created by 2D lines, circles, arc segments or NURBS 2D curves. There are two methods of profile detection:

 Detect Profile Segments (or press E) - define the profile segment by segment

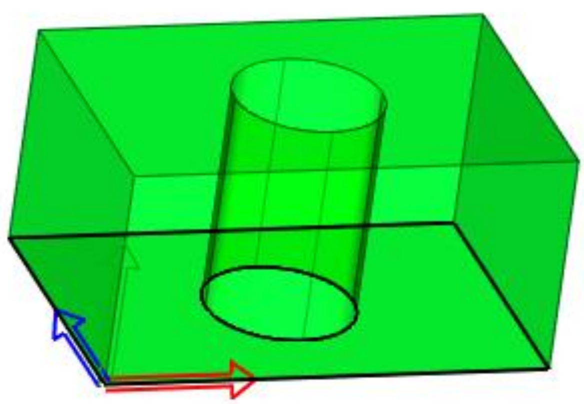
 Detect Profile (or press F) - select one segment and the entire chained profile is detected

Apart of automatic detection of profile's segments, you can select objects with standard methods of 2D selection – see *Selecting 2D Objects (page 45) (page 27)*. Press Enter or right-click to finish the profile definition.

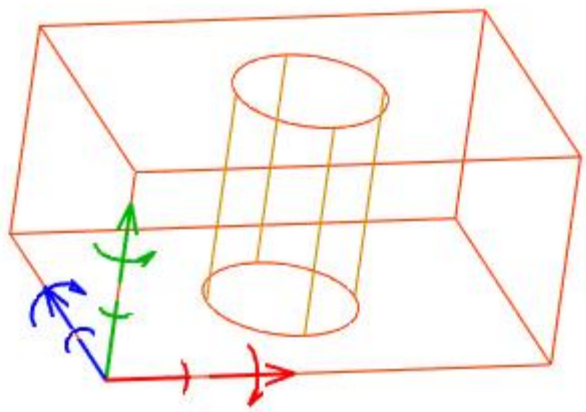
Profiles used for 3D solids must be continuous. If multiple profiles are used, they cannot intersect; one profile must completely encompass the other profiles. Profiles used in a Revolve operation cannot intersect the revolving axis. Lines, circles or circular arcs can be selected for all type of solid creations. If a profile contains gaps or intersections of segments, you can optionally highlight a location of the error.

Solid Insertion Point

Solid insertion point is equal to center of XYZ axes of base sketching plane. For solids created from a single profile, the base plane is the only sketching plane. During profile creation or editing, you can transform the sketching plane. Together with the plane, the entire solid is transformed, too. If profiles are created in sketching plane in 3D, the solid insertion point is defined automatically.



Axes of sketching plane



Transformation axes of the solid

Transformation axes are identical with axes of the base sketching plane, until you define their new location – an XYZ offset from the original location.

Solid Insertion Point for Profiles Selected in 2D Mode

Prior to selecting a 2D profile, it is necessary to enter values like extrusion height or rotation angle. Along with these parameters, you can also identify the solid insertion point and set the X axis direction. If you do not select an insertion point, the point at lower left point of profile will be used. If you do not set the X axis, the default 2D X axis will be used. The insertion point and X axis direction are used when inserting the solid into 3D space.

Run “CFG” command to select whether the insertion point and X axis direction is defined automatically, if a profile is selected in 2D mode.

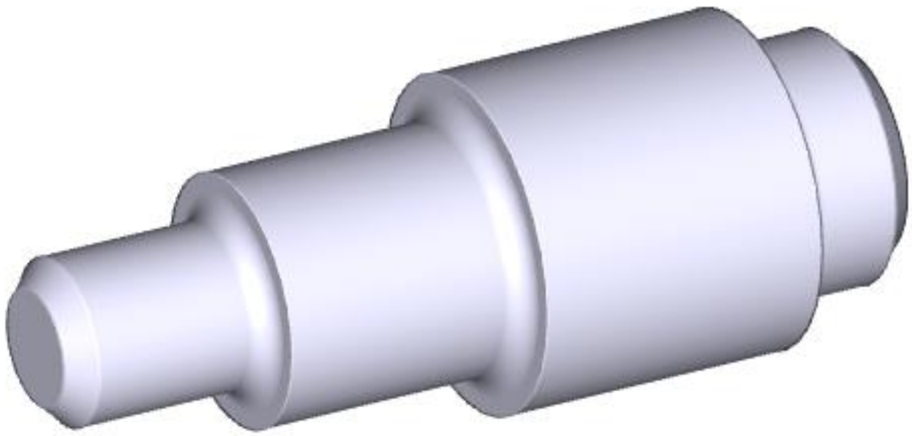
Rotating, Extruding, and Lofting Profiles

Rotated Solids

 **Full Rotation - RSO**

Rotates one open or closed profile 360 degrees around a rotation axis. The rotation axis is the X axis of the base sketching plane.

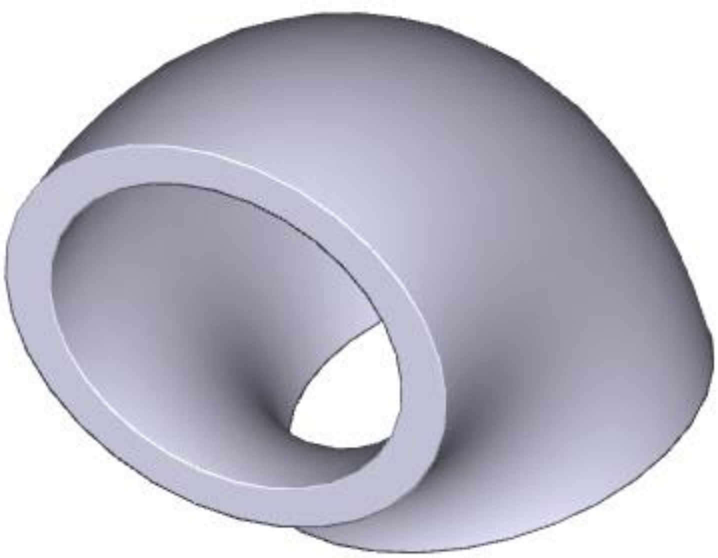
If a profile is selected in 2D mode, you must define the axis. For an open profile, the axis is connection of the profile endpoints. For a closed profile, you must define the axis by two points. If the insertion point is defined automatically (according to configuration), it is located at the first defined point of the revolving axis. When selecting closed profiles, multiple profiles are allowed inside one outer profile - this will create holes in the solid.



Example of Full rotation of 2D open profile

Partial Rotation - RSOP

Similar to Full Rotation, except that you can enter angle less than 360 degrees. During sketching, you can define the angle by cursor movement.



Example of partial rotation using closed profiles

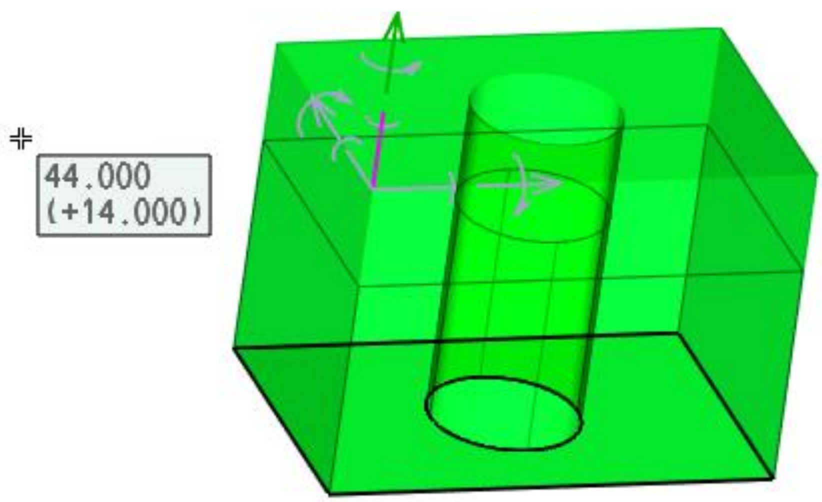
Extruded Solids

Extrude - ESO

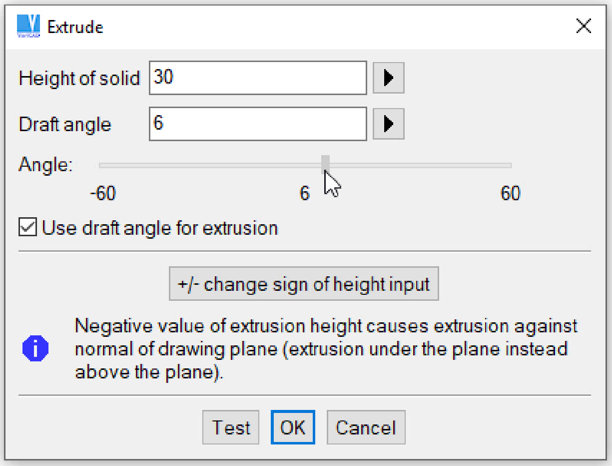
Command extrudes a profile to specified height, forming a solid. Profiles must be closed and multiple profiles are allowed inside the outer profile - this will create holes in the solid. Optionally, you can select a draft angle of extrusion.

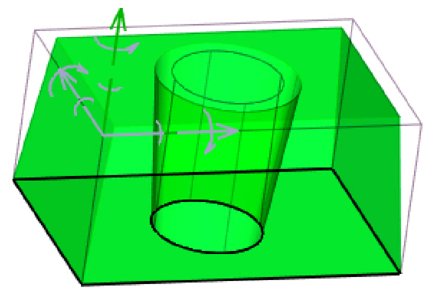
You can define extrusion height by cursor movement during sketching. The height can be both positive - the solid is created above the sketching plane, in direction of Z axis, or negative. Then, the solid is created under the sketching plane.

It is convenient to use cursor movement in increments. To define increment, or to turn this option on or off, right-click when you change extrusion height or rotation angle by cursor.



Definition of extrusion height by cursor movement.





Draft angle defined by cursor movement

Lofted Solids

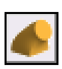
Lofting, Multiple Profiles Lofting - MPL

Lofted solids are created by connection of two or multiple profiles. Each profile is defined at its own sketching plane. Each profile must contain the same number of segments (like lines, arcs, circles or NURBS curves). Unlike extruded profiles, lofted profiles may contain only one inner (inserted) profile.

You can connect one rectangular profile (also with rounded corners) with one circular profile. The connection is done automatically. Contrary to this, if you want to connect multiple profiles in combination like multiple rectangles vs. circles, circles vs. ellipsis or closed NURBs cure, you must divide the circles or curves to the same number of segments.

Loft between Two Planes' Outlines – LB2P

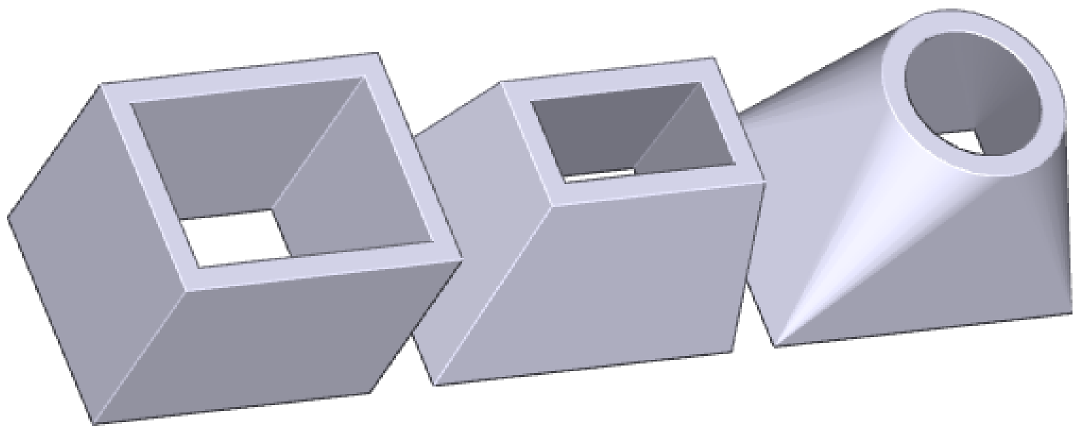
This type of lofting connects two planes selected at two different solids. After first plane selection, sketching mode is activated and a second plane is selected. Then, you can continue with sketching or you can finish solid creation, if planes can be correctly connected.



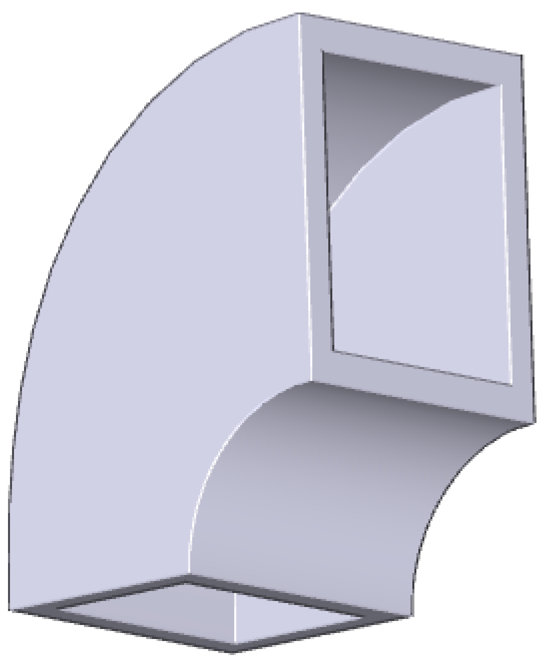
Rotate and Loft between Two Planes – MPLR

This command combines lofting between two planes and rotation around an axis. If a first plane is selected at a solid, the rotation axis is defined as X axis of sketching plane. You may need to redefine the rotation axis and redefine the rotation angle.

For more information, related to multiple sketching plane, see *Multiple Sketching Planes (page 181)*



Example of extrude, prismatic loft and loft rectangle to circle

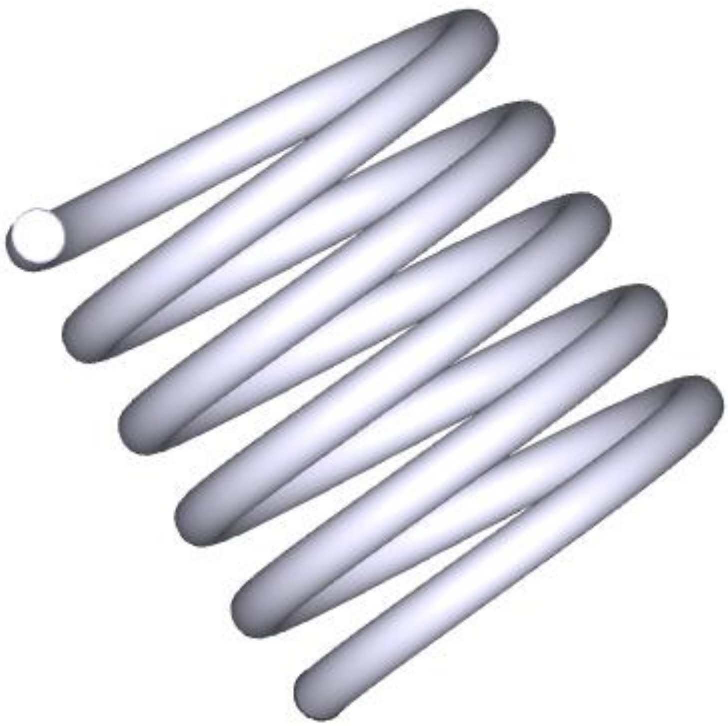


Example of solid coil, created by lofting and rotating profiles

Helical Surfaces

 **Helix - HLX**

Creates a solid by extruding and rotating a 2D closed profile. You can choose whether the profiles represent the radial, normal or axial section. For example, a spring uses a circle as the normal section. Using a negative rotation angle creates a helix with left ascent.



Example of spring helix defined by a normal section

Basic Solid Volumes

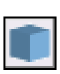
Basic solids are easy to create; you need to specify only basic dimensions. Editing involves changing the basic dimensions, or modifying any 2D profiles that are used. When entering solid dimensions, you have the option of copying any or all dimensions from another solid of the same type. Primarily, dimensions of solids are edited using spatial dimensioning. Optionally, you can select entering dimensions in panel. In panel, you can also change basic shape, like chamfering cylinders etc.

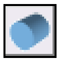
Corresponding images displayed in panels are changed according to currently set options. For instance, a cylinder options allow you to select chamfers or radii. Changing these options, images are displayed with/without chamfers etc.

When a basic volume is inserted and you select one step back (Ctrl + Z, or undo mouse button), the basic volume is displayed together with spatial dimensions and you can change dimensions by editing these dimensions.

Cylinders, Cones, Boxes, Pyramids, Pipes, Spheres

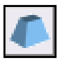
The basic solids are as follows:

 **Box - PRS**

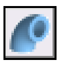
 **Cylinder - CYL**

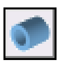
(with option of rounding or chamfering either end)

 **Cone - CON**

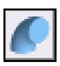
 **Pyramid - TPY**

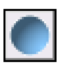
(uses rectangular base)

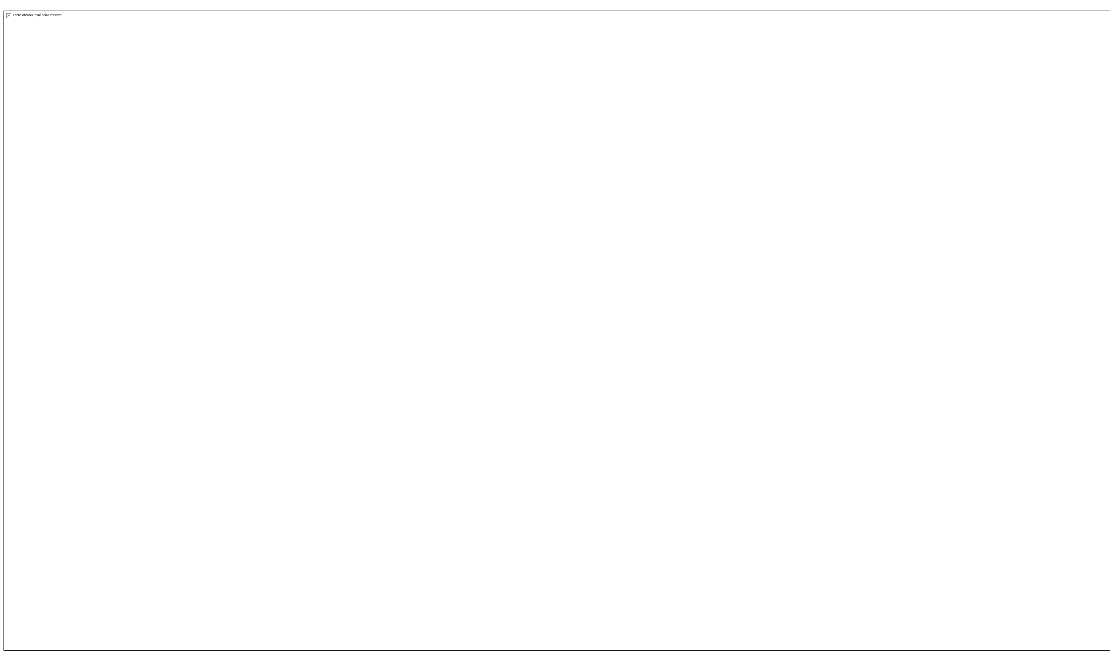
 **Pipe Elbow - PEL**

 **Pipe - PIP**

 **Cone Pipe - CPI**

 **Solid Elbow - ELW**

 **Sphere - SPH**



Example of definition of cylinder’s dimensions and shape

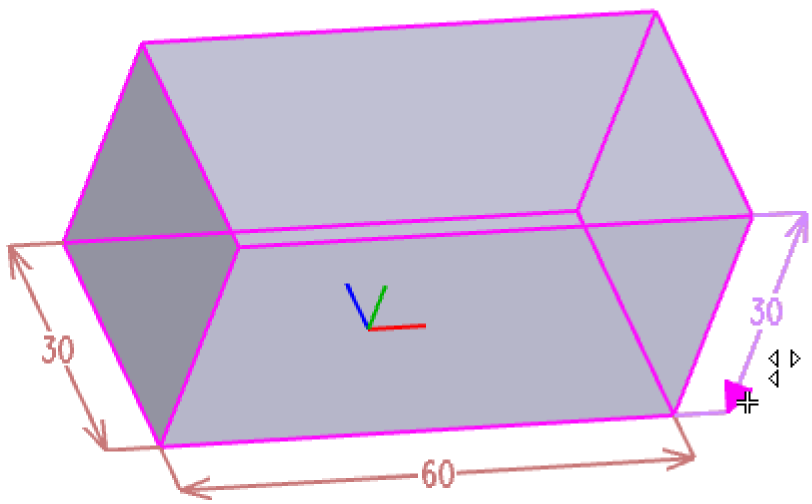
Editing of Spatial Dimensions of Basic Solids

If you create a new solid, first define dimensions in a table. Then, a new solid is located. You can right-click it and select redefinition of dimensions, by spatial dimensioning or in a table.

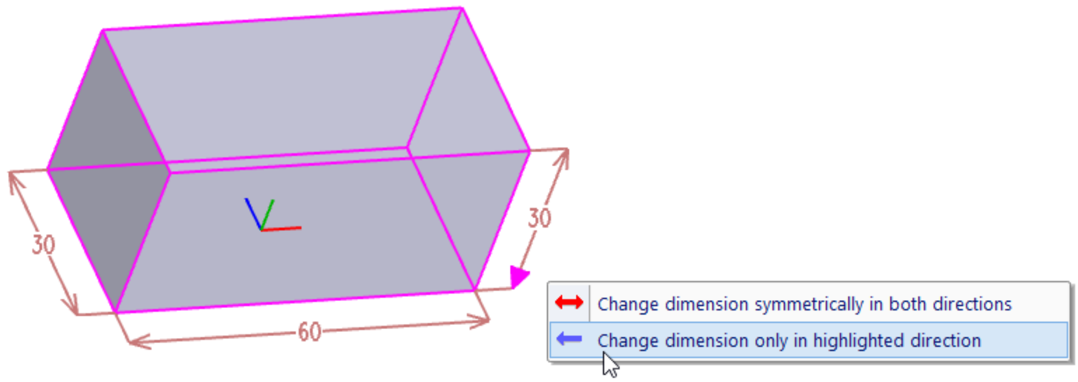
Spatial dimensions are used as a first option, if you select a solid for editing. To change the selected dimension, click it and rewrite value in a dialog window.

You may click a witness line or an arrow of the dimension. Then, you may drag the dimension by cursor movement. Together with it, the shape of entire solid is changed. Some situations even permit to change the dimension in direction of the selected arrow, or symmetrically.

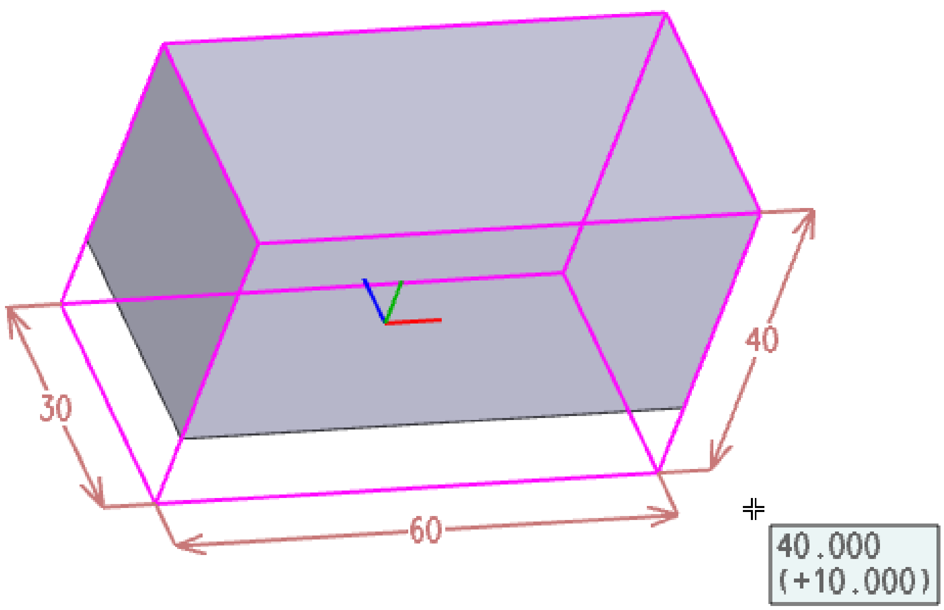
It is convenient to use cursor movement in increments. To define increment, or to turn this option on or off, right-click when you modify dimension by cursor movement.



Selecting a dimension to be edited. Here, multiple options are allowed.



Selecting direction of dimension change. This option is a default option and can be selected by single click at the arrow.



Changing the selected dimension by cursor movement.

Editing Solids

Solids can be edited by:

- Changing the location of their components (see *Inserting and Transforming Solids (page 229)*)
- Deleting their components (see *Deleting Solids (page 224)*)
- Adding or removing volume (see *Boolean Operations (page 208)*)

- Changing basic solid shapes (see *Editing Basic Solids (page 224)*)
- Changing visibility, color, shading, or method of detection

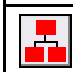
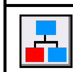
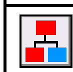
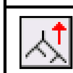
Selecting Solids

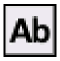
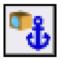
Edit functions, like other functions, require you to select objects. You can select objects one at a time, or use methods for selecting groups of objects. Many functions allow you to select an entire solid or only its components. For example, you can delete entire mechanical parts or only a hole or fillet.

Objects are detected for selection when the cursor passes over them. If wire-framed displaying is used or if a particular solid is displayed as wire-framed, object is detected if the cursor passes over an edge or wire. The density of wires for automatic detection can be defined in *3D Colors and Wires of Solids (page 157)*. Selected objects are stored in a temporary work set, and are highlighted and displayed in wireframe. You can add or remove objects from this set. Pressing Enter or right-clicking finishes the selection and processes the objects in the set.

In some functions, only certain types of objects can be selected. In such situation, you can't detect all objects. For instance, you can't edit basic solid's shape, if the solid is imported from STEP. Such objects can't be detected in solids selection for basic shape editing.

During object selection, a temporary toolbar appears. It contains the following selection options:

Icon	Hotkey	Selection
	T	Entire Solid - solid and its components will be selected.
	S	Single Elements - only the basic solid will be selected. This is useful for selecting a basic solid added to another solid, such as a rib or fillet.
	B	Branches - components of solids (Boolean branches) will be selected. Components are detected as the cursor passes over the solid, according to how they are connected to the root solid.
	N/A	Selecting objects from a list of the Boolean tree.
	A	All Solids - selects all visible solids in the file.
	R	Completely Inside - selects objects completely inside the selection window.
	I	Inside - selects objects completely or partially inside the selection window.
	U	Outside - selects objects completely or partially outside the selection window.
	O	Completely Outside - selects objects completely outside the selection window.
	P	Previously Selected - once again selects objects selected in previous action

	G	3D Group - selects a group of 3D objects. The list of groups is displayed and you can select the group.
	E	Names or Attributes - select objects according to their names or attributes
	N/A	Selects a group of constrained solid elements.
	N/A	Selects a group of constrained solids.
	N/A	Selects objects from a constrain scheme.
	N/A	Undo Selection – cancels last selection step
	X	Select/Deselect - switches between adding and deleting objects from the selected set.

To select single solid elements (parts of Boolean tree, like holes, fillets…) while selecting objects between commands, press and hold Ctrl key and move cursor over objects.

3D Selection Settings

3D selection can be set in command “CFG”, in section 3D. You can modify:

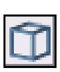
- Highlighting of wires of non-planar patches, if the cursor passes over them. Edges are always highlighted.
- Size of the cursor aperture.
- If the wires of the selected solids are displayed always up. Otherwise they can be displayed as partially hidden under other solids, if such solids are above them.
- Configure whether and how the selection window (marquee) is started, if you click the left mouse button and no 3D object is under the cursor.

Visibility of 3D Objects

For large files, you do not always need to have all items visible. Blanking objects can improve legibility of a 3D assembly, if space contains some auxiliary objects. When loading files, you have the option to be warned when objects are blanked. Blanked objects are not processed in any operations except unblank.

 **Blank - BL3, Ctrl + B**

Makes selected objects invisible.

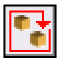
 **Unblank - UB3, Ctrl + U**

Unblanks objects that were blanked. To make objects visible, you can select several methods:

- Unblank selected objects. Invisible objects are temporary made visible, as wire-framed in distinctive color. You can select objects to be visible.

- Unblank objects by 3D group.
- Unblank all invisible objects.

If there are some invisible objects in 3D space and if you right-click an empty area, command Unblank is a part of the pop-up menu.



3D Groups Management – 3GR, Ctrl + F1

Among other possibilities, this function also changes visibility of 3D groups. See *Groups of Solids (page 243)* for details on solid groups.

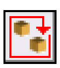
Shading and Colors of Individual Solids

You can shade all solids or only selected solids. If the display is switched to shaded, all objects will be shaded except those objects set to be wire-framed. Wires drawn on edges or surfaces are hidden behind shaded objects. Turning off the shading of selected objects allows you to see inner parts of an assembly.

Instead of wire-framed solids, you can use transparent solids. Transparent solids are detected the same way as wire-framed solids – you must move cursor over their edges.

Shade/Wireframe Selected Solids - SHC

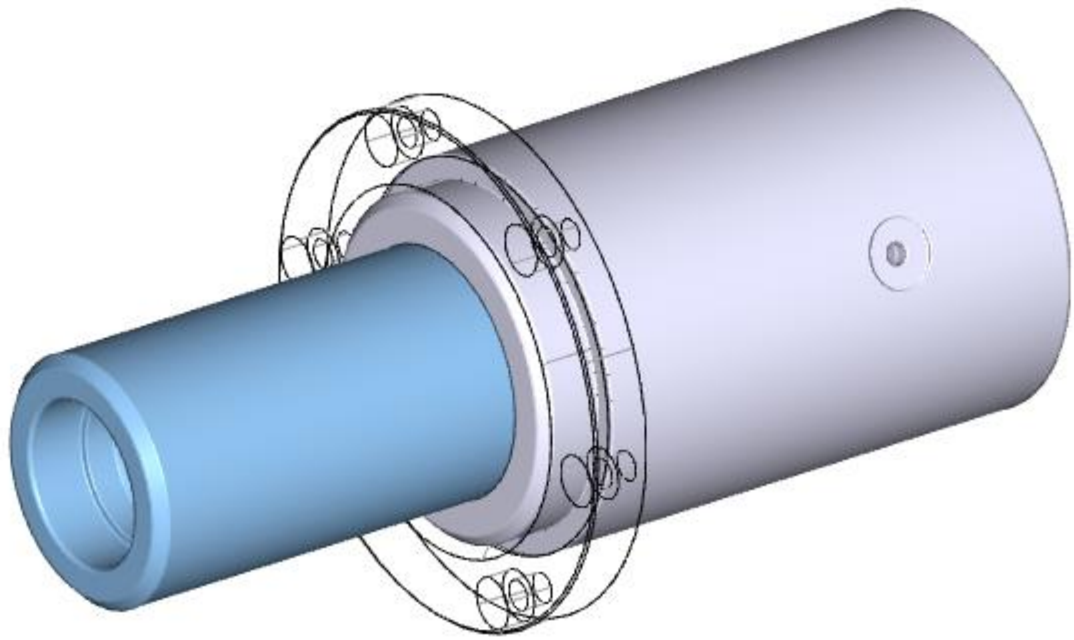
First select, if the selected solids will be shaded, transparent or displayed as wire-framed. Then, select solids to change their display method.



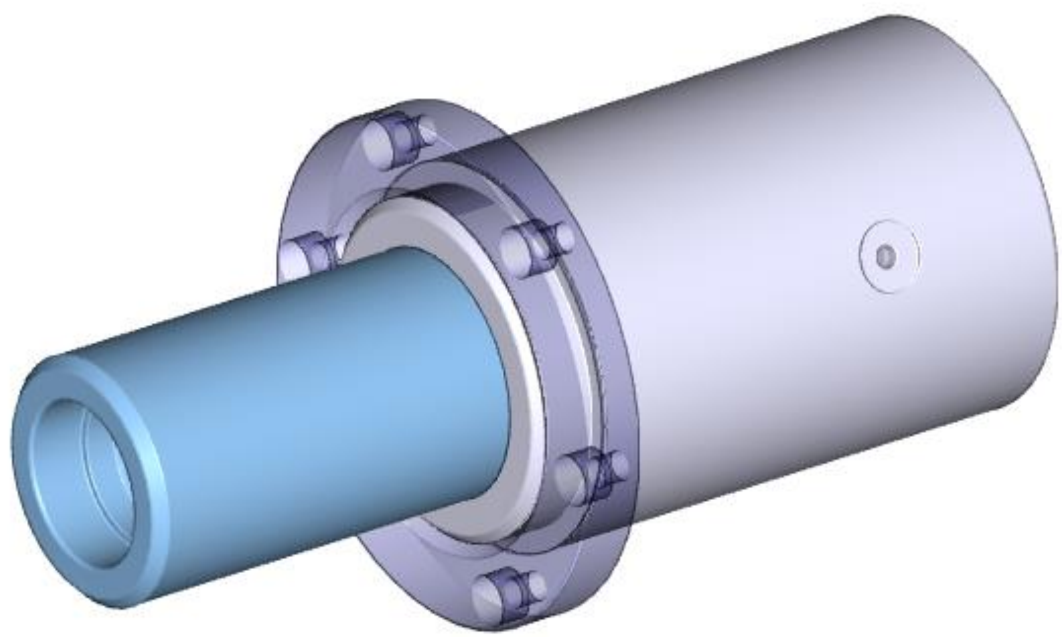
3D Groups Management – 3GR, Ctrl + F1

Among other possibilities, this function also changes shading or transparency of solid groups. See *Groups of Solids (page 243)* for details on solid groups.

Note that transparent solids are not supported if you work with old OpenGL 1.1.



Example of a wire-framed solid together with shaded solids



Example of a transparent solid together with shaded solids

 **Change Color - CS3**

Changes the color of selected solids or their components. When solids are set to all be created in the same color, changing a component color will not have any effect because components use the color of the root solid. See *Colors and Wires of Solids (page 157)*.

Boolean Operations - Adding and Cutting Solids

Adding solids together and using one solid to cut volume from another are called Boolean operation. These operations can be performed when solids have volume in common (overlap) or have at least one common (or partially common) surface. In addition to Boolean operations described in this section, there are other predefined Boolean operations including drilling holes, milling, creating grooves, filleting and chamfering.

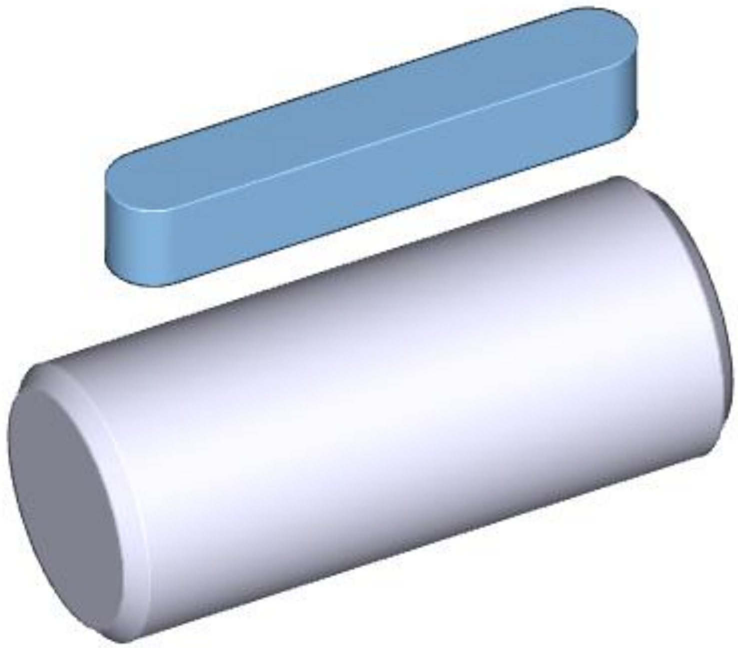
Boolean Operations

 **Add Solid – ADD, Ctrl + A**

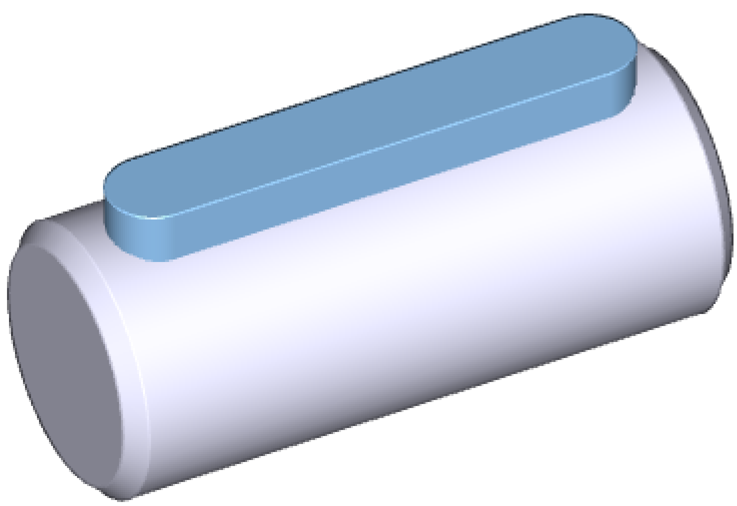
Combines two solids into one object. First select the solid to be added, and then select the solid to be added to. Although the final result is the same, the selection order can be important. If the solids have properties such as certain attributes or group membership, the final solid will have the properties of the second selected solid - the root solid.

 **Cut, Delete Cutting Solid – CUT, Ctrl + W**

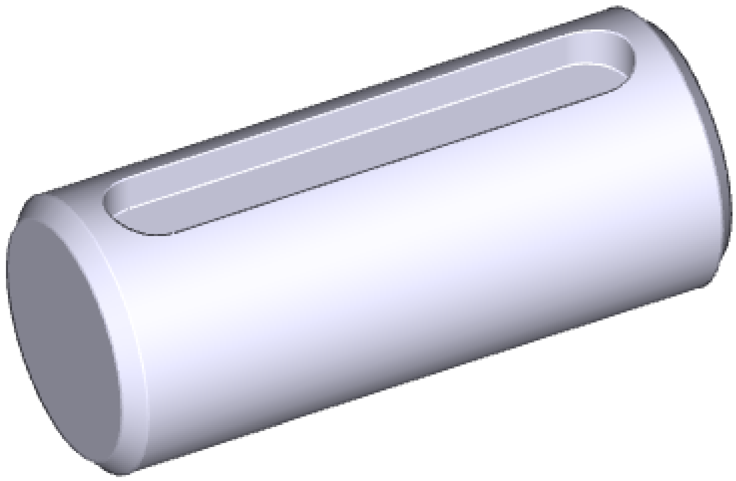
Uses one solid as a cutting tool to remove volume from another solid. The cutting solid is then deleted. For example, to create a conical hole, use a cone as a cutting tool to remove volume from a cube.



Example of Cut, Delete Cutting Solid. The spline is the cutting tool; the pin is the root solid.



The spline is embedded into the pin.



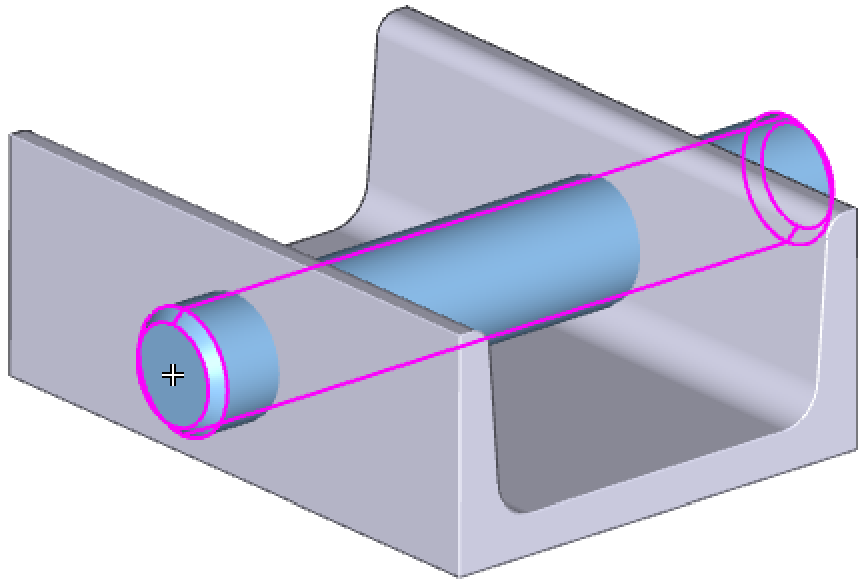
The resulting cut solid. The spline is deleted.

 **Cut, Keep Cutting Solid - CUTS**

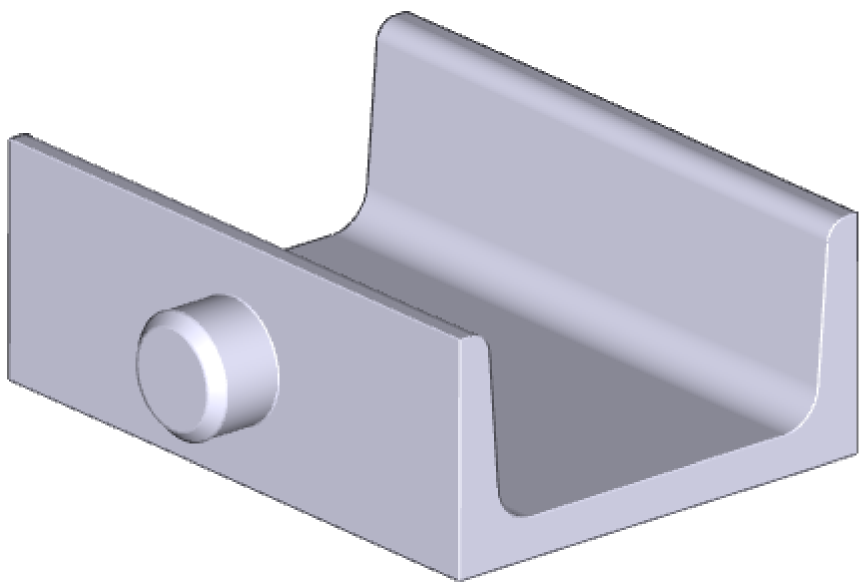
Similar to Cut, Delete Cutting Solid, except that the cutting solid is not deleted.

 **Selective Add - ADDPC**

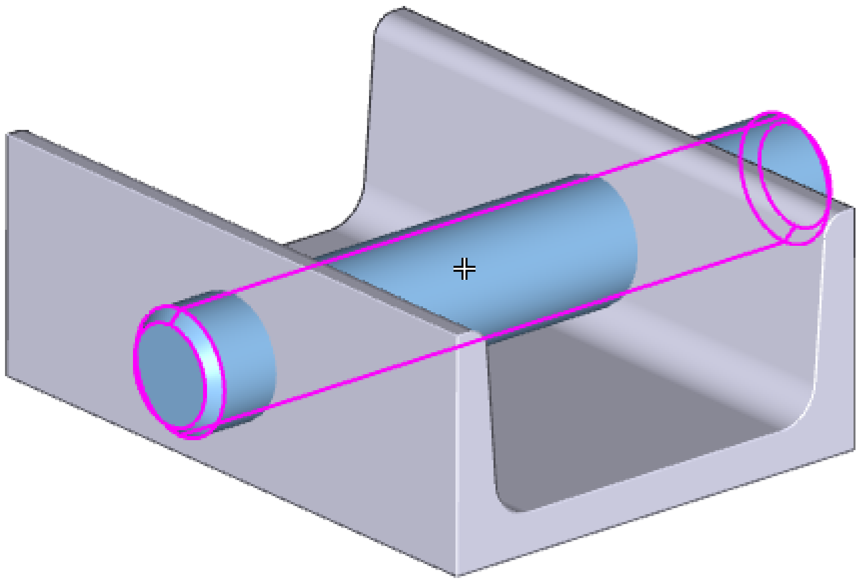
Similar to Add, used to add solids that overlap or extend past the root solid. Select the added solid at the section you want to remain; other sections will be trimmed.



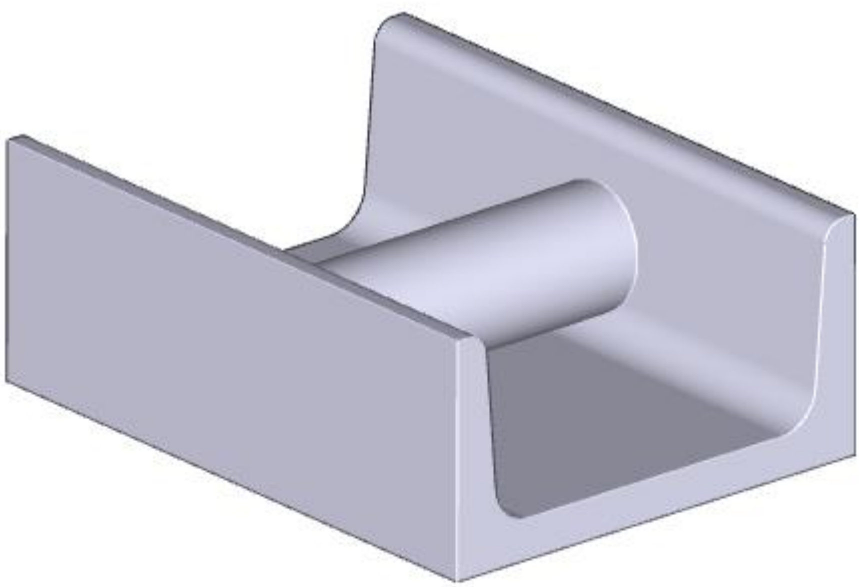
Example of Selective Add. The cylindrical pin is to be added to the U-iron. The pin is selected where indicated.



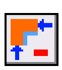
The resulting added solid. Only the section selected was added, the rest of the solid was trimmed.



Using the same initial solids, the pin is selected at a different location.



The resulting added solid. The middle section only was added.

 **Selective Cut, Delete Cutting Solid - CUTPS**

Similar to Cut, Delete Cutting Solid. For the cut solid, only the section you select will remain; other sections will be deleted. The cutting solid is also deleted.

 **Selective Cut, Keep Cutting Solid - CPSS**

Similar to Selective Cut, Delete Cutting Solid, but the cutting solid is not deleted.

 **Solid Intersection - SIN**

The result of the solid intersection is a volume common for both selected solids.

 **Add Solid, Perform No Intersection - NADD**

This command adds a solid into solid tree structure, but performs no Boolean operation. This way, you can create solids from elements which do not intersect or touch each other, or which touch each other only in one point or line. Typical example is creation of a ball bearing. You can add balls to inner ring and then outer ring to the rest.

If solids containing multiple closed volumes (lumps) are exported into STEP, they are divided into multiple single solids.

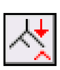
Mechanical parts inserted from libraries (like screws, pins, rings and others) allow you to modify a counterpart. See *Modifying Counter-parts, Drilling Holes for Screws (page 129) (page 120)* in section Libraries of Mechanical Parts.

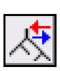
Boolean Tree Structure Editing

 **Boolean Tree Structure Editing – TREE**

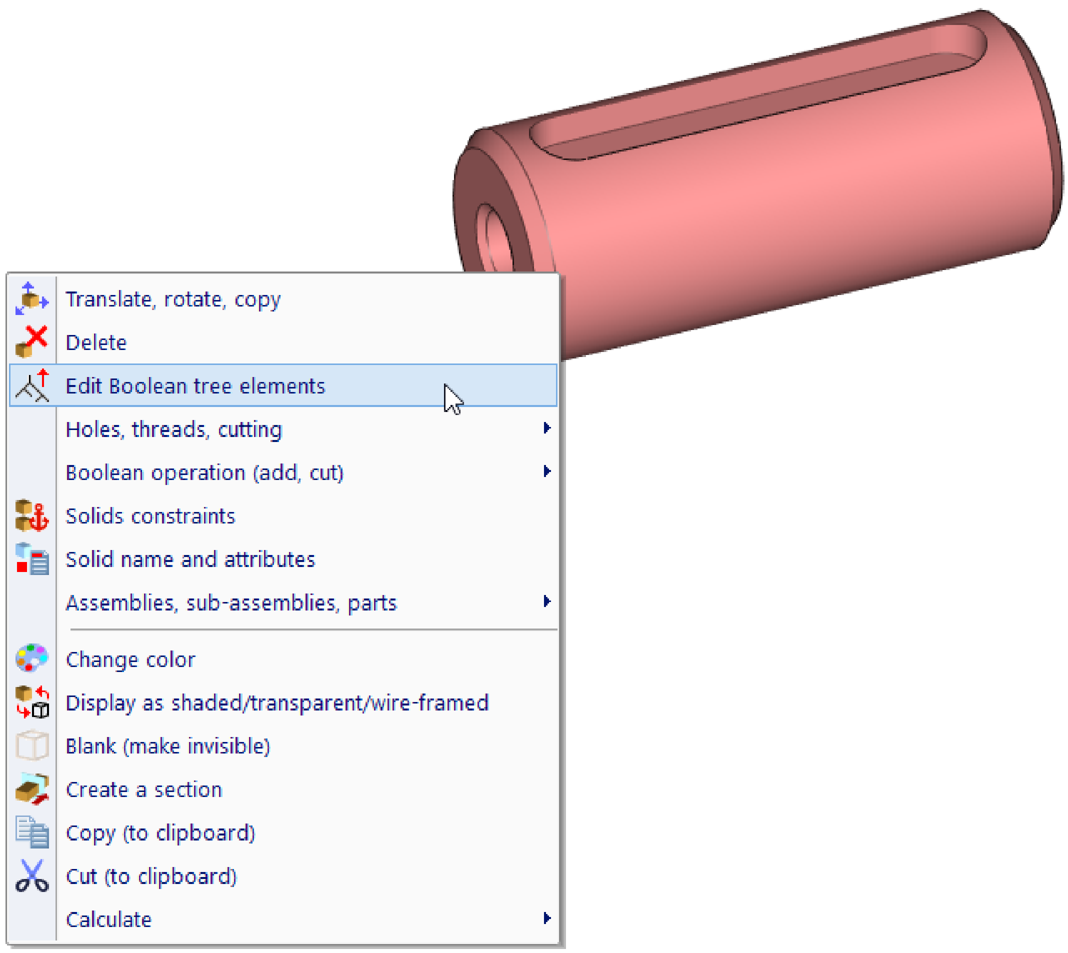
You can change the Boolean structure of a selected solid. It is possible to change an order of Boolean tree branches or to copy or move out Boolean tree branches. Each branch can be selected in Boolean tree scheme or can be detected interactively in 3D space.

- Boolean tree branch is a substructure of Boolean tree. It may be, for instance, a sub-solid previously added to another solid or removed from a solid (a tool cut from a solid). A branch may contain, of course, only one solid element (like a drill in case of drilling a hole).

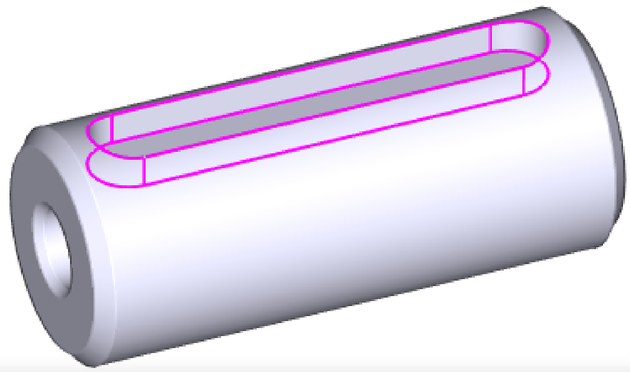
 **Copy Tree Branches** Select if the Boolean tree branch will be copied out of the solid or moved out of the solid. Then select a branch and locate it into 3D space. Location works with the same tools as insertion of new solids or as transformation of solids - see *Transforming and Copying Solids (page 229)*. Lone blending cannot be selected.

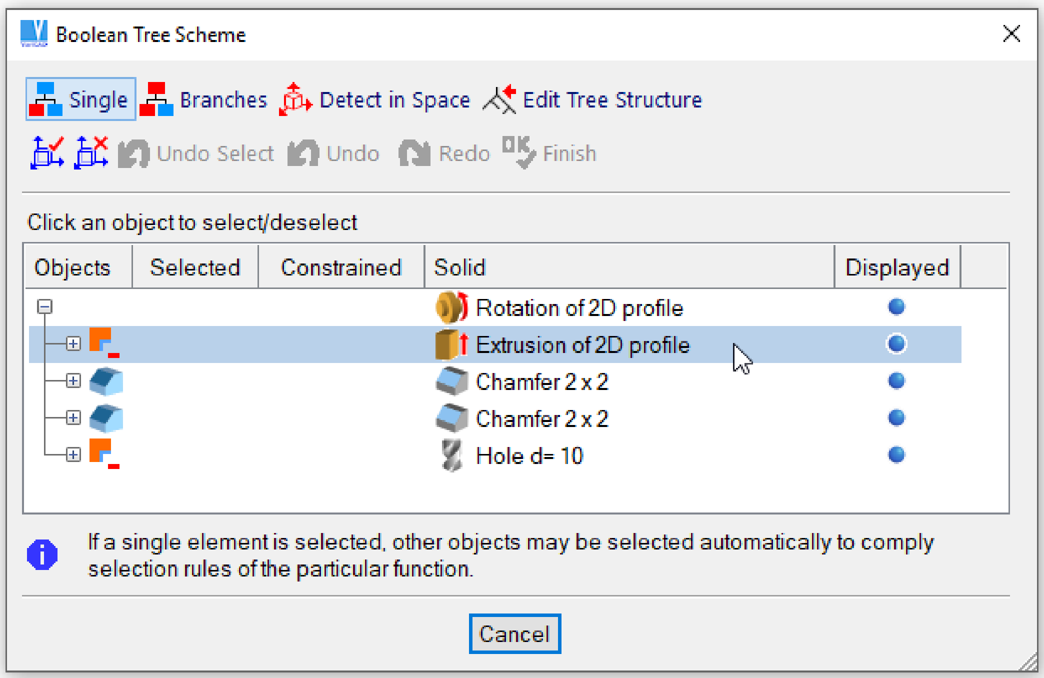
 **Edit Tree Structure** Select a Boolean tree branch and then a branch the previously selected object will be moved after (or optionally, in front of). You can move at the same level or to a higher level. Each time the Boolean tree is rebuild, the order of performed operations is from up to down as displayed in scheme. If you change the order of operations, you can obtain a different shape of the solid. Another benefit may be that you can easily remove or copy out branches. Also, filleting or chamfering can create different results.

The Boolean tree scheme window is displayed at second monitor, if VariCAD works at two monitors. Following examples show possibilities of editing solids, using Boolean tree scheme.

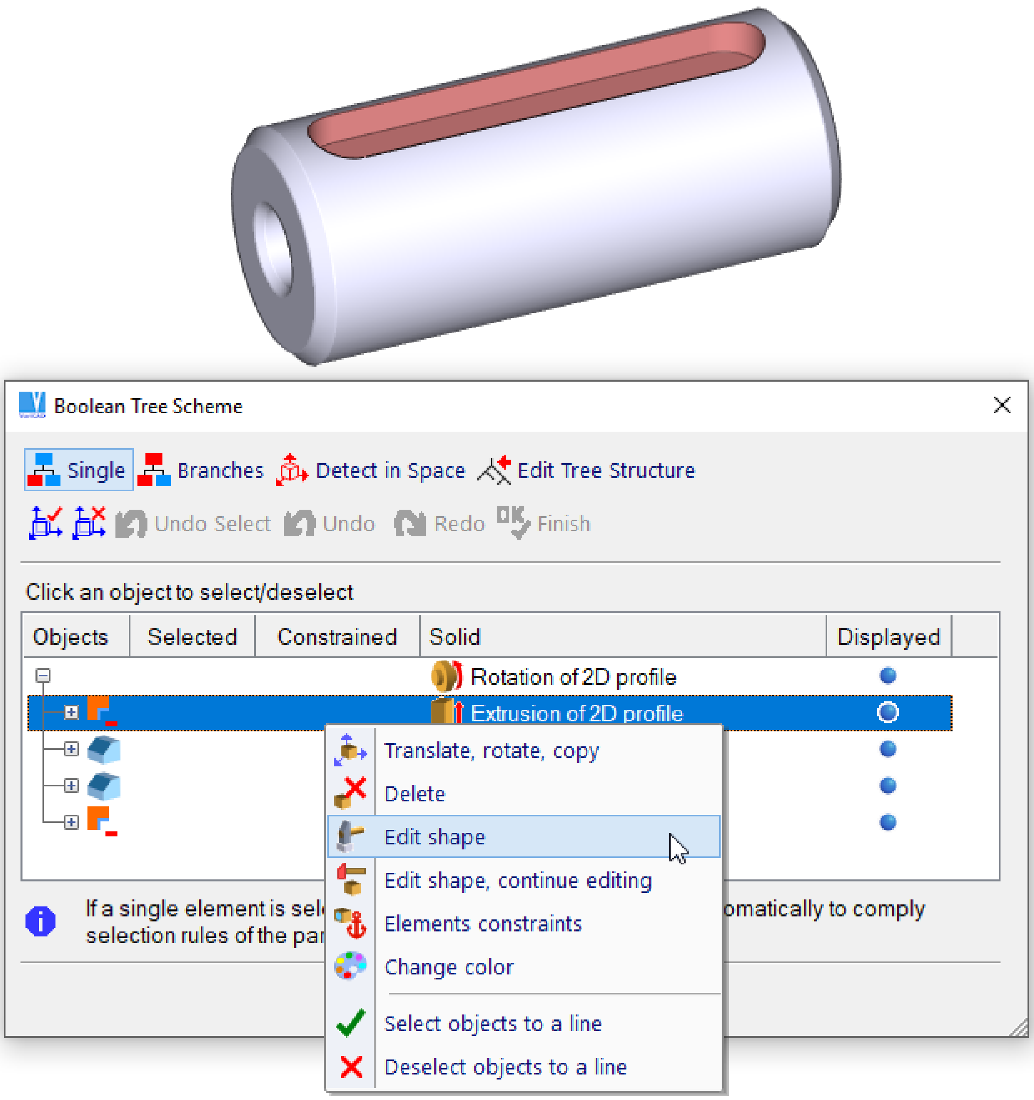


Right click a solid, then select “Edit Boolean tree elements”.

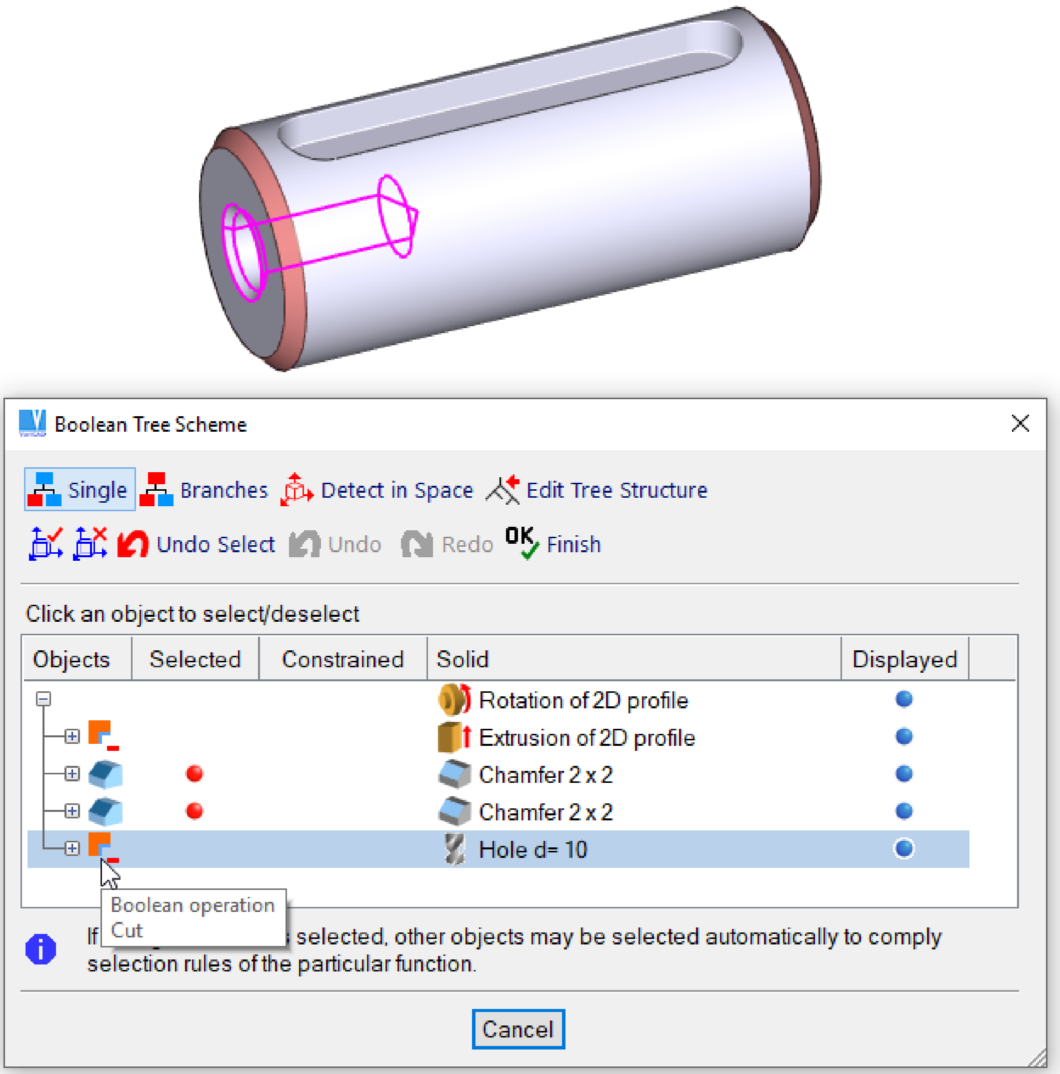




When you move cursor over the elements in the schema, corresponding 3D objects are highlighted.



Right-click a line and then select a command from menu



A group of elements can be selected by left-click, then processed after right-click. This example also shows tool-tip under cursor hovering above an icon.

Common Boolean Operations

Hole, Mill, and Groove all use a cutting tool to remove volume. The dimensions of the cutting tools are specified similarly as for basic 3D elements; you can also copy dimensions from another object. For example, in the Hole function, you can copy the dimensions of an existing hole.

You can select these cutting commands from tool-bar or also, if you right-click a solid and then, from pop-up menu. After the command is selected, you have to define dimensions of a cutting tool in a table. Then, the tool is inserted into space. You can redefine dimensions the same way as if you create a new solid – see *Editing of Spatial Dimensions of Basic Solids (page 202)*.

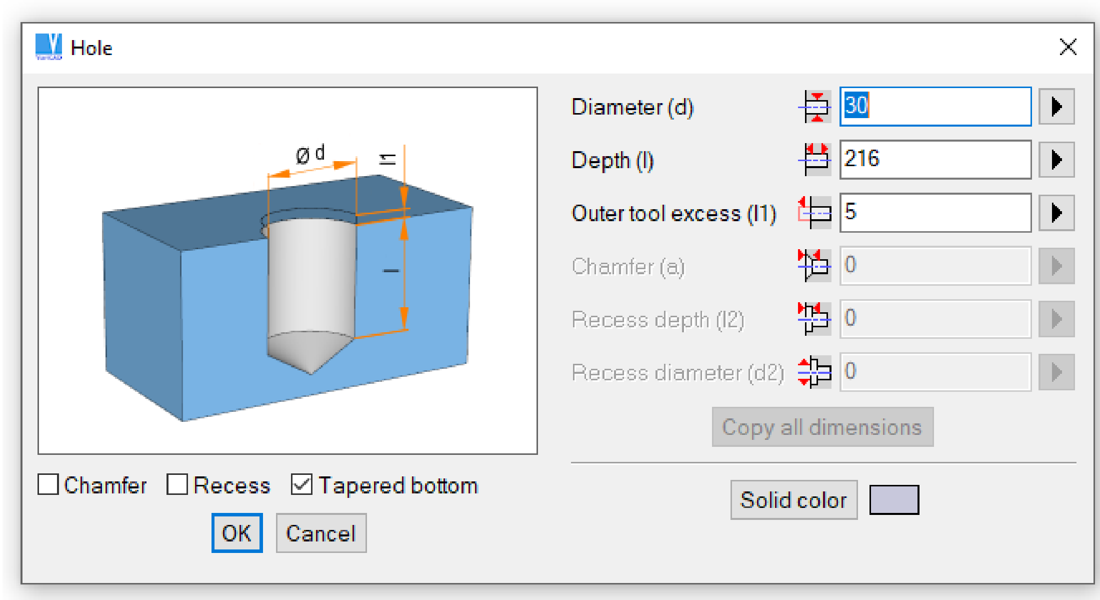
The tool is pre-inserted according to a solid's surface – it means where you clicked the solid to be cut (or to be drilled, for instance), there is the tool inserted. Also, the tool is properly oriented according to normal of the surface. Next, define the final tool location.

If you drill a hole and if you click an existing hole drilled into a solid, the drilling tool is positioned exactly as the tool which created the detected hole.

Holes, Grooves, Cutting by Planes

Hole - HOL

Drill holes into objects. You can optionally create recesses, or use a chamfered or tapered bottom.



Definition of a new hole

Groove - GRV

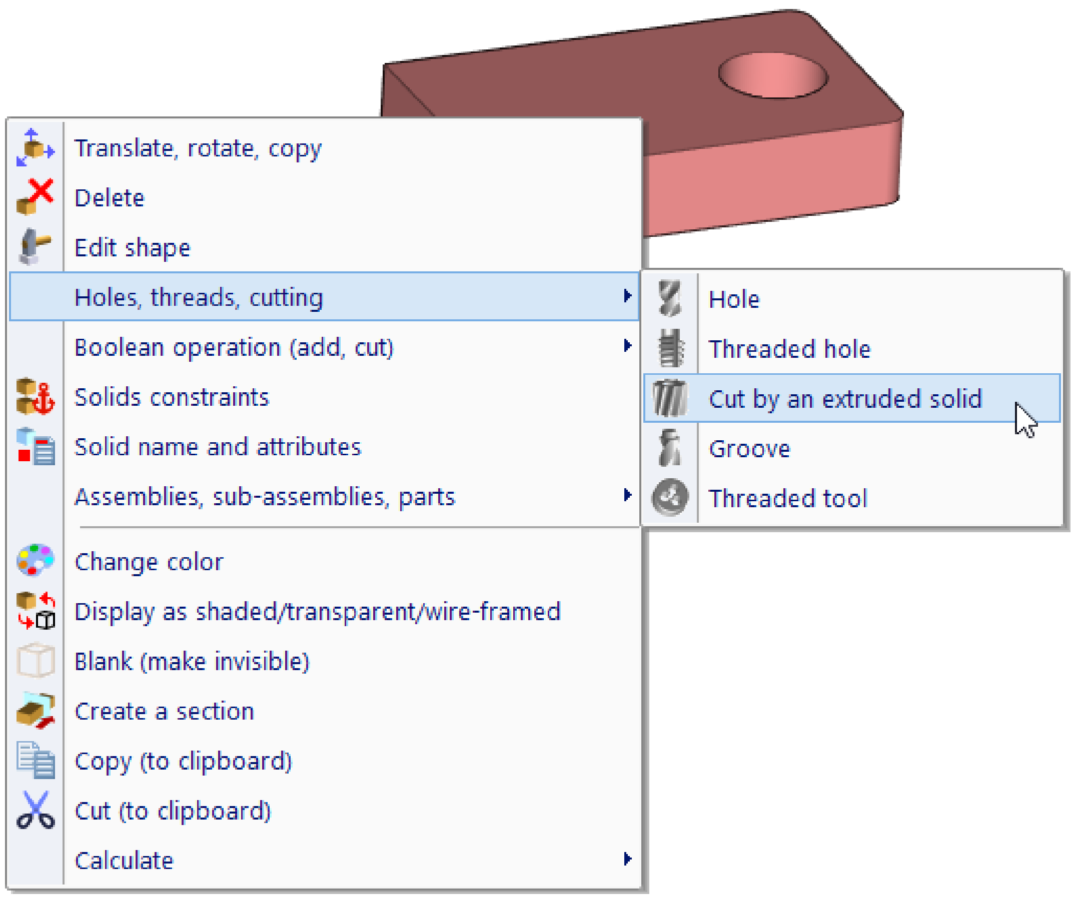
Removes volume by using a spline tool.

Cut by a Box (Milling) - MIL

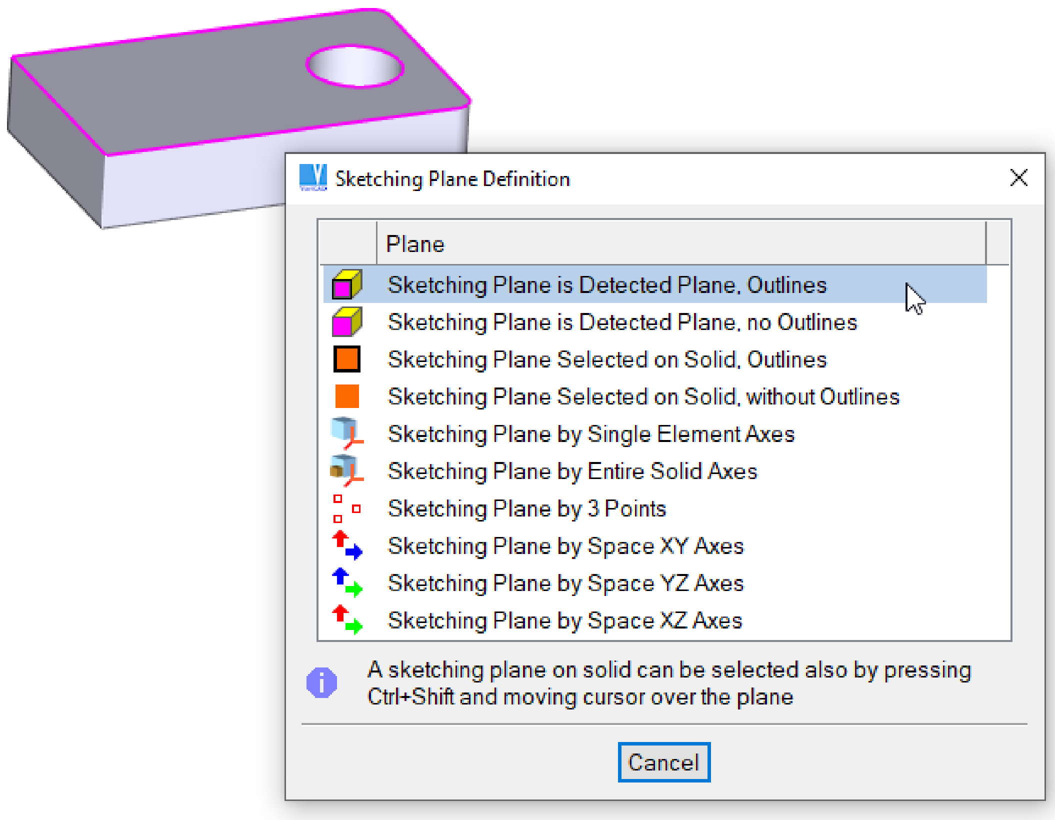
Removes volume by using a box as the cutting tool. If the box dimensions are large enough, you can, in effect, cut by plane. We recommend to use rather command "MILX", see below.

Cut by an Extruded Solid - MILX

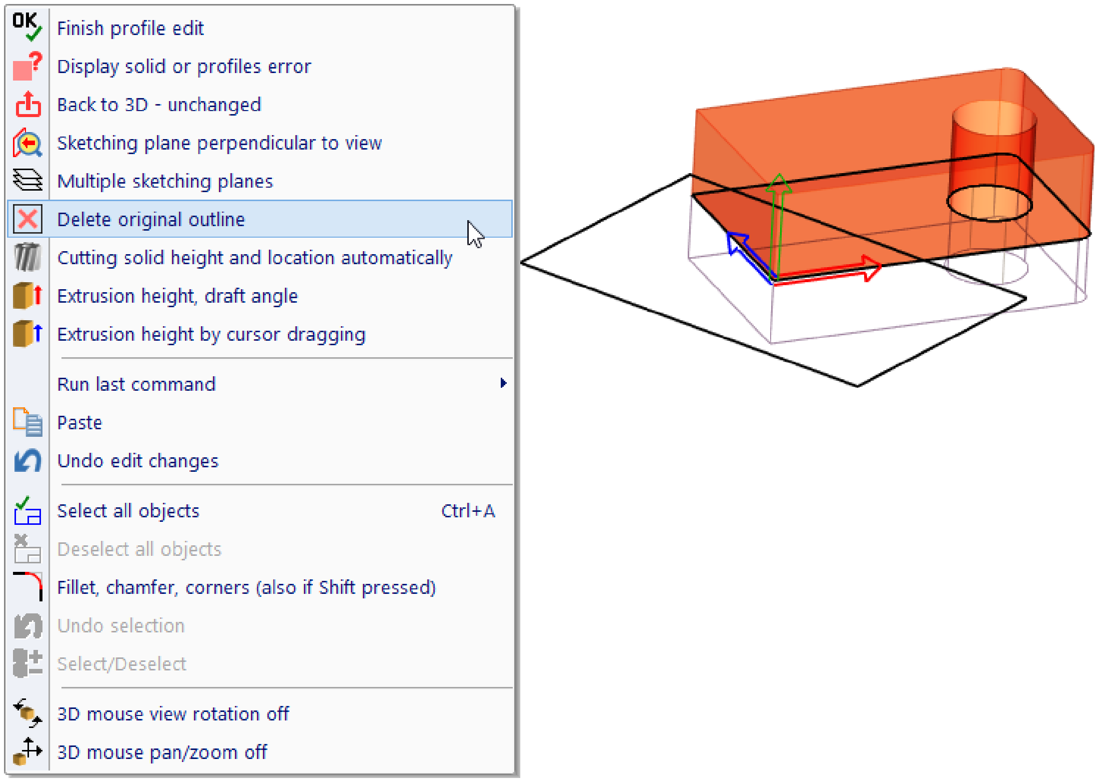
Select a sketching plane and create a contour of the cutting tool. After confirmation of contour shape and location, the tool is created and automatically cut from the solid.



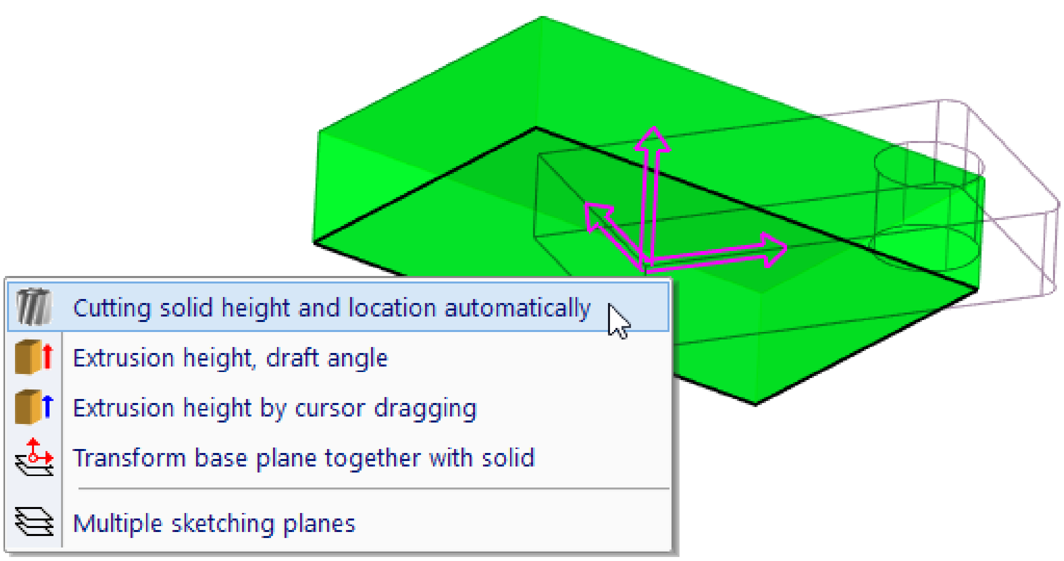
After right-click the upper plane, a pop-up appears. Select cutting by an extruded solid.



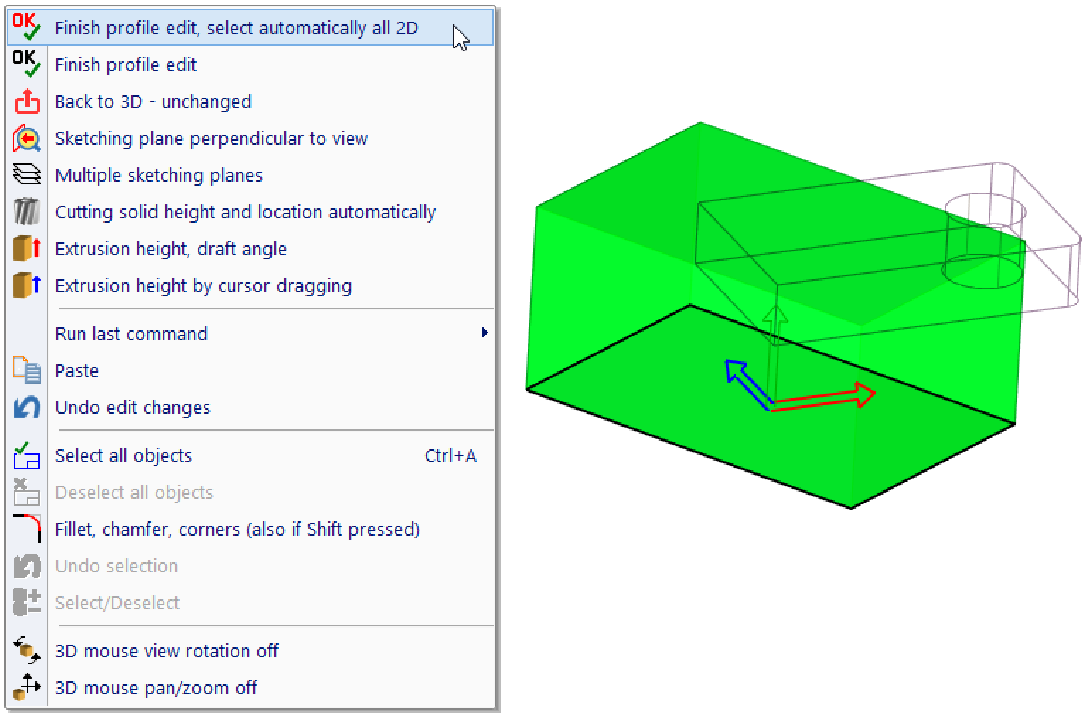
Select a sketching plane.



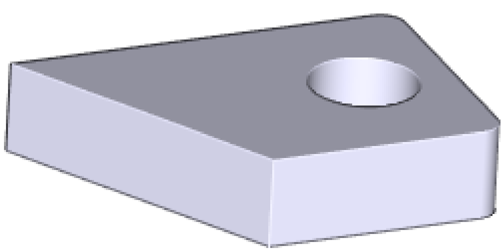
Create a contour of the cutting tool. Original outline still remains, remove it by option as displayed here.



After removing extra lines, you can define height and location of cutting tool automatically. Click the axes, or right-click an empty location and select automatic definition from menu.



The tool created by sketching, before final confirmation.



After confirmation, the tool is located and cut from the solid at one step.

Other commands cutting material are those creating threads – see *Threads in 3D (page 275)*

If you define hole or groove dimensions, images in panel are changed according to selected option. Also, you can define dimensions by spatial dimensioning similarly as for basic volumes – see *Basic Solid Volumes (page 200)*.

Resolving (Exploding) Solids

Explode Boolean Tree - TRX

Resolves a selected solid into its basic parts. Solids used for adding or cutting are changed back to their original objects. It is not recommended to use this function, if you only want to remove parts of Boolean tree (like holes or fillets) or if you want to change position of parts of Boolean tree relatively to other parts.

3D Filleting and Chamfering

When using these blending functions, first select the edges to be filleted (rounded) or chamfered. Then specify the fillet radius or chamfer distance. Rounded or chamfered edges are kept if the solid changes during editing. The Fillet function handles fillet overhangs and undercuts in most cases.

Selecting Edges for Blending

Select Continuous Edge – this is the default option. One or multiple edges are detected by cursor, if they are all connected tangentially.

Select Single edge – each edge must be detected and selected individually.

Select Patch Outline – patch outline edges are detected.

You can select multiple edges. Edges may not be continuous, but they all must be from the same solid.

Selecting Edges for Blending between Commands

To detect edges between commands, press and hold Shift and move the cursor. You can also choose edges selection from pop-up menu, if you click an empty area and if you don’t have already selected solids.

For filleting, you can select optionally:

- Whether the rounded corners will be created, if two edges at the corner are rounded and the third edge is rounded finally. You can suppress the creation of rounded corners if the surface of the solid will be selected for further unbending.
- If the filleting of the edge will be cut in case when the selected edge segment has another adjacent segment tangentially connected and not selected for blending (continuous edge is not selected completely, only some segments are selected).

3D Fillet - RN3, Ctrl + F

3D Chamfer - CH3, Ctrl + R

For chamfering, you can define a chamfer distance different for each side of the chamfered edge. If you select an edge of a threaded surface (for instance, end of screw or edge of threaded hole), optionally, the chamfer distance may correspond to the respective thread.

Deleting Solids

Delete Solids – RMS, Ctrl + D

Deletes selected solids. You can select entire solids as well as their components. For example, you can use this function to delete holes, fillets, ribs, etc. If such parts are selected, the solid will be regenerated, which may take time for complex objects.

Editing Shape of Solids

Edit Solid Element Shape

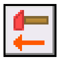
Edit Solid - MSO

This function allows you to:

- Edit the shape of one or more similar basic solids together. This is the default option whenever editing of basic solids is started.
- Change basic solids locations within the entire solid. See *Transforming and Copying Solids (page 229)* for more details how to change object's location.
- Translate basic solids according to previously changed dimensions.

Method of solid's shape editing is selected from toolbar:

Icon	Key	Use
	N/A	Edit shape of basic solids or blending
	N/A	Transform elements within solid
	N/A	Translate basic solids according to previously changed dimensions
	N/A	Change diameter of pipe or wire
	N/A	Change thickness of shell
	N/A	Shell pattern edit
	Enter	Finish selection of elements to be edited

	N/A	Undo edit change
	Enter	Finish editing
	N/A	Skip editing and set creation properties
	N/A	Back to 3D, unchanged

The Enter key (finish) is available either for “finish selection” or for “finish editing”.

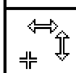
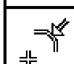
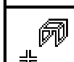
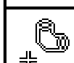
Edit Solid Element Shape

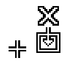
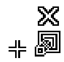
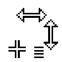
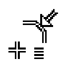
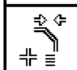
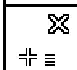
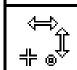
Select a basic solid to be edited. After selecting the first object, you can select more same basic elements from solid. Then confirm selection. If multiple objects were selected, shape change is performed for all of them at once. For instance, you can change several holes, fillets or any same elements together. Following objects are considered as the same elements:

- Basic shapes as boxes or cylinders, see *Basic Solids (page 200)*.
- Holes, predefined cuts or predefined grooves.
- Fillets or chamfers.
- Objects created by profile extrusion, rotation or lofting.

For instance, if the first selected object is a box, then other selected objects can be only boxes until the selection is finished. If the first object is a fillet, then next can be only fillets, etc. To distinguish how an object will be edited, the cursor is changed according a type of the object.

Cursor types:

Cursor	Use
	Object cannot be selected – object belongs to different solid than first selected object or object is different than first solid
	Spatial dimensions will be used for object’s shape modification
	Object is fillet
	Object is chamfer
	2D creation profile will be edited
	Object is shell; thickness will be changed
	Object is pipe or wire; diameter(s) will be changed.

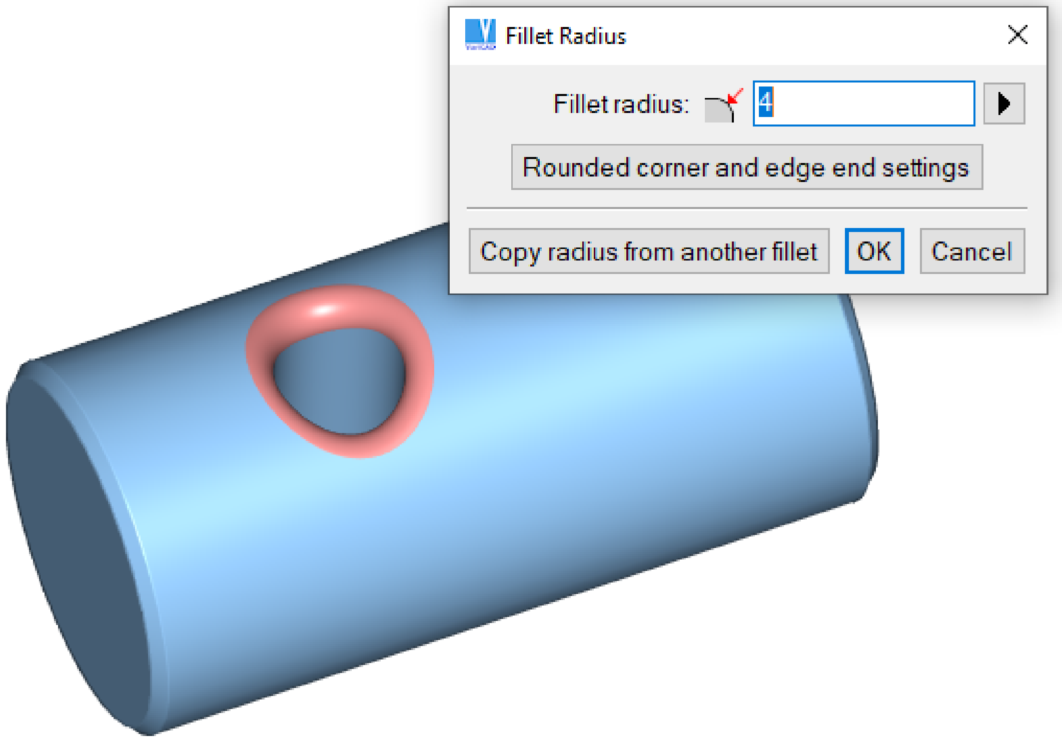
	Object cannot be changed. Object is imported from STEP.
	Object cannot be changed. Object is in active section.
	Shell shape editing, spatial dimensions will be used.
	Shell shape editing, fillet radius will be changed.
	Shell shape editing, chamfer distance will be changed.
	Shell shape editing, 2D creation profile will be edited.
	Shell shape editing, element cannot be changed – object is different than first selected object or object is imported from STEP
	Pipe segment editing, shape of straight segment or elbow will be changed.

As a basic solid, you can select solids used in Boolean operations - a solid added to another solid, or a cutting solid used to remove volume from another solid. As a basic solid, you cannot select any object imported from STEP. If the solid to be edited was created from a 2D profile, such as an Extrude or Revolve, you can edit the 2D profile. If the cursor passes over such an object during solid selection and the object is highlighted, the creating profile is also displayed. Together with this profile, its axes are displayed, too. This allows you easy orientation while editing the profile. After selecting the solid, the system switches to 2D drawing in 3D, and 2D editing functions are available. The profile is displayed with its original X and Y axes. When the editing is completed, click the icon on the Edit 2D Profile toolbar. Then reselect the profile's 2D objects. See also *Defining a 2D Profile (page 171)*. When editing 2D creation profile, you can leave the task only by clicking any icon in 2D Edit toolbar. This toolbar offers you to:

- Finish editing. After such an option, select a new profile and editing is completed.
- Step back to another solid selection.
- Edit solid using spatial dimensioning or table of solid dimensions, if the edited object was originally created as a box, cylinder, pyramid, hole or another predefined 3D solid.

Predefined solids (box, cylinder, pipe, etc.) or results of predefined Boolean operations (hole, etc.) can be comfortably edited using spatial dimensioning. Optionally, you can select solid's dimensions in the table (window). Such an option allows you to change also shapes, like add a recess to a hole. See also *Editing of Spatial Dimensions of Basic Solids (page 202)*.

For more information, related to editing of solids' profiles, see *Creating 3D Solids from 2D Profiles (page 171)* or *Rotating, Extruding, and Lofting Profiles (page 195)*.



Example of fillet editing

After specifying changes, the entire solid is regenerated. It is possible that editing will cause a situation in which the solid cannot be regenerated. Example: a 10mm hole is drilled into a cube with side 20mm. If the diameter of the hole is changed to 50mm, the solid cannot be regenerated. In cases like these, you will receive an error message, and the Boolean tree remains unchanged.

Change Basic Solids Locations within the Entire Solid

See *Transforming and Copying Solids (page 229)* for more details how to change object's location. For changing the object's location within entire solid, you can select object of any type (unlike the selection for shape editing), except the blending.

Translate basic solids according to previously changed dimensions

This option is available only if the translation distance and direction can be exactly determined. For instance, if you change height of a box, you can translate any other objects in direction of Z-axis of the box at a distance predefined as a difference of the previous and new height. Thus, you can easily preserve position of any objects relative to changed side of the box.

If the option is available, an arrow in a respective direction is displayed. You can select objects to be translated and then click arrow or icon. You can also click the icon first and then select objects to be translated.

If you need only to change the solid element's location use *Transforming and Copying Solids (page 229)* instead.

Editing of Shells

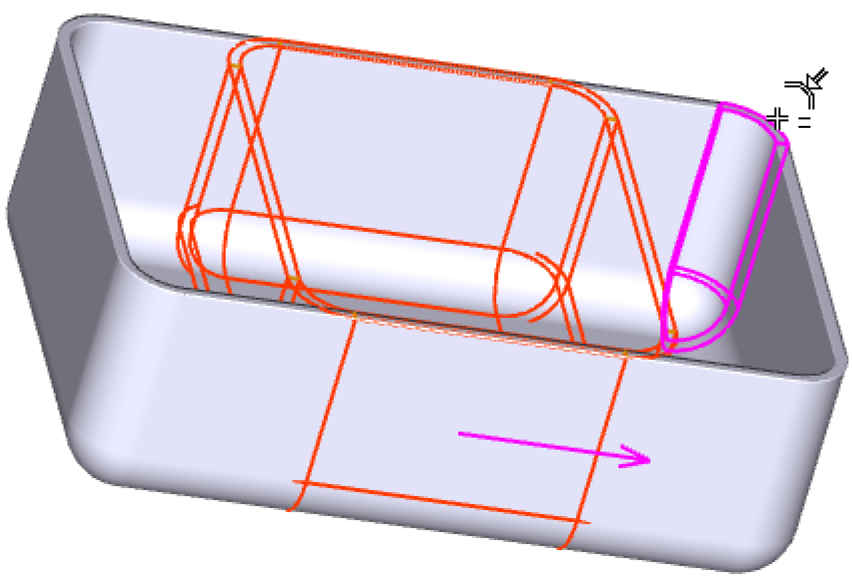
Shell is created as offset patches connected to selected patches at a given thickness. You can change:

- Thickness of entire shell (see edit options above).
- Shape of elements of solid the patches are selected from (shape of pattern).
- Position of elements of solid the patches are selected from.

Selecting elements for a shape change, the cursor type is changed differently than for other ordinary solids - see *Cursor types (page 224)*.

The shape of the pattern can be changed separately. You may extract a copy of the pattern solid back into 3D space. Then, you may perform any editing functions. After the all changes, select again the corresponding shell for pattern editing. Confirm, add or remove patches. The shell will be rebuilt. This method allows you to rebuild the shell, if you need:

- Pattern solid changed with Boolean operations.
- Work with geometric constraints within the pattern solid.
- Rebuild the shell from different pattern patches.



Example of the shell editing

Editing Pipes or Wires

Pipes or wires are created as a set of cylindrical segments and elbows.

You can change:

- Diameters of the entire pipe or wire at once (see the respective option above).
- Shape of selected elements of a pipe or wire. If used and more elements are selected, all dimensions are copied to all selected elements. Do not use such a method if the only diameter is changed. Use this

method for more elements only if all elements should have the same lengths and diameters or in case of elbows with the same angles and radii.

Transforming and Copying Solids

Solid Object Coordinate System

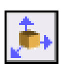
When locating a solid in 3D space, it is placed by its insertion point. The insertion point of each solid is defined during the solid creation. When solids are joined together, the default insertion point becomes the point of the object to which the other object is added (the root solid). When inserting multiple objects, the insertion point of the first selected object is used. You can redefine the insertion point location at any time.

Each basic element has its own axes. These axes are displayed when the object is inserted, or whenever its position changes. The origin of a solid's axes is at its insertion point. These axes can be used for translation and rotation for the attached solids, as well as for other solids.

3D Space Coordinate System

The global 3D X, Y, and Z axes are always displayed at the lower left corner of the 3D area. These axes indicate only the axis directions; the origin may be elsewhere. When first inserting a solid, the solid axes are identical to the global axes, until the solid position is changed.

Inserting and Transforming Solids



Translate, Rotate, Copy Solids - STC, Ctrl + Q

First select the objects to be transformed or copied. Then define their new location. You can select entire objects as well as their separate components such as holes. If components are selected, some parts may not regenerate after transformation. For example, if you move a hole outside its object, the hole cannot be generated. In this case you will receive a warning message. Methods of insertion, translation, copying, etc. can be selected from the 3D Location toolbar. This temporary toolbar appears whenever solids are inserted or transformed. There are also case-sensitive hotkeys for most options.

Defining Vectors and Rotation Axes

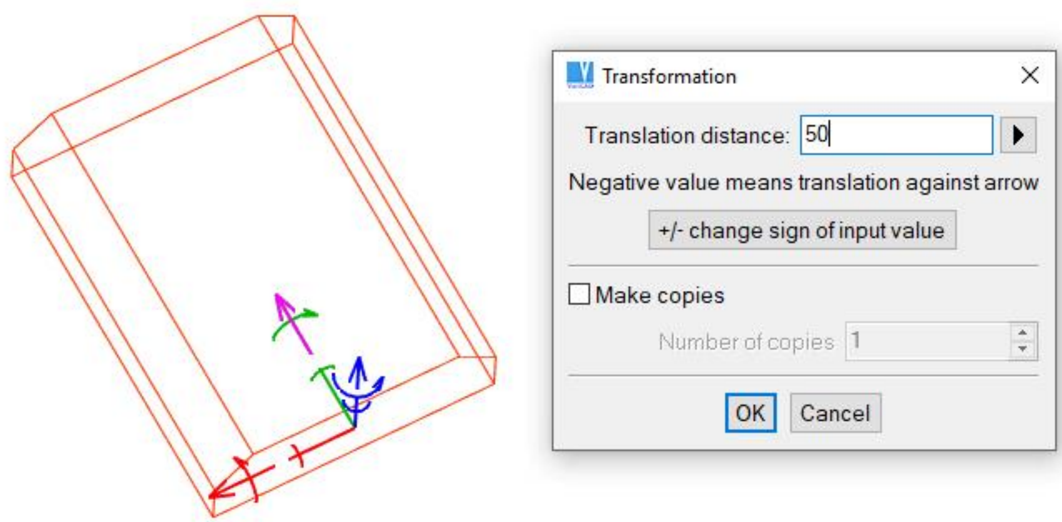
Some functions require a predefined vector or rotation axis. You can define these by the following methods:

- Two points
- Axis of a rotation surface (for instance a cylindrical surface)
- Selecting a point and normal plane
- Selecting X, Y or Z axis of the entire solid
- Selecting X, Y or Z axis of a solid's part. If this or previous option is selected, axes preview is displayed whenever the cursor passes over each object.

Transforming Objects Using their Axes

Whenever 3D objects are transformed or inserted, their axes are displayed at the location of the object's insertion point. Although you can use any transformation methods clicking icons in the panel (see the next sections), the most comfortable method is to use the axes directly:

- Click the outer section of the axis (arrow). Then you can define a translation distance for translation in arrow's direction. You can also make multiple copies in this direction.
- Click the rotation arrow. Then you can define an angle of object's rotation. Rotation is performed around the corresponding axis.
- Click the inner section of the axis. Then dragging along the axis is started.
- Right-click the inner section of the axis. Pop-up menu appears and you can select dragging, axis direction along or against the defined vector or additional rotation around the corresponding axis.
- Click the small rotation circle. Dynamic rotation starts. In next step, you must define a point the object is rotated with.

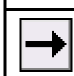


Example of solid's transformation. Solid is translated along its Z axis.

The color of the axes depends on current color palette. By default, for a dark background the X axis is red, Y axis white and Z axis green. For a light background, X axis is red, Y axis blue and Z axis green.

Translating by Distance

Moves solids according to a specified axis and distance.

Icon	Key	Use
	x	Moves along the solid's own X axis

	y	Moves along the solid's own Y axis
	z	Moves along the solid's own Z axis
	t	Defines a vector along which to move
	T	Moves along the predefined vector

Dynamic Translation

First, define the axis the solid is dragged along. If you want to move the solid along own X, Y, or Z axis, then simply click the inner part of the corresponding solid axis.

Then, define a location, which is projected to a dragging vector. The solid insertion point is translated along the vector to the projected location. Move the cursor to translate objects dynamically. Press Enter or left-click to finish translation. You can drag objects while moving the cursor or the objects are moved only if you detect a new location. See *3D Locations Settings (page 242)*.

If you press and hold left mouse button during dragging, automatic detection is turned off. This is convenient if you use dragging in increments.

Icon	Key	Use
	N/A	Moves dynamically along the solid's own X axis
	N/A	Moves dynamically along the solid's own Y axis
	N/A	Moves dynamically along the solid's own Z axis
	N/A	Defines a vector along which to move dynamically
	N/A	Moves dynamically along the predefined vector

Rotating by Angle

Rotates solids around a defined rotation axis, by a specified angle.

Icon	Key	Use
	u	Rotates along the solid's own X axis
	v	Rotates along the solid's own Y axis
	w	Rotates along the solid's own Z axis
	r	Defines an axis around which to rotate

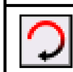
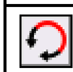
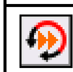
	R	Rotates around the predefined axis

Dynamic Rotation

First, define a rotation axis. If you need to rotate only around solid's own axis, then click the small circle at the inner part of the axis.

Then, define a reference point. Objects are rotated by movement of the reference point. As a reference point, you can use a tip of one of the two remaining axes.

Press Enter or left-click to finish rotating. You can drag objects simply moving the cursor or the objects are moved only if you detect a new location. See *3D Locations Settings (page 242)*.

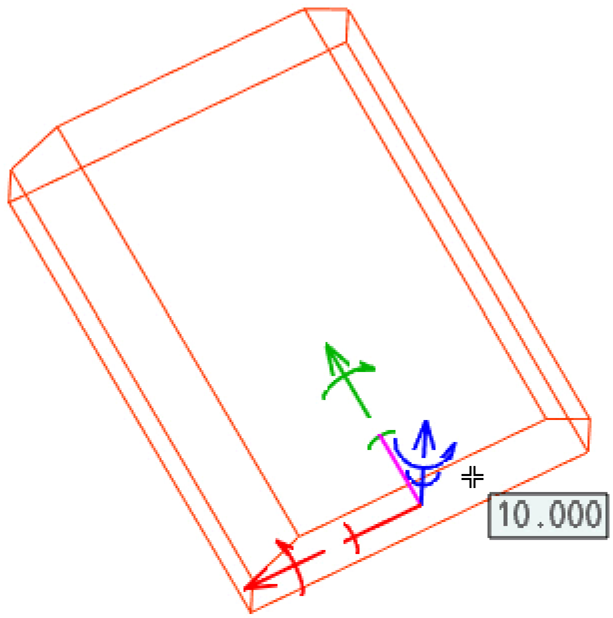
Icon	Key	Use
	N/A	Rotates dynamically along the solid's own X axis
	N/A	Rotates dynamically along the solid's own Y axis
	N/A	Rotates dynamically along the solid's own Z axis
	N/A	Defines an axis around which to dynamically rotate
	N/A	Rotates dynamically around the predefined axis

Dragging in Increments

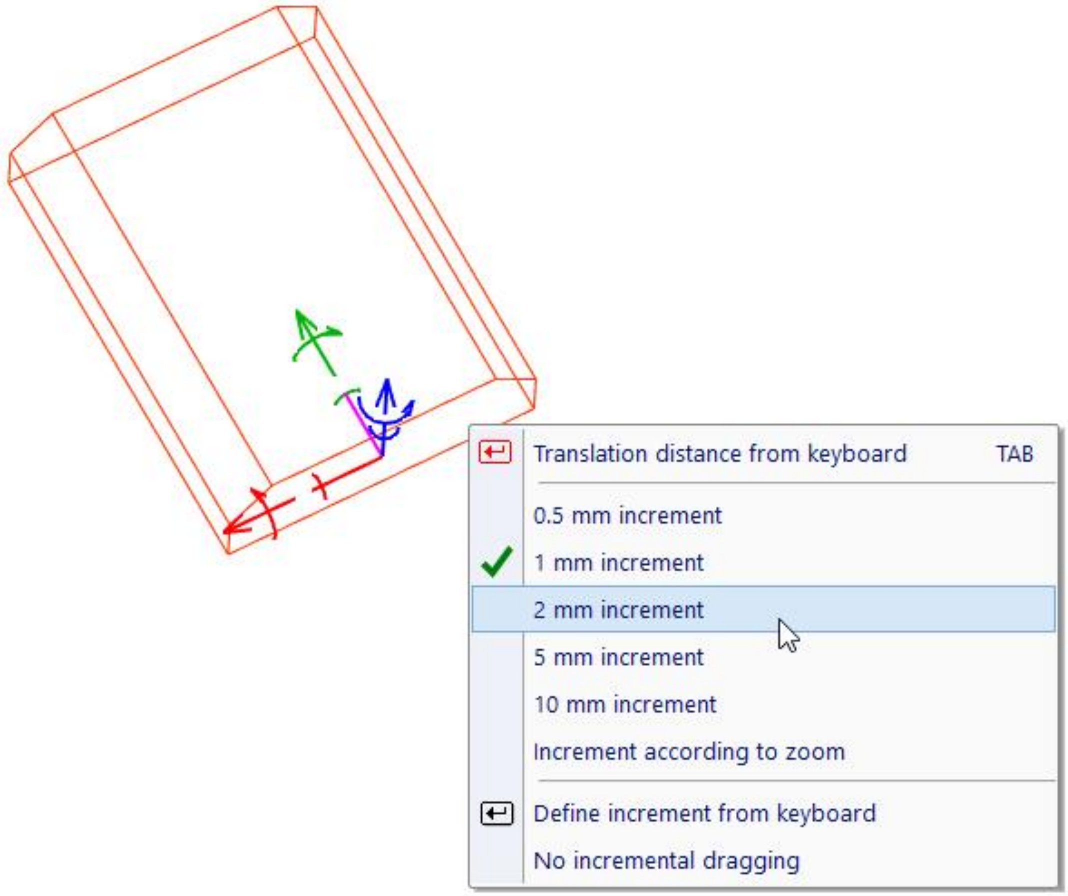
You can drag objects (move or rotate) incrementally. If you move the cursor over edges or edge endpoints, the location is always defined by projection of a detected point to the dragging vector (or similarly for rotation). However, if you turn on the incremental dragging and if the cursor is not crossing any detection points, the movement distance (or rotation angle) from the initial location is rounded according to currently used dragging increment. You can check the distance near cursor or in status-bar.

To set dragging increments or to turn them off, right-click during dragging and select values from menu.

This option also allows you to turn on or off the incremental dragging and to set values of increments for translation and rotation.



Example of solid dragging, distance from start of dragging is displayed near cursor. The solid is dragged along its Z axis.

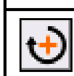
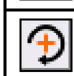


Example of solid dragging, after right-click. Pop-up menu offers increment options.

Additional Rotation around an Axis

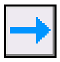
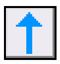
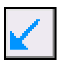
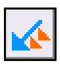
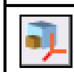
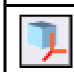
VariCAD allows you to perform additional rotation of transformed objects around an axis. You can select rotation from a panel containing transformation commands or from a pop-up menu after clicking the inner part of the translation axis. Then enter a rotation angle and the objects are rotated around the corresponding axis. Axes remain in the same position.

Additional rotation can be conveniently combined with geometrical constraints and parameters.

Icon	Key	Use
	N/A	Additional rotation around own X axis
	N/A	Additional rotation around own Y axis
	N/A	Additional rotation around own Z axis

Setting the Direction of Solids Axes

These functions can be used to reorient a solid by changing the direction of its axes.

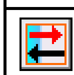
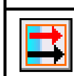
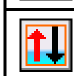
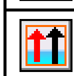
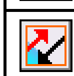
Icon	Key	Use
	N/A	Sets solid X axis direction along defined vector
	N/A	Sets solid X axis direction along previously defined vector
	N/A	Sets solid Y axis direction along defined vector
	N/A	Sets solid Y axis direction along previously defined vector
	N/A	Sets solid Z axis direction along defined vector
	N/A	Sets solid Z axis direction along previously defined vector
	N/A	Sets all solid axis directions along all axes of another entire solid
	N/A	Sets all solid axis directions along all axes of another part of solid

Positioning and Location at Surface

These functions enable you to direct a selected axis according to a surface and simultaneously, locate at a surface. Surface location is the location detected by cursor movement, under the cursor at a solid. It can be combined with location at an edge or edge endpoints.

For example, you can position a drilling tool, when a hole is created. The tool's X axis must be directed against a normal of detected surface. The tool is automatically located at a surface. As you move cursor, the tool remain stuck under the cursor at a surface.

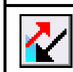
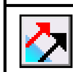
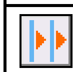
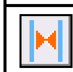
These features are accessible from pop-up menu, if you right-click the inner part of corresponding solid axis.

Icon	Key	Use
	N/A	X Axis against Normal and Locate at Patch
	N/A	X Axis along Normal and Locate at Patch
	N/A	Y Axis against Normal and Locate at Patch
	N/A	Y Axis along Normal and Locate at Patch
	N/A	Z Axis against Normal and Locate at Patch

	N/A	Z Axis along Normal and Locate at Patch

Positioning by Plane

These functions enable you to position solids relative to a selected plane. The solid axes can be directed along or against the plane normal. See also *Selecting Planes (page 242)*.

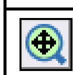
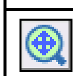
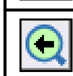
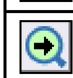
Icon	Key	Use
	N/A	Sets X axis against plane normal
	N/A	Sets X axis along plane normal
	N/A	Sets Y axis against plane normal
	N/A	Sets Y axis along plane normal
	N/A	Sets Z axis against plane normal
	N/A	Sets Z axis along plane normal
	N/A	Sets normal of any selected solid's plane along another plane normal
	N/A	Sets normal of any selected solid's plane against another plane normal

Zoom in on Transformed Solids

Whenever you want to insert a new solid, the solid axes are at the last inserted location by default. Or, if no objects were inserted yet, the axes are at the coordinate center. Often, the axes may be outside of the current zoom.

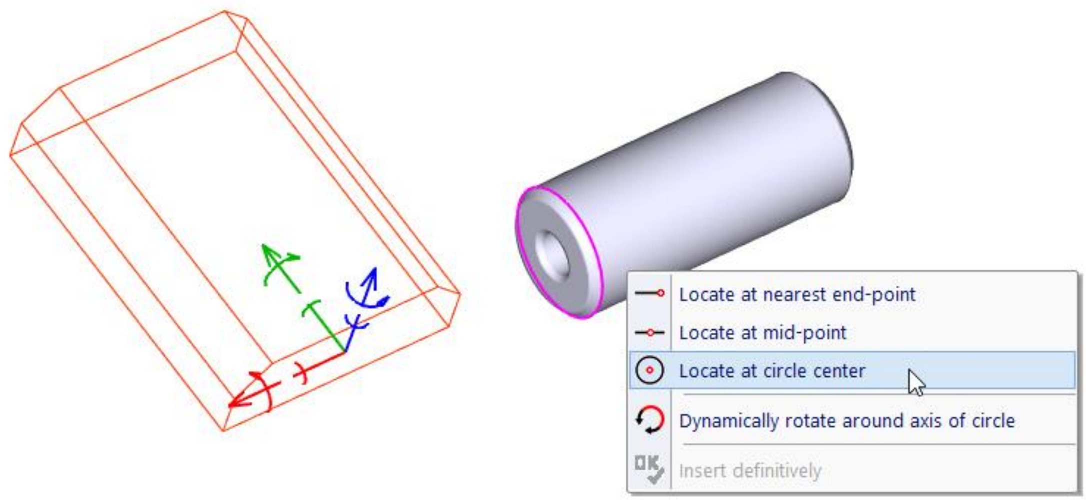
 **Zoom in on Objects' Transformation Axes**

This command opens a pop-up menu with following options:

Icon	Key	Use
	N/A	Zoom in on Objects' Transformation Axes
	N/A	Objects' Transformation Axes to View Center
	N/A	Undo View
	N/A	Redo View

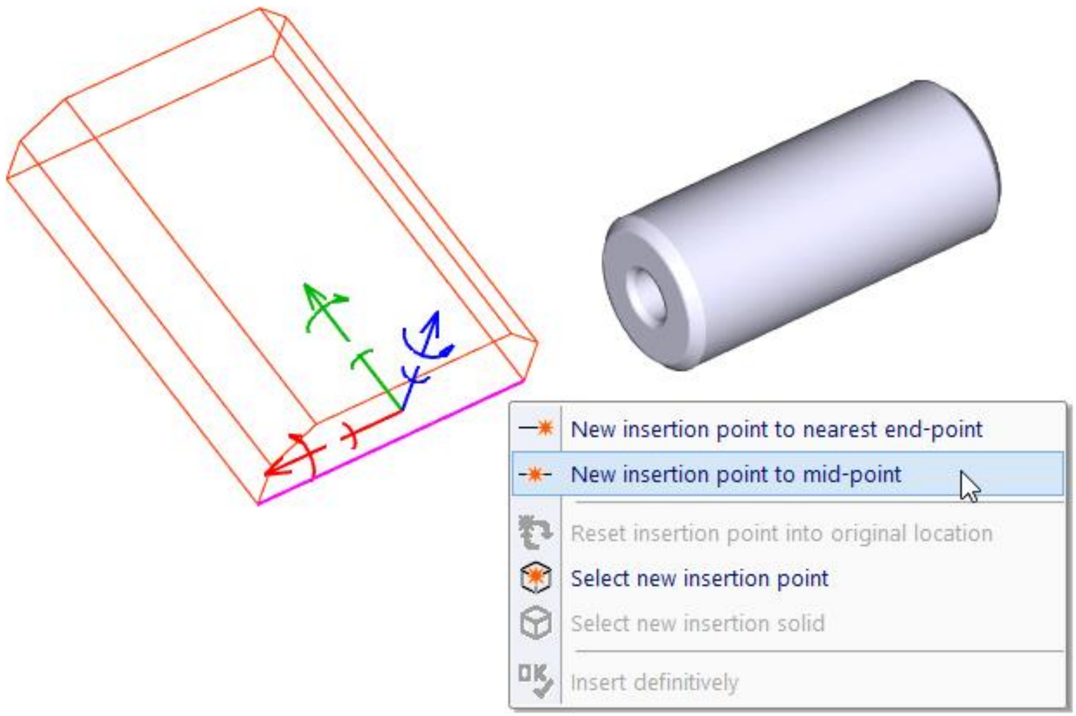
Additional Options for Right-Click Objects during Transformation

Transforming object location, you may right-click other objects outside the transformed group. Right-click an edge opens a pop-up menu with locate modes. This way you may comfortably locate object at an end point, center point, circle center etc.



Solid transformation, right-click an edge of another solid

Right-click the transformed solid, you open a pop-up with options related to transformation, like new insertion point etc. Previous option is available for multiple objects types (not only solids, but also axes locations). This option is available only for solid transformation.



Solid transformation, right-click an edge of transformed solid

Additional Boolean Operation, Constraints Definition

After the final position of transformed or inserted objects is defined, you may optionally perform a Boolean operation. Objects can be added to or cut from a selected root solid.

You may also select repeated Boolean operations. In such a case, the operation is performed each time you press Enter or click the corresponding icon (the Insert icon). The object is used as a Boolean operation tool, the operation is performed and then the copy of the original object is transformed again. Using this method, you may repeatedly insert and add solids to one selected root solid. You may also add multiple solids to one root solid at a time.

You can also define geometric constraints. Constraints are defined after a Boolean operation. For more information about the geometric constraints, see *Geometric Constraints (page 246)*. Available options are:

Additional Boolean Operation after Location

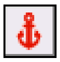
Insert and add to a selected solid.

Insert and cut from a selected solid.

 Insert and add to a selected solid repeatedly.

 Insert and cut from a selected solid repeatedly.

 Stop the repeated Boolean operations.

 **Define or Edit Constraints after Location**

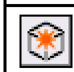
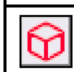
 Insert and add to a selected solid.

 Insert and cut from a selected solid.

 Insert and define constraints of entire solids.

Changing Insertion Point, Displaying Axes

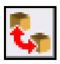
You can redefine a solid insertion point at any time. If you are transforming multiple solids, the insertion point of the first selected solid is used, but you can choose to use another insertion point. There are also options for displaying solid axes. See also *Solid Object Coordinate System (page 229)*.

Icon	Key	Use
	n	Defines new insertion point
	N	Defines solid whose insertion point will be used

Inserting and Copying

Transformed solids are inserted into their final position by pressing Enter or right-clicking. You can also choose to copy instead of insert. In this case, you must insert the transformed objects first. After insertion, the objects are copied and you can continue inserting. You cannot insert a copy into the position of the original object.

When an object is copied, its attributes are copied as well. Therefore, we recommend assigning attributes before copying. You can also create identical (linked) copies that share future edits (see *Identical Copies of Solids*.

Icon	Key	Use
	Enter	Insert - inserts object into their final position
	N/A	Undo - goes one step back, undoes previous transformation
	+	Copy
	-	Cancel Copy - turns off copying
	N/A	Copy, creates identical copies

Identical Copies of Solids

When copying solids, you can choose to create identical (linked) copies. This means that if any member of the copy group is changed, the change is automatically implemented on all group members. When one object in the group is selected, all identical copies are highlighted. Identical copies are also created when inserting a part into an assembly.

When copying members of a group of identical copies to another location, newly created objects are members of identical copies only if identical copying is turned on. In case of plain copying, these objects do not belong to a group of identical copies.

 **Break Identical Copy Link - RSI**

Enables you to select objects to be removed from the group of identical copies or from group of solids inserted from a part (assembly-part connection). This breaks the link between the objects.

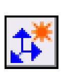
 **Break Identical Copy Group - RIC**

Breaks connection between all solids and a corresponding identical copy group.

 **Add Solids to Identical Copies – ASI**

This command creates an identical copy group from selected solids. First, select a solid – it will replace entire group of solids selected in next step. Such method may be useful if you originally copied solids as plain copies and later, it is necessary to have identical copies instead of plain copies.

Permanent Change of Axes of Solids Imported from Step

 **Permanent Change of Imported Solid Axes - CHAX**

Solid axes are located at a solid according to solid creation method. Although you can define offset of axes, system still remembers their default position and you can reset the user-defined offset. For more information, related to solid axes see *Solid Insertion Point (page 194)*.

If solids are combined into a Boolean tree, entire solid inherits the axes from so called root solid.

If a solid is imported from STEP, the axes are determined automatically and their position at a solid and their rotation relative to a solid may not be convenient. This command allows you to change axes of imported solid permanently – not only to define an offset, but also to rotate them.

Defining 3D Locations

Using snap points and significant locations can be used when inserting and transforming solids, as well as in other functions such as measuring and checking. You can use either toolbar icons or keyboard keys. The cursor automatically detects solid edges and curves. If the cursor approaches a snap point, a symbol appears next to the cursor. Clicking when you see this symbol selects the point. The following letters indicate snap points:

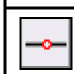
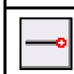
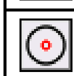
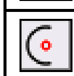
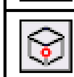
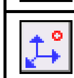
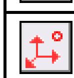
- E - edge endpoint
- M - edge midpoint

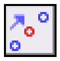
To snap at any location related to solid edge, you may use the same method as for solid transformation – see *Additional Options for Right-Click Objects during Transformation (page 237)*.

To snap to an endpoint E, midpoint M, or arc/curve center of gravity point C, press the corresponding key when the edge is highlighted. To use the toolbar icon, click the icon first and then click the edge or object. If you click on an edge when no snap point is indicated, the location is defined at the point on the edge nearest to the cursor.

If you select any specific location mode clicking icon in select toolbar, then the only selected mode is performed. For instance, if you select location of midpoint of edge, whenever you approach any edge, its midpoint is highlighted. The point can be detected clicking left mouse button wherever over corresponding edge. Predefined location mode persists until any selection is performed, or until any other mode is selected or until you click the same icon again.

There is a difference between the center of gravity of a curve and the center of an arc. The arc center is the point from which all arc's points are at same distance. Only for a full circle are the center and center of gravity identical.

Icon	Key	Use
	m	Midpoint of edge
	e	Edge endpoint
	2	Circle or arc center
	c	Center of gravity of edge
	N/A	Snap to nearest point on selected edge
	k	X, Y, Z coordinates
	d	Delta X, delta Y, delta Z from a specified point

	g	Between two points, at a defined distance from the first point
	b	Halfway between two defined points
	p	Solid (element) insertion point
	q	Entire solid insertion point
	N/A	Intersection of a rotation surface axis and a plane
	N/A	Intersection of a line and a plane

Selecting Planes

The cursor automatically detects planes. If plane selection is required and the cursor moves over a plane, all plane boundaries are highlighted. The plane is selected by clicking when the plane is highlighted. It is possible to have plane boundaries common to more than one plane and select plane of wire-framed object. In such cases, approach the plane boundary from inside the plane, proceeding toward the boundary.

3D Locations Settings

You can set 3D locations settings from command “CFG”. The following location options are available:

- Allow dragging by cursor - if used, you can dynamically change location by dragging the cursor. If not used, location is defined by specifying the insertion point. See also *Dynamic Translation (page 231)* or *Dynamic Rotation (page 232)*.
- Allow detection of inserted or translated solids - if not used, you cannot detect any points or planes of inserted or translated objects.
- Insert new solids at the location of the previous insertion (selected by default) - this is convenient for most situations. However, when changing zoom and pan settings, you might not see the new object. If this occurs, define the location at some specified point and the object will appear. If this option is not selected, solids are inserted into the origin of coordinate system.
- Wires of transformed objects are displayed always up. Otherwise they can be partially hidden by other solids above them.

Mirroring and Rescaling Solids

Mirror - MIRR3

The mirror plane must be defined first, by one of the following methods:

- An existing plane
- By three points
- By two axes (XY, XZ, or YZ) of a solid, or for Boolean solids, two axes of any element
- By two default axes of a solid, or for Boolean solids, two axes of its root solid

Then select the objects to be mirrored. The mirrored copies contain all attributes (if any) of the original solids. If attributes are copied, you will receive a warning message and you can verify that names and attributes are correct for the copies.

The copies are not identical (linked) to their originals. Therefore, some attribute names should be different. For example, the material can be the same, but the name “Right Side” should be changed to “Left Side” for the mirrored copy.

Scale - RSSO

Rescales solids. Select the objects and define the scaling center and value.

Exploded View of Assemblies

Exploded View of Assemblies – EXV

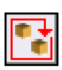
If you activate exploded view mode, you can change locations of assembly parts into their exploded alternatives. These changed locations are saved together with the standard locations. Next time you turn the exploded view mode on, assembly parts are automatically repositioned according to their exploded alternatives. In exploded view mode, you can transform solids to define their alternative locations, you can create 2D views, create bitmap outputs or print 3D images. Exploded view also stores own angle of view, zoom and pan. The mode can be turned on/off by the same command. Most of other commands turn the exploded view off.

Exports of 3D views into 2D drawing area are defined either for standard view or for exploded view. When the re-export of 3D views is performed, set of predefined exports is selected according to current view (set of exports of standard view vs. set of exports of exploded view).

By default, threads for exploded view are drawn with helix at threaded surface. For more information, see *2D View from 3D (page 262)*.

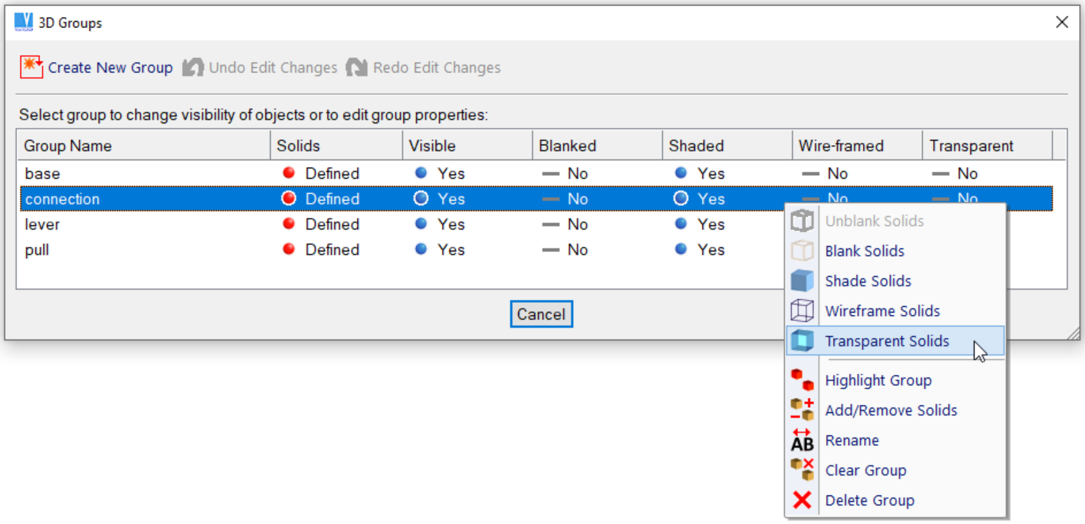
Groups of Solids

Solid groups are identified by a unique name. Each solid can belong to any defined group. Selecting solids, you can also select objects from a group. Groups can be used for changing visibility or shading.



3D Groups Management - 3GR, Ctrl + F1

Manages groups of solids. You can define a new group, change a group name, add or delete group objects, delete all group objects, or highlight objects belonging to a selected group. You can also change visibility or shading of all objects belonging to a selected group.



3D Groups window, pop-up with options for a selected group.

Parameters

Parameters may be used whenever a 3D object requires defined values. The value input can be a numerical constant, single parameter or mathematical expression, optionally containing parameters. Parameters may define object's dimensions or position within a corresponding solid, if used together with geometric constraints. Changing parameter values, all the affected 3D objects are rebuilt. Parameters are not supported in 2D.

Definition of Parameters

Parameters may contain letters and numbers. A parameter must begin with a letter, not with a number. Allowed letters are a... z and A... Z, usage of diacritical marks, Greek or Cyrillic letters or Japanese or Chinese characters is not possible. Definition of mathematical expressions is described in: *Mathematic Expressions (page 53) (page 27).*

Parameters may be defined in the function PAR (see below), or may be written into the input field instead of numeric values. If a parameter does not exist, you can confirm the creation of a new one. The default numeric value for numeric input is also offered as a default value of the new parameter.

Parameters in File

Once defined, a parameter is stored in a parameter table. The parameter table is a part of 3D space. Parameters are saved and loaded together with the corresponding 3D/2D file (document). If a document contains objects inserted from part files (if a document is an assembly file), then each assembly group has its own parameter table. The parameter table used for an assembly group is, in fact, the parameter table inserted from the corresponding part file.

If a file is inserted into the current file (document), a parameter table of the inserted file is compared with the current parameter table. If new parameters are not defined in the current table, they are accepted. If they exist and have different values, they are rejected and corresponding dimensions are changed to constant values.

Parameters in Scaled Solids

If you scale selected solids and the solids use parametric values, all such values are changed into constants. Similarly, all parametric values are changed into constants if you change units of the current document (millimeters to inches or vice versa). In both cases you are acknowledged before the operation.

Type of Parameters

Parameters are divided into three types:

- Linear parameters. These parameters are used for definition of length, diameter, thickness, fillet radius etc. Linear parameters may be used in mathematical expressions. Value of a parameter corresponds to current units (millimeters or inches).
- Angular parameters. These parameters are used for definition of angles. Angular parameters may be used in mathematical expressions. Value of a parameter is related to angular degrees.
- Thread parameters. This type of parameter can be used for thread definition. For instance, you may use the parameter “t” instead of “M10”. Whenever the parameter “t” is redefined, the correspondent thread is changed. Thread parameters may be used only as single parameters and not in expressions.

Working with Parameters

Add, change or remove parameter values – PAR

This function allows you to work with all parameters used in the current 3D space and assembly groups. You can:

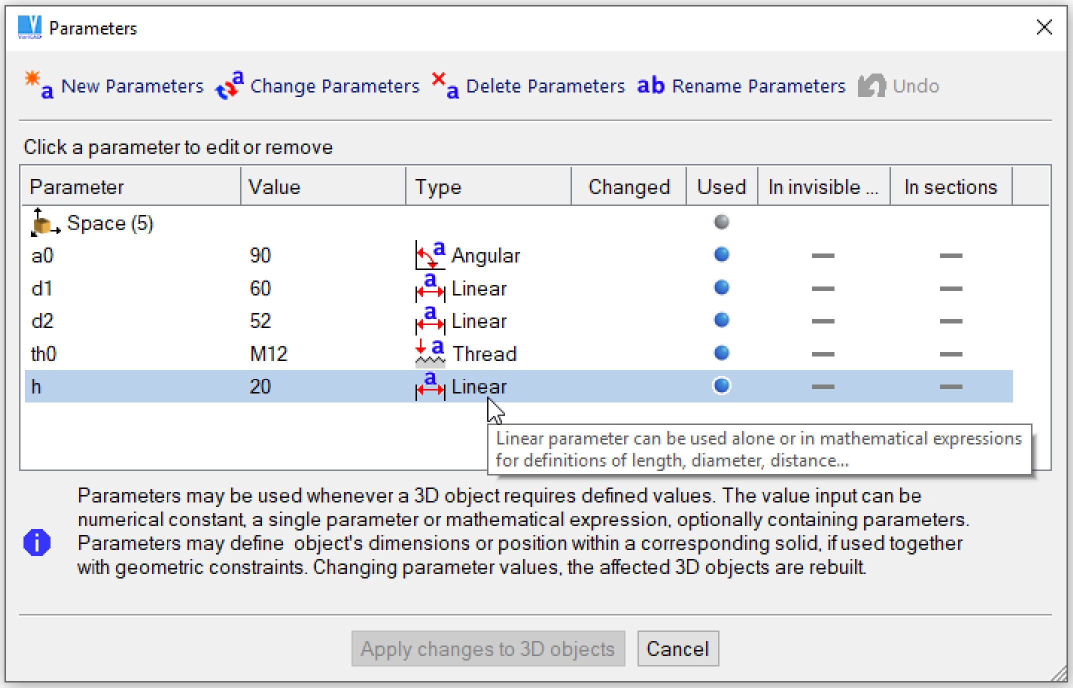
- Define a new parameter. Select a type of the parameter; define a parameter name and value. Thread parameter values are selected from a list of threads. If assembly groups are present, select a new parameter location – 3D space or an assembly group.
- Change a value. Select a parameter and change its value. You may change values of multiple parameters before the changes are applied to corresponding solids.
- Rename parameters.
- Delete parameters. If a parameter used in geometry is deleted, the corresponding parametric value is changed into constant value.

After changing parameter values, system checks:

- If new values of objects comply. For instance, you cannot define the inner pipe diameter greater than the smaller diameter.
- If the mathematical expression can be evaluated. You cannot divide by zero or value close to zero, you cannot calculate an angle if the sinus is greater than 1 or smaller than -1 etc.

Then, all affected parametric values are recalculated and solids are rebuilt. It is possible that element shapes or positions are incorrect for new parameters and the corresponding Boolean tree cannot be rebuilt.

Such solids are highlighted and changes are not finished. You may correct new values and rebuild solids again.



Parameter window. If solid hovers over icons, tool-tip with additional information appears.

Geometric Constraints

Geometric constraints can be defined among solid elements inside a Boolean tree, or among entire solids in 3D space.

If constraints are defined inside a Boolean tree (inside a solid created from multiple elements), they allow you to fix a position of a particular element. Whenever a shape of the solid is changed, the position of constrained elements is redefined according to a new shape. Such definition of constraints is very convenient – without them, you must often perform additional transformations of the element. Sometimes, it may not be possible.

If constraints are defined among entire solids, a position of a constrained solid or solids is changed, whenever a shape or location of related solid is changed.

In essence, constraints are based at:

- Additional transformations, performed whenever a shape or location of related objects is changed.
- Removing of degrees of freedom.

Both aspects describe how constraints work, but from different angle of view.

Constraints as Additional Transformations

Additional transformations are performed according to a constraint type, see *Available Types of Geometric Constraints (page 253)*. For instance: Define a constraint fixing an object at a distance from a patch and in direction of Y axis. Then, an intersection of Y axis and the patch is calculated. The constrained object is kept at a given distance, measured along the Y axis.

Additional transformation is performed:

- If you change a location of constrained object. After the change, object is moved along Y axis to keep the defined distance.
- If you change a location of object the patch is selected at (if you move the anchor)
- If you change a shape of object the patch is selected at (if you change a shape of the anchor), because the patch may be relocated.

After some changes, one or more constraints may not be performed. In the example above, it may happen if the patch is a plane and if the Y axis is parallel with the plane, after objects transformation. In such case, the constraint is ignored.

Constraints as Removed Degrees of Freedom

Each object in 3D space has six degrees of freedom. Three of them are rotations – around X axis, Y axis and Z axis. Three of them are movements – along X axis, Y axis and Z axis. If an object is constrained, one or several degrees of freedom are removed. For instance, object is fixed at intersection of Y axis with a selected path – removed degree of freedom is movement along Y axis.

If you fix rotation around two axes, the result is, in fact, fixed direction of the remaining axis. For instance, some constraints fix direction along a normal of a selected plane. Consequently, this constraint fixes rotation around remaining two axes.

Fixed movements, rotations or directions of axes are displayed either in constraint panel or in various dialogues – see below. And mainly, fixed movement or rotation is shown at a corresponding axis – as a short line across the tip of the axis.

Definition of Constraints

Constraints may be defined:

- If a new object is created and its location is defined
- If objects are inserted from a file or clipboard and their location is defined
- If an object is selected for transformation. In such a case, you may select an element from the Boolean tree (for instance, a hole). Constraints are defined within an existing solid. Or, you may select one or more entire solids. Then, the constraints may be defined only if you perform an additional Boolean operation and these solids are added to or cut from an existing root solid. The constraints are defined within such a complete solid.

Constraints cannot be defined:

- If you select the entire solid and a solid element.
- If you select multiple elements belonging to multiple solids.

- If an object is inserted from a mechanical part library.
- If the selected element belongs to a Boolean tree, but the other elements are only blendings.
- If you select multiple elements and they are already constrained in different groups.
- If you perform complex solid editing.

To define a constraint, click the inner part of corresponding object's axes. Optionally, you may click an icon in the Constraint panel. To edit a constraint, click the outer part of an axis. Then you may select:

- Delete the constraint.
- Highlight the constraint.
- Edit a distance used in the constraint (not available for all types of constraints)

You may work with constraints within a function transforming solids or their elements, see *Additional Boolean Operation, Constraints Definition (page 238)*. The constraints can be created or edited after the location is defined. Another option is to skip a location definition:

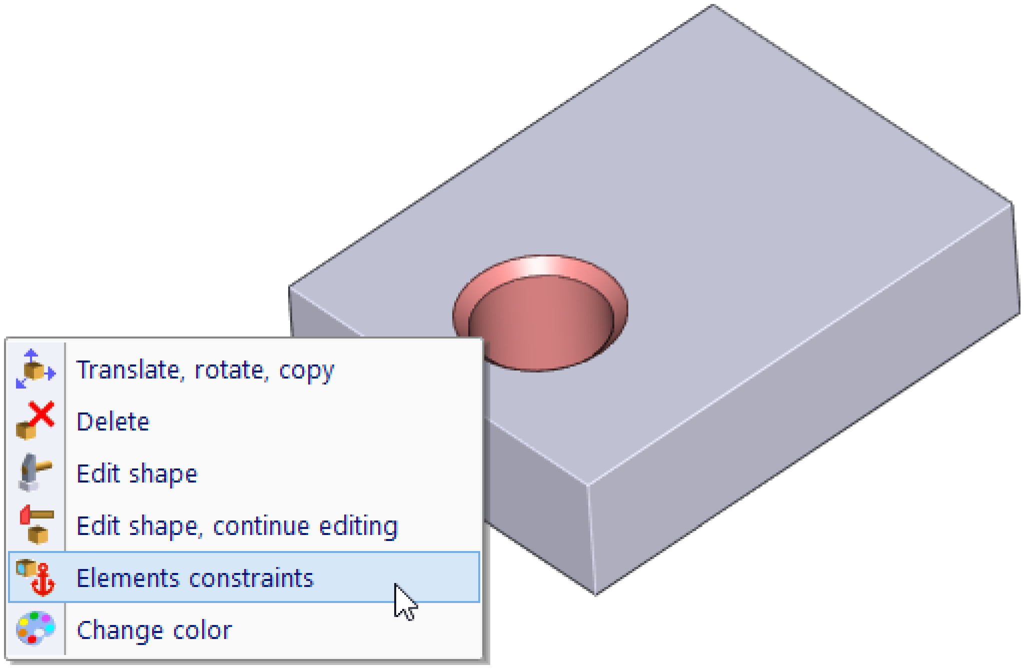
Create, Change or Remove Geometric Constraints among Solid Elements, CST

Create, Change or Remove Geometric Constraints among Entire Solids, CSTS

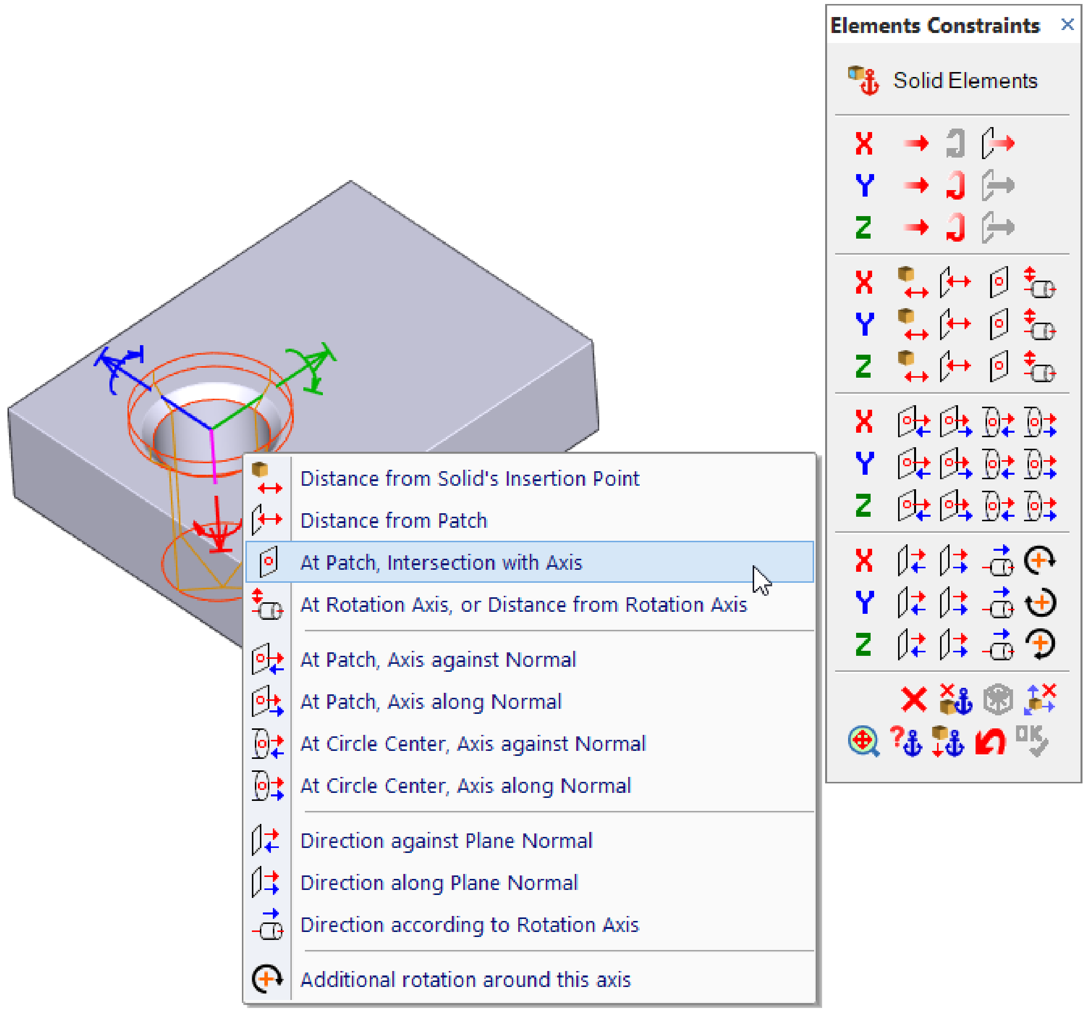
These functions work with constraints without necessity to define a location. It is possible to define constraints either for solid elements within one solid, or among entire solids in 3D space. Both functions work identically.

Either for the function "Solid Transformation" or "Geometric Constraints", transformations necessary for a particular constraint are always performed. It is more convenient to define a location and then to switch to constraints definition than to define a location only within the constraint definition. Transformation possibilities within the constraint definition are limited.

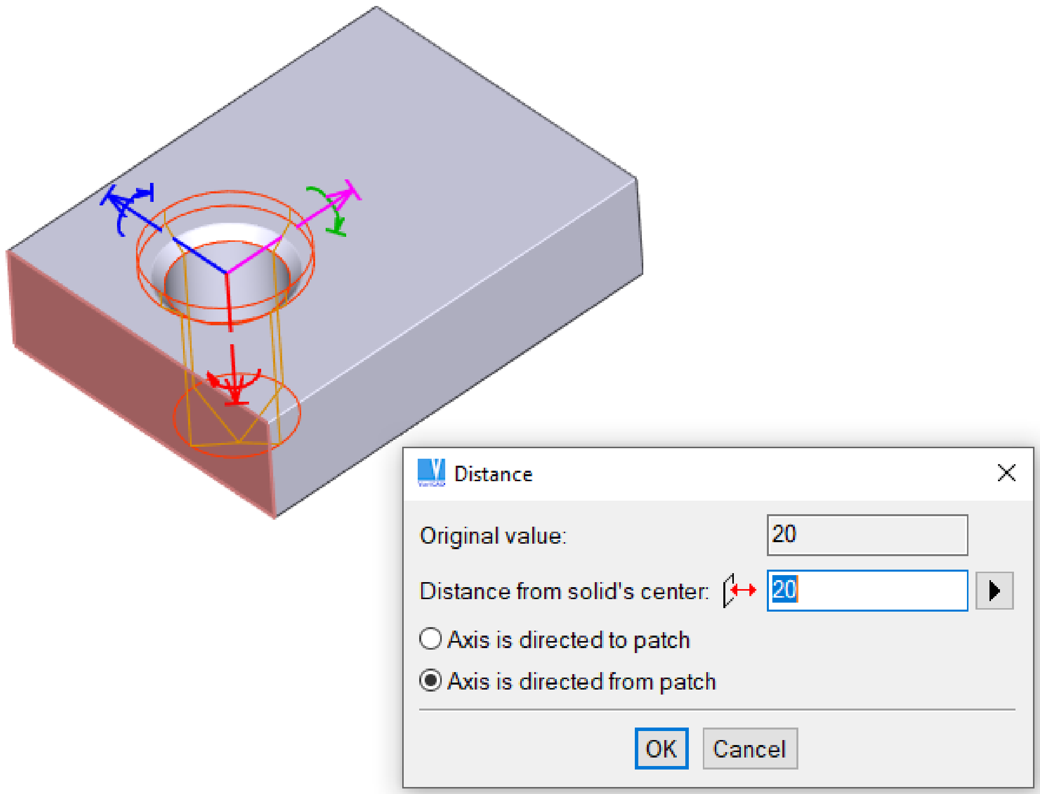
To create, change or delete constraints, you can right-click an object and select the command from pop-up menu. In case of constraints of a solid element, press and hold Ctrl and move cursor over objects. Then, single elements are detected and menu called by right-click is related to detected element.



Selecting constraint definition from pop-up, for a detected element (hole)



Constraint definition, Constraint panel



Constraint definition, example of constraint fixing a distance to selected patch

Constrained Objects

Constraints fix a constrained object in its movement along an axis or in rotation around the axis. The constraining is performed for the insertion point of the object. You may change a location of the insertion point until constraints are defined. Then you may change the location of the insertion point only temporarily within the current function. The change is not accepted permanently.

Also, you cannot change permanently a location of the insertion point, if the corresponding element is an anchor for another constrained object and this object is constrained right to the insertion point.

Selecting Constrained Objects

 This option, which is available for objects selection, allows you to select an entire constrained group of solid elements.

 Similar as previous, allows you to select an entire group of constrained solids.

If you select objects for transformation, the complete constrained group is detected. You cannot change a location of a single element of the constrained group individually.

Automatic detection of a constrained object is displayed at the cursor:

Cursor	Use
	A constrained group or constrained object is detected
	The already constrained object cannot be selected for the current constraint definition

Selecting Constrained Objects from Scheme

This option, which is also available for objects selection, allows you to select either constrained group of objects or to select objects the group is constrained to.

Constrained objects are displayed in the scheme. Such a scheme shows you legibly all dependencies – you can see the entire chain of constraints. Before selection, you can optionally highlight a constrained group – so you can see exactly which object or objects will be selected. Also, you can optionally click an object in 3D space and if constrained, it is highlighted both in the scheme and in 3D.

If you want to select anchors (or master solids, or solids the group is constrained to), click a corresponding column in the scheme. Then you can highlight each constraint individually – again because you will see which object will be selected. Or you can select the anchor of each constraint.

Generally, a group (often containing only one object) can be constrained to multiple different objects.

Constraining Multiple Objects

You may select multiple objects for constraining. The insertion point is the point of the first selected object. If you need to add a new object into a constrained group, select the group and a new object (or multiple new objects). They are added automatically.

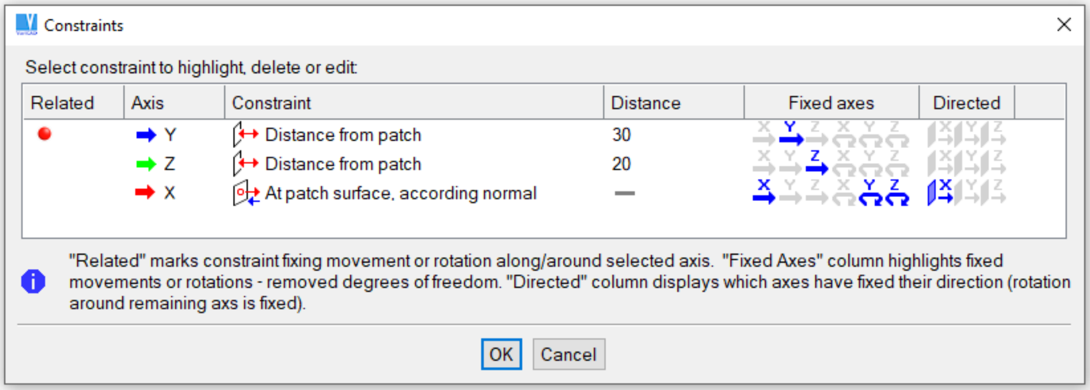
This option, which is available in the Constraint panel, allows you to remove an object from the constrained group.

Display Constrained Links

This option displays scheme of all constraints, so you can see the currently defined constraints in relation to other constrained objects.

Display, Edit or Delete Defined Constraints

Allows you to work conveniently with all constraints currently defined for the selected object (objects).



Dialog panel containing a list of currently defined constraints

Cancel All Constraints

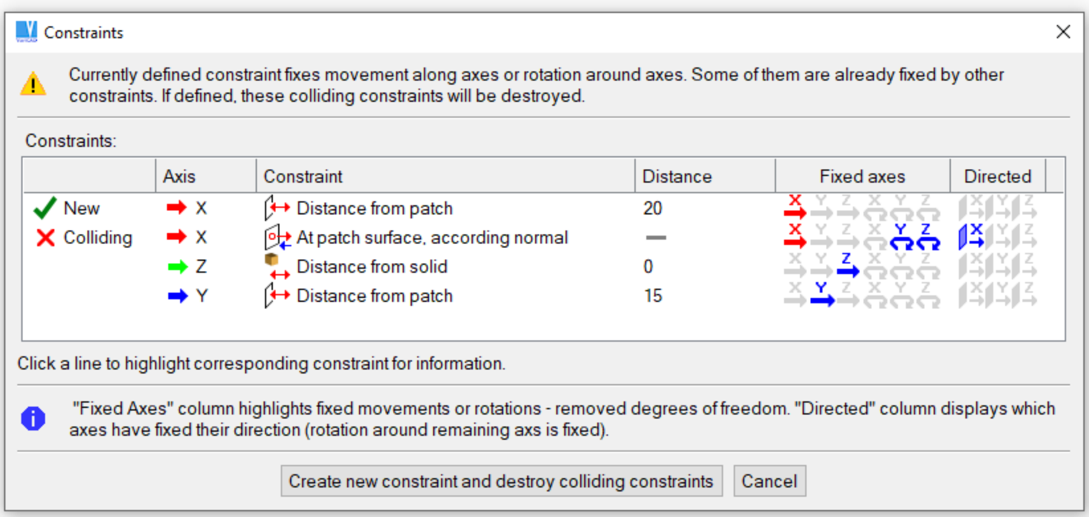
This option, which is available in the Constraint panel, cancels all constraints of the currently defined group.

Types of Geometric Constraints

All constraints are related to a specified axis. Direction of the axis may be selected along or against to a normal or another axis in dependence on a particular situation.

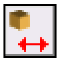
Definition of a constraint rewrites all existing constraints whose fix any axis of the object the same way as the new constraint. Fixed movements, fixed rotations or defined directions for the constrained object are displayed in the upper part of the Constraint panel.

If any of existing constraints has to be removed by redefinition, a dialogue panel is displayed and you can confirm such change.

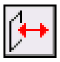


Example of a new constraint colliding with existing ones

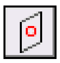
Available Types of Geometric Constraints



Distance to solid's insertion point. Constrained object is at a defined distance from the insertion point of a selected object (another solid element). Distance may be either positive or negative (along or against the axis arrow), or zero. This type of the constraint is often used for definition of a distance between holes or other solid elements.

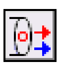


Distance from a patch, the axis is not directed. Intersection between the axis and the selected patch is calculated. The constrained object is fixed at a specified distance from the intersection.

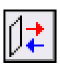


Location at a patch, the axis is not directed. This constraint is similar to the previous, but the distance is always zero.

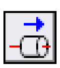
Location at a patch, the axis is directed according to a normal. The nearest point at a patch is found. Then the object is moved to the nearest point and the corresponding axis is directed against or along the normal of the patch at the location. This type of the constraint fixes the object at a surface and adjusts always its orientation. It can be used often for location of a hole – the object is always at the surface and the axis is always oriented perpendicularly to the surface.



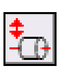
Location at a circle center, the axis is directed according to the normal of a planar surface. This constraint is especially useful for joining pipe or shaft segments.



The axis is directed according to a plane normal. This constraint is performed always as first before the other constraints. It defines orientation of the constrained object. Only one axis can be oriented this way.



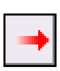
The axis is directed according to a rotation axis. The axis is parallel to the axis of the rotation surface.



Object is constrained at a distance from a rotation axis. An axis selected from the remaining two axes is simultaneously directed according to the rotation axis.

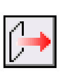
Removing Constraints According to Corresponding Axes

Clicking an outer part of object’s axes, you may select deletion of the corresponding constraint (see above). If you need to cancel the constraining of a selected axis, use following options:



Constraint fixing movement along the axis is deleted.

Constraint fixing rotation around the axis is deleted.



Constraint fixing alignment of the axis is deleted.

Chain of Constraints

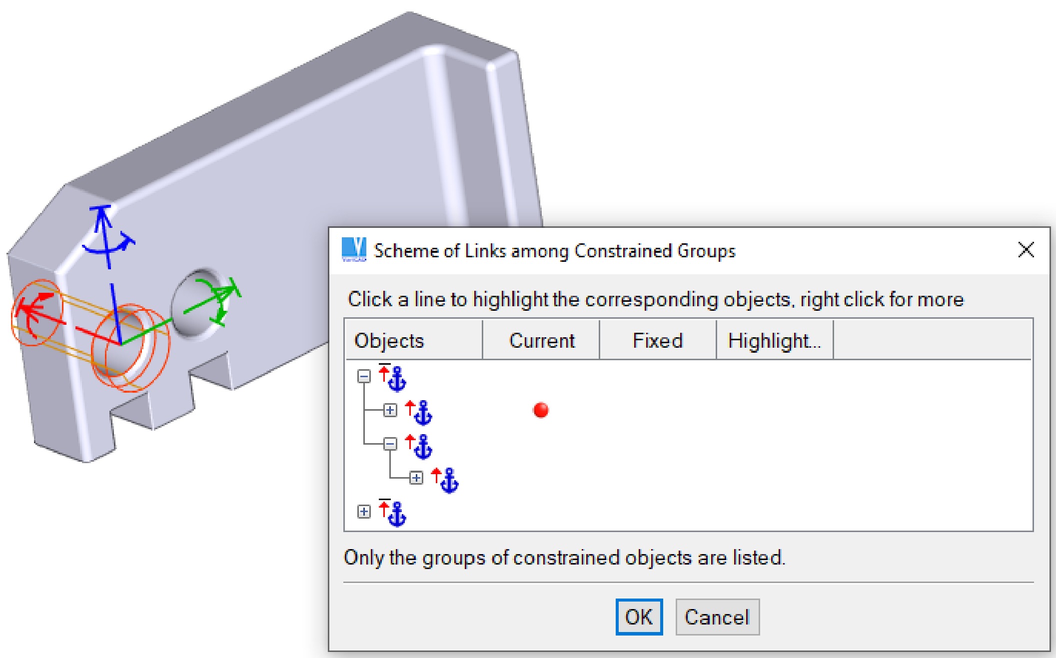
Constraints within a solid cannot be defined without a limitation. Basically, if an object A is constrained to an object B, then the object B cannot be constrained to the object A. All objects can be constrained to an object which is not constrained further.

If the object A is constrained to the object B (for instance, fixed at a distance from a patch of the object B), and the object B is constrained to the object C, a sequence of objects A, B and C creates a constrained chain.

Regarding the already existing constraints, you cannot anchor a constrained group at an object, if:

- The object is from a different solid.
- The object is from a different chain of constraints,
- The object is constrained at the currently defined group, not only directly, but also over multiple constrained links.

System automatically blocks the selection of anchors, if the constraint is not possible. The cursor is automatically changed at such a situation (see *Selecting Constrained Objects (page 251)*

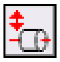


Chain of constraints

Fixed Object within Constraints

When the position of constrained elements is redefined according to a new shape, it may be moved inconveniently. For instance, you may define a chain of constraints within a shaft. After changing a length of a segment, all remaining segments are moved. You may want objects to be moved in opposite direction. You can fix a selected element from the entire solid. Then the element remains always in the same position

– it is not translated or rotated. If no element is fixed, then elements without constraints remain at the same location.



This option, which is available in the Constraint panel, allows you to fix a selected element to the current location.

Constraining Angles

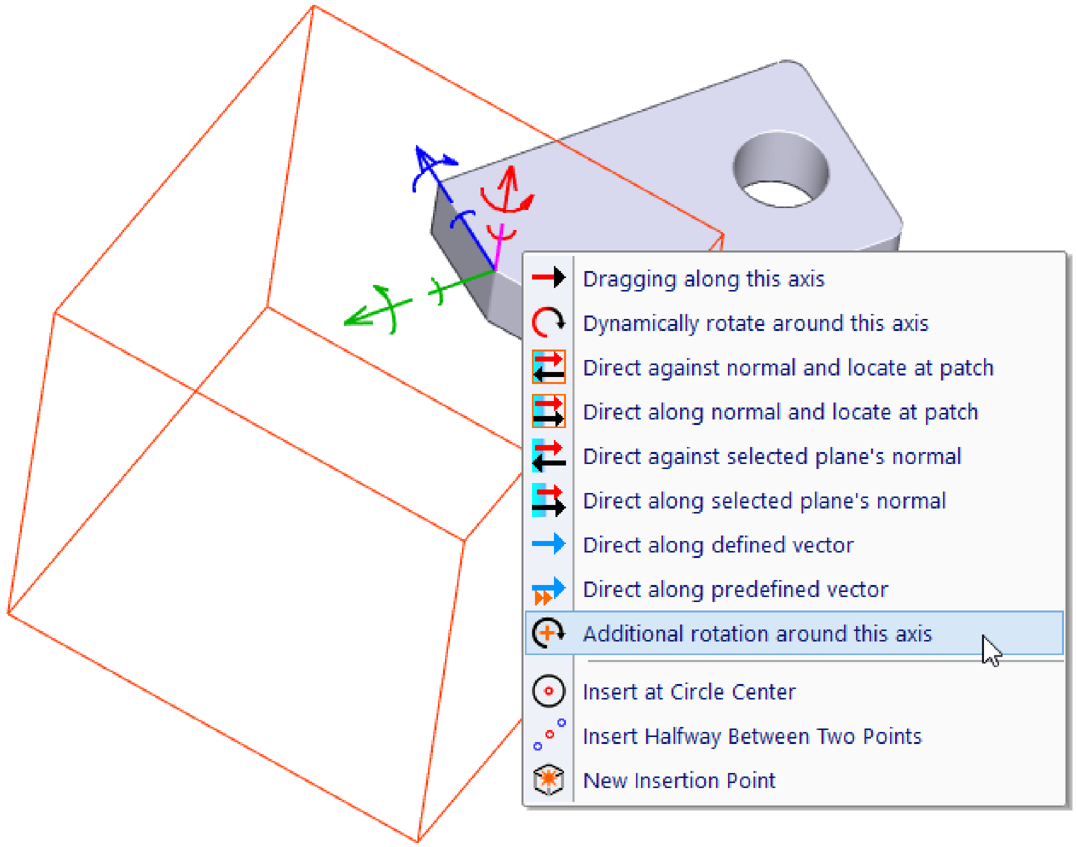
To constrain an angle, perform an additional rotation around an axis first.

See *Additional rotation around an axis (page 234)*. As angular value, enter or define a new parameter. Then constrain selected objects, if they are not already constrained.

It is necessary to fix direction of at least one of remaining two axes. For instance, the example displayed below constraints angle of rotation around X axis. Here, you should fix direction of Y or Z axis. Preferably, fix a direction of one of these axes along a normal of planar surface.

If you change the corresponding angular parameter, the angle is recalculated and constrained objects are rotated according to the new angle. The necessary condition for fixing angles by parameters is that the objects must be already constrained at least once. Direction of any perpendicular axis must be fixed. Otherwise, after a change of the angle the axes are rotated instead of the objects.

Another advantage of additional rotation around axes combined with constraint definitions is a possibility to constrain objects in any direction, regardless of the initial state of their axes. Objects are always constrained in direction of X, Y or Z axis. The axes are directed according to how a solid was created, and the direction may not be always convenient. After transformation of objects and additional rotation around an axis, you may constraint objects exactly in desired direction.



Additional rotation around X-axis

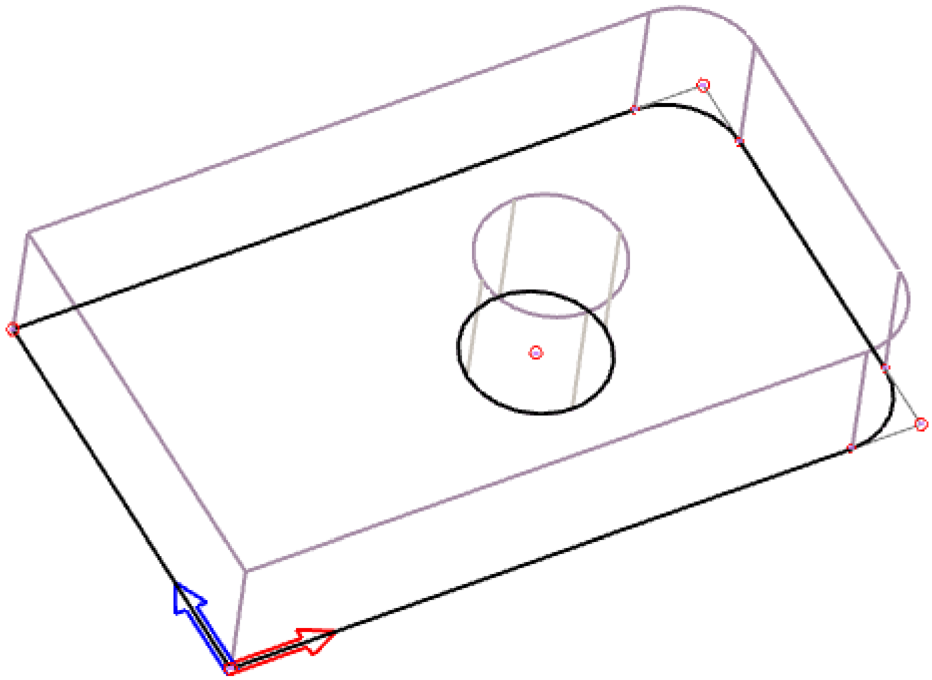
Constraints in Solid Creation 2D Profiles

You can change the shape of a solid created by rotation, extrusion or lofting of a 2D profile, when you define constraints and parameters for objects of the profile.

Constraining Objects in 2D Profile

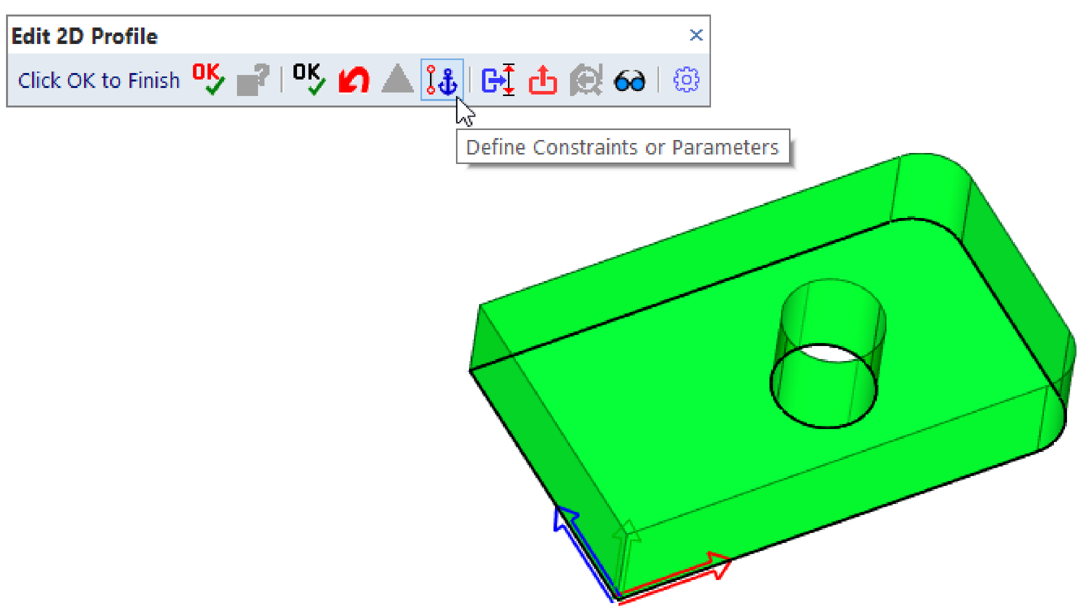
You can define constraints for a group of vertexes. Vertexes are:

- Endpoints of a line curve or arc
- Centers of a circle or circular arc
- Interpolation points of NURBS curve



Vertexes of a solid creation 2D profile

If you need to work with 2D creation profile constraints and parameters, select a solid for editing. Then click the corresponding icon in the edit toolbar – 2D edit mode is switched to profile constraint mode. After performing a change, the profile is always edited in constraint mode. From constraint mode, you can switch editing back to 2D edit mode similarly – clicking an icon in the constraint toolbar.



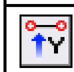
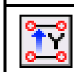
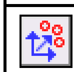
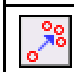
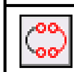
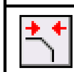
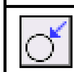
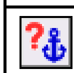
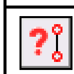
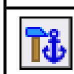
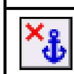
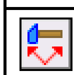
Switching 2D edit mode to profile constraint mode

Vertexes can be constrained at a distance to the center of coordinates or to another vertex (to an anchoring point). The distance may be a constant value, parameter or an expression containing parameters. If the corresponding parameters are changed, position of vertexes is relocated.

Vertexes can be constrained in direction of X axis, Y axis or in radial direction. Constraints can be defined for X axis and Y axis independently. However, if vertexes are constrained in radial direction, then constraints in x-direction or y-direction are not available and vice versa. Trying to define such an unavailable constraint, you are informed that the existing constraint will be overridden.

A constraint is defined for a group of one or more vertexes. After selecting vertexes, define a referent point. The constrained distance is a distance from the coordinate center or anchoring vertex to the referent point (vertex). All other vertexes are moved together with the referent point, if the constrained distance is changed.

List of Available Constraint Methods

Icon	Method
	Constrain objects in X direction to the coordinate center
	Constrain objects in X direction to another object
	Constrain objects in Y direction to the coordinate center
	Constrain objects in Y direction to another object
	Constrain objects under an angle and distance to the coordinate center
	Constrain objects under an angle and distance to another object
	Changes definition of an arc: 2 points, radius vs. 2 points, radius, and center
	Change a chamfer distance
	Change a radius of an arc, circle or fillet
	Display constraints and coordinate systems
	Check profile dimensions
	Edit an existing constraint
	Delete an existing constraint
	Edit an existing coordinate system

	Redefine the coordinate system for the created constraint. X-Y coordinates are defined under an angle.
	Finish editing
	Skip editing and set creation properties
	Back to 3D, unchanged

Selecting Vertexes

You can select vertexes clicking them one by one, or use following options:

Select vertexes inside the selection window

Select vertexes outside the selection window

Toggle between adding and removing from the selection set

If certain vertexes cannot be selected, they are displayed in different colors. Selection of some vertexes can be blocked, if:

- The vertexes have defined different coordinate systems than first selected vertex - if a constraint is created.
- The vertexes belong to a different constrained group than the group selected for editing or deleting.

You can also pick a set of vertexes first, finish the selection and then select a following step from pop-up menu.

Display Options

Working with a 2D solid creation profile, you can change display similarly as in edit mode:

Toggle between thick and thin outlines in 2D

Toggle entire display between shaded and wire-framed

Filleting, Chamfering and Radii of Circles or Arcs

Radius of a circle, arc or fillet can be defined as a parametric value. Chamfer distances can be also defined as parametric values. It is not necessary to define a constraint to change a value of radius or chamfer distance. Select an object and redefine the value.

Rounded or chamfered corners are detected as filleting or chamfering, if the blend operation was performed in 2D edit mode in 3D environment. If the profile was created in 2D drafting module, only filleting of two linear segments is recognized. In case of a rounded or chamfered corner, the end-point vertex is at location of the original corner. Connection of a fillet arc or chamfer line to the original linear or arc profile segments is displayed differently. The connection cannot be selected for constraint definition, but can be selected for values measurement.

Constraining Circular Arcs

By default, arcs are defined by two endpoints and radius. If necessary, you can add the arc center to the definition. In such case, the endpoints must be explicitly calculated if they are constrained - otherwise they may not be at the radius distance from the arc center.

An exception of constrained endpoints of circular arcs is a situation when the endpoints are constrained only in x-direction or in y-direction. For example: if an endpoint is constrained in x-direction, the x-coordinate is exactly defined. If this point is an endpoint of a circular arc, its distance from the arc center is also exactly defined. The y-coordinate of such an endpoint is recalculated after change of an x-constraint or radius of the arc. According to a corresponding geometry, there may not be a solution. If the solution exists, a new arc endpoint location is defined.

Constraining NURBS Curves

NURBS curves are created as interpolation curves defined by a set of points. The shape of the curve can be modified, if you change locations of the interpolation points. The interpolation points can be constrained as other vertexes – either each point individually or the set of points as a constrained group.

Editing Constraints

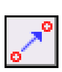
To edit a constraint, select a vertex first. If the vertex is a member of two constrained groups (for instance, a group constrained in x-direction and a group constrained in y-direction), select which group is edited. Add or delete vertexes to or from the selected group. Finally, confirm or edit the constrained distance. It is not possible to change the reference or anchoring point, if the edited constraint is constrained to another vertex.

Deleting Constraints

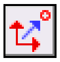
Select a vertex from a constrained group to be deleted. Confirm removal of the selected group.

Coordinate Systems

By default, the world coordinate system is used. X axis is directed to the right, Y axis is directed up. Location of the center of the coordinate system is defined during profile creation. If necessary, you can redefine the coordinate system for a group of selected vertexes. The coordinate system can be defined with following methods:

 Center at a selected vertex, X-direction to a selected vertex

 Center at a selected vertex, X-direction under a defined angle

 Center defined by XY, X-direction to a selected vertex

Center defined by XY, X-direction under a defined angle

Reset the coordinate system to default

If the center coordinate or X-axis angle is defined, you can use parameters instead of constant values. If the center position or X-axis angle is changed, position of all vertexes in the group is recalculated, too.

If the coordinate system for a set of vertexes is redefined, you can create a constraint only for vertexes with identical coordinate system.

Exporting Views and Sections from 3D to 2D

You can create 2D drawings from your 3D model by exporting views. For each view export, you can use all objects or only selected objects. Exported objects can be clipped to a defined rectangle, if you need to create only a small detail from a larger part.

Once 3D views are exported into 2D area, corresponding dimensions, axes or hatches are updatable. After changes in 3D and switching mode to 2D, they are adjusted automatically. See *Automatic Updates of Dimensions, Axes and Hatches after Changes in 3D (page 112)*

Creating 2D from 3D

2D View from 3D - 32E

The following options are available for exporting views:

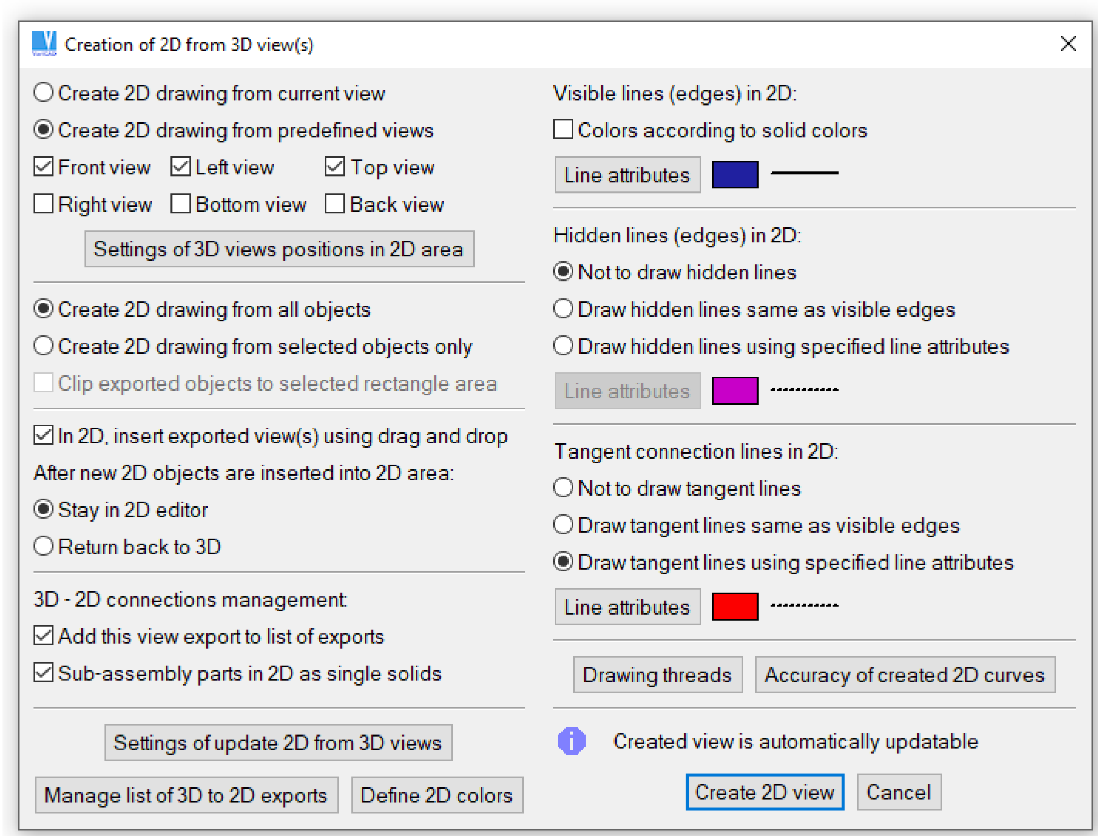
- Visible lines - you can select the layer, color or line type of created lines. Colors can be copied from the solids.
- Hidden lines - set to be removed by default, but you can turn this off. If hidden lines are to be created, you can set their 2D line attributes.
- Tangent connection lines - smooth connections of tangent surfaces, such as a fillet and its neighboring surface. Tangent connections can be removed, or created in different 2D line attributes.
- Define whether the current view is exported or define export of selected basic views, like the front view, left view etc.
- Optionally define a clipping rectangle, if the current view is exported.
- Select a method of insertion into 2D area.
- Select a method of creating threads. By default, threads are created according to common 2D drawing rules. If the threads are projected in the axial direction or direction perpendicular to their axes, (threads are in basic views like a side view, front view...), you may select line attributes of created 2D thread objects. You can distinguish ends of threads as well as secondary thread diameters.
- Settings of update of 3D views

- Manage list of 3D views exported into 2D
- Define 2D colors. In 3D, you can work with 32 colors, while in 2D the number of colors is only 9. This option maps colors of 3D solids to colors of 2D objects. Colors corresponding to this map are used only if the 2D objects are created according to the color option.

Define the export method, and select the exported objects, if necessary. Define the view position in the 2D drawing. Optionally, you can define 2D position using drag and drop or by translation new objects. If the latter possibility is used, define a translation vector (position “from”, position “to”) and confirm the insertion. 3D section outlines are exported to 2D as boundaries that can be detected as a single object. This boundary can be easily used in hatching functions. See *Hatching 2D Objects (page 102) (page 27)*.

You can also set a position of each view in 2D area:

- For created basic views, define a mutual position of the front view and other views. For instance, you can define whether the left view is inserted into 2D to the right or to the left side of front view.
- Define gaps between each view and distances from 2D area margins.



3D View export definition

List of 3D View Exports, Updating Views

The view export is added to a list of predefined exports, if during the export creation the option “Add this view export to list of exports” is selected.

The view export list is used whenever you need to update 3D views after 3D changes. Each stored view export contains:

- Export method - hidden line removal, attributes, etc.
- List of exported objects (optionally)
- Definition of corresponding 3D view
- Definition of active sections (optionally)
- Definition of clip rectangle, if any (if used, the window clips 3D objects before exporting is performed)
- 2D position of exported view - if this position is changed, all such changes are stored in the 2D list of transformations. You can translate, rotate or rescale the exported view. It is necessary to select all objects created from one view. See *Selecting 2D Objects (page 45) (page 27)* - selecting view. If only some 2D objects are transformed, this change is not stored in the list of transformations. To select entire 3D view, press and hold Ctrl and move over 3D view objects. The view is detected automatically.

If the view update function is invoked, each export from list is performed. For each export the view is set, the corresponding section is turned on, objects are selected and the 3D view is exported to 2D. These objects are inserted into 2D. All this is done automatically. You can select whether the old 2D objects will be removed or changed, to distinguish line attributes before and after the update.

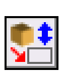
If you choose to change line attributes of old 2D objects, you can switch between old 2D objects and new 2D objects. This switching always highlights changes, and you can see what is new or what was deleted. Exporting a view creates only outlines. If there are any dimensions, hatches, or other 2D annotations, you can adjust them according to the changes. When all additional 2D changes are complete, remove the previous view export. If this view is not removed, the 2D update cannot be performed.

3D section outlines are exported to 2D as boundaries that can be detected as one object. This boundary can be used in hatching functions. See *Hatching 3D Sections (page 102) (page 27)*.

Update 2D after 3D Changes - 32EN

Creates new 3D-to-2D view exports. If reexports are already defined (if 2D views are already created, reexports are defined by default), you can update 2D from pop-up menu. Click an empty area to obtain menu containing general options. Or, 2D is updated if you switch into 2D mode and select update from pop-up menu.

Management of method of 2D drawing updates and management of individual 3D views exported into 2D is available from dialog panel *Exporting Views and Sections from 3D to 2D (page 262)* - see description of command above.



Old/New View Export, Updated 2D - SON

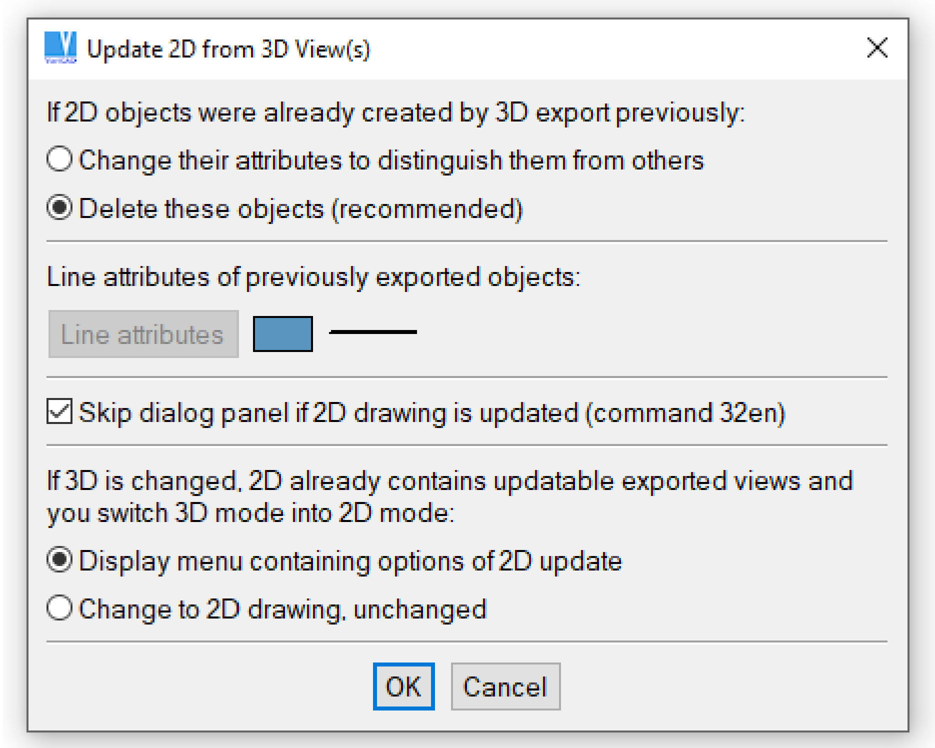
In 2D, switch between old and new 3D to 2D view exports to compare and update 2D after 3D changes. This allows you to easily distinguish changes and modify the corresponding 2D objects – especially dimensions and hatching.

Remove Previous View Export - ROL

In 2D, remove all old 2D view objects exported before the last export. Perform the function after all changes in 2D are finished.

We do not recommend to keep old and new exports, if dimensions are automatically updated.

Methods available for checking of automatically updated dimensions are described *here.* *(page 116) (page 112)*



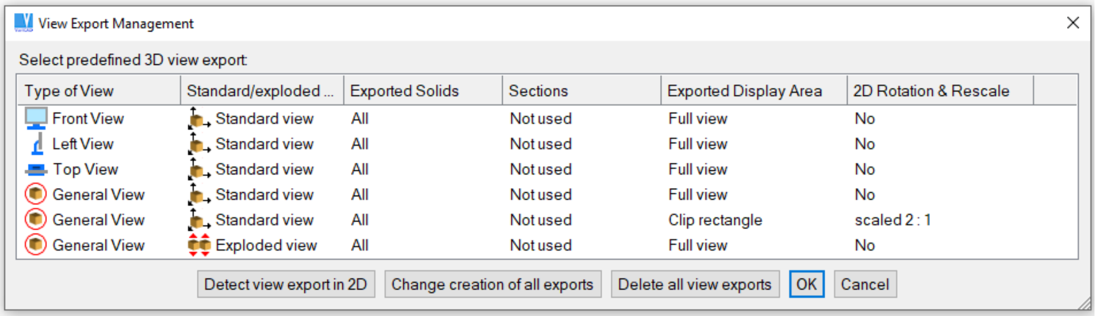
Updating 2D after changes in 3D. We recommend to set skipping of this dialog for each switch into 2D mode.

Update 2D after 3D Changes Setting – 32SET

Manages a list of predefined view exports. This function is available also from Update 2D after 3D Changes or from Creating 2D from 3D. The view export can be selected from the list as well as from corresponding 2D objects. You can:

- Redefine method of 2D creation for selected export
- Highlight corresponding objects in 2D
- Redefine method of 2D creation for all exports together
- Remove selected export

- Remove all exports



Type of View	Standard/exploded ...	Exported Solids	Sections	Exported Display Area	2D Rotation & Rescale
Front View	Standard view	All	Not used	Full view	No
Left View	Standard view	All	Not used	Full view	No
Top View	Standard view	All	Not used	Full view	No
General View	Standard view	All	Not used	Full view	No
General View	Standard view	All	Not used	Clip rectangle	scaled 2 : 1
General View	Exploded view	All	Not used	Full view	No

Management of defined view exports

There are differences between export definitions from standard view and from exploded view. For more information, see *Exploded Views of Assemblies (page 243)*

3D Sections

With VariCAD you can create 3D sections. Sections can be turned on or off at any time. If a section is active, displayed solids are cut by the sectioning tool. If the solid is part of an active section, some functions cannot be performed. If this happens, you will receive a warning message.

Each section is defined by:

- Name
- Section planes (sectioning tool)
- Solids that are sectioned

If you need to change the color of the section planes, use function *Change Color (page 206)*. Switch the select mode to selection of single solids (single parts of Boolean trees) and select a section plane as an object for color change.

For more information about exporting sections to STEP or IGES, see *How 3D Objects Are Converted to STEP or IGES (page 7) (page 6)*.

Section Planes, Sectioning Tool

Section planes are the planes of the sectioning tool. If the section is turned on, operation similar to Boolean cut is performed and the sectioning tool cuts the sectioned solids. As a sectioning tool, you can select a box or any solid created by extrusion. If the extruded profile contains more lines, the section has more section planes. Shape or location of sectioning tool defines how the solids are cut by sectioning.

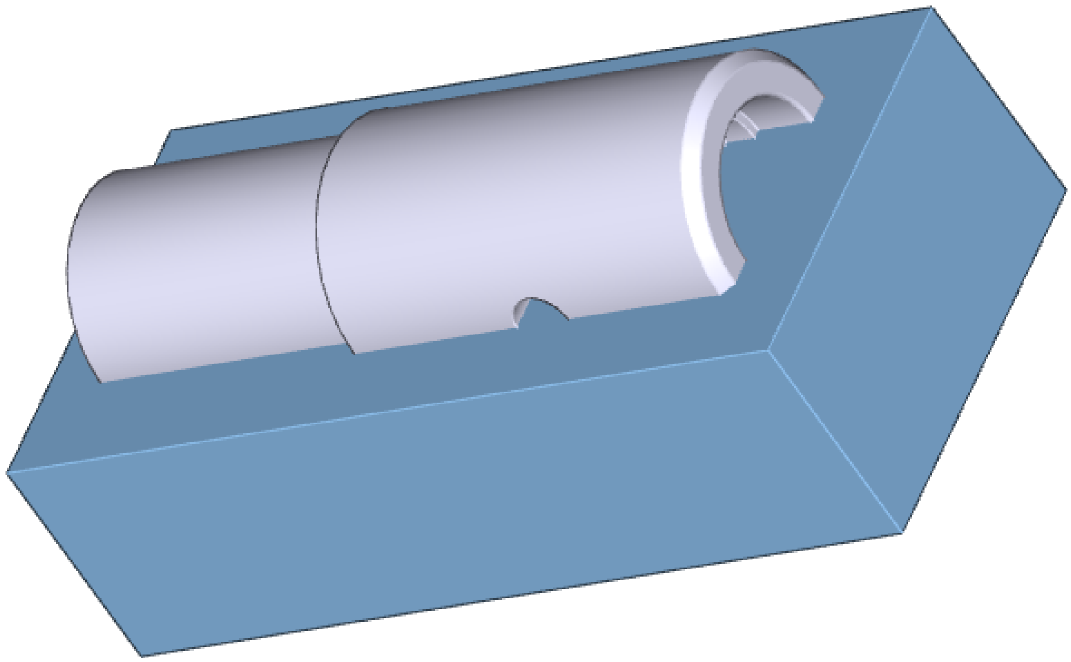
Instead of insertion of a single solid (box or extruded solid), you can select a section plane directly – using similar method as in *Cut by an Extruded Solid (page 218)*. Then, create a section tool contour and select its height and location.

Such section tool is edited by sketching and redefinition of extrusion height.

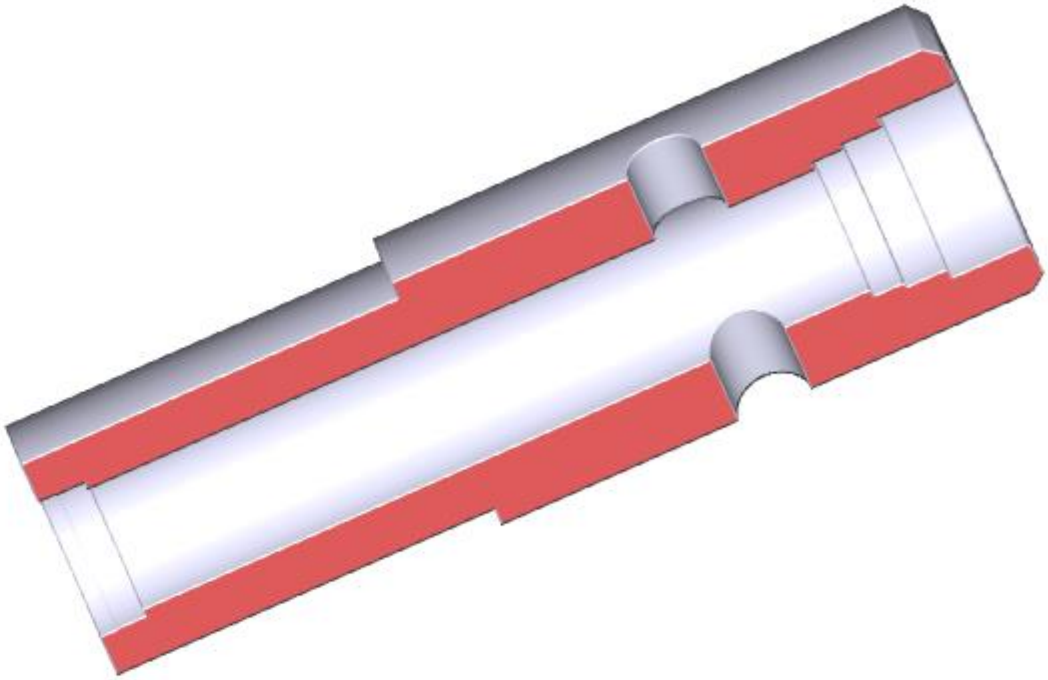
We recommend to create section tools with certain excess to all directions. If you enlarge sectioned solid by shape editing, section may be compromised – the section plane is not intact. In such case, select editing of section tool and enlarge it too.

 **3D Section Management - SEM**

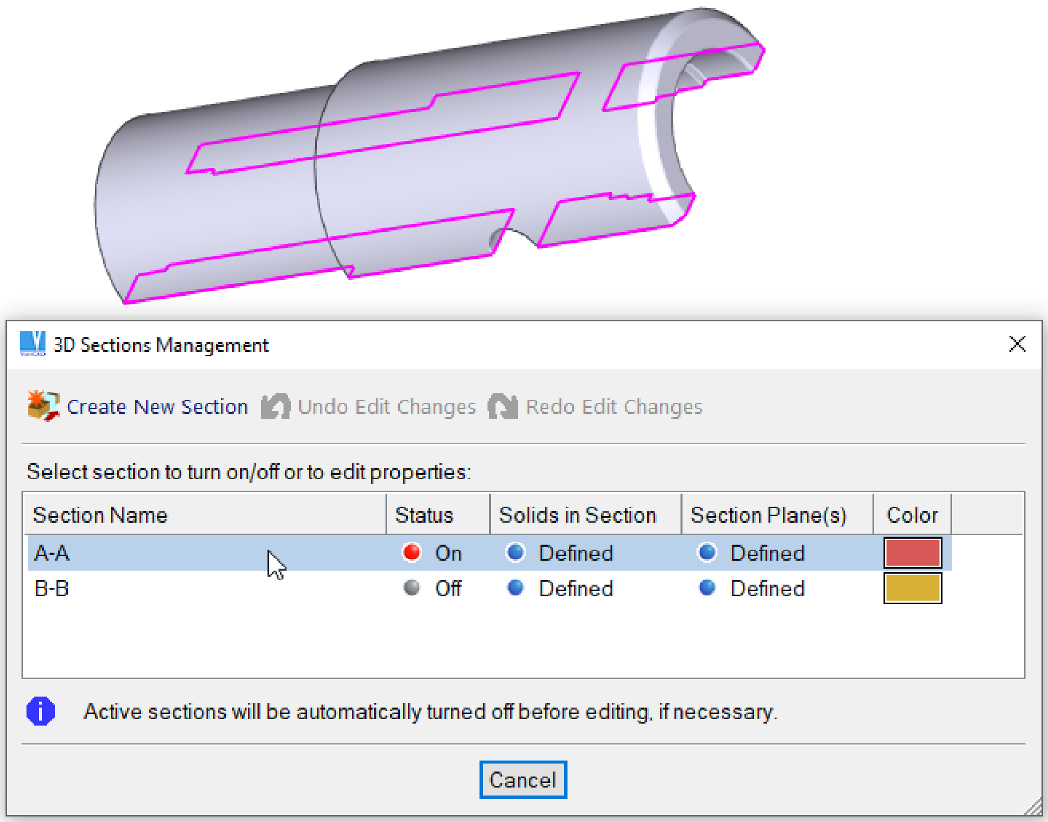
Turns on or turns off selected section. Allows you to define a new section, define the sectioned solids, section planes or cancel definition of 3D section.



3D section - the box is the sectioning tool



Sectioning result, the view is rotated



3D sections management. If a line in the table is highlighted under cursor, corresponding section contours are highlighted in 3D.

3D Comprehensive Shapes

Creating and Editing 3D Texts

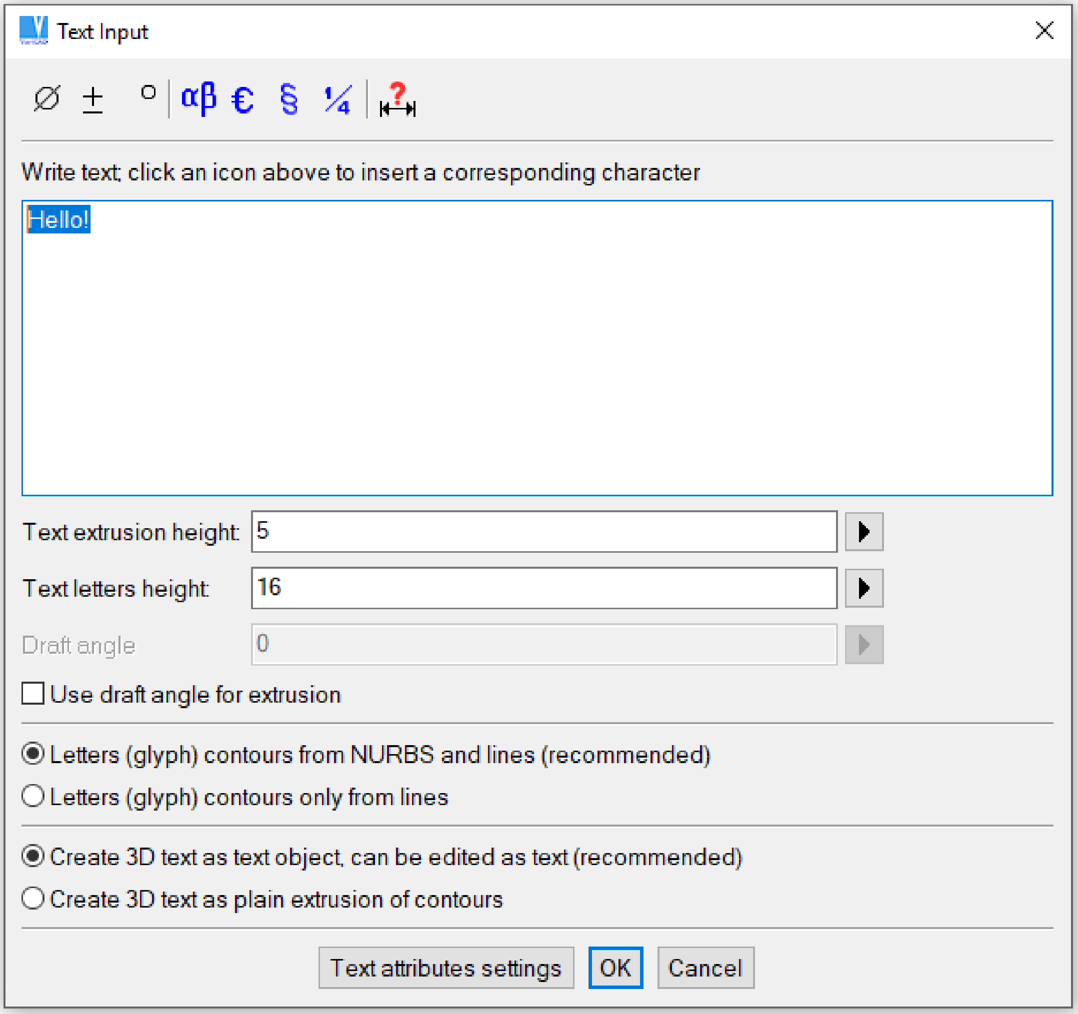
3D texts extruded into space can be defined and edited the same way as 2D texts. You can write text, select text height or text font. For 3D, extrusion height and draft angle is defined. Text font is exploded into 2D contours and these contours are then extruded. Optionally, you can explode fonts into lines and NURBS curves, or only into set of short lines. Also, you can create set of single extruded solids instead of a complex text object. 3D text objects can be added to or cut from any selected solids.

 **Create 3D Text – TXT3D**

This command creates 3D texts.

To edit an existing 3D text object, right-click any letter and select 3D text editing from pop-up menu. Edit of the entire text object rebuilds all text letters in one step. It is always possible to edit contours of letters individually (as for any extruded solids), and to change their locations individually. Moreover, a 3D text can be created as plain extrusion with no further connections to original text value.


Edit 3D Text - this option is available, if you click a letter created as a part of 3D text object. Then, you can edit the entire text at one step.



3D text definition

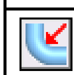
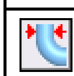
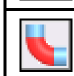
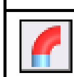
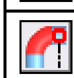
An example of 3D text

Pipes and Wires

Pipes or wires are created as a set of cylindrical segments and elbows. Define diameters and the elbow radius first. Then define a path of the pipe or wire in space. To define the path, you can use similar tools as for solids insertions or translations. Location of the tri-axes tool defines a location of the pipe segment endpoint. Before each confirmation of the location, you can easily redefine tri-axes tool position or adjust this position according to other spatial objects. See *Transforming and Copying Solids (page 229)* for basic location modes.

You can define straight segments, while each connection is automatically rounded by an elbow. You can insert an elbow directly at the finished straight segment and define its rotation. You can also bend a pipe around a corner in space as well as around or according to another pipe's elbow. You can finish the pipe selecting an axis (for instance flange's axis or hole's axis) - an elbow and straight segment is created. The straight segment is finished right at the selected axis. The next location selected anywhere at the selected axis allows you to create a pipe straight into the desired location under the desired direction.

For definition of pipes or wires in space, you have more additional options available than for locations of solids:

Icon	Use
	Elbow radius redefinition
	Diameter redefinition
	Creates elbows between straight segments automatically, segments are defined by endpoints
	Creates single elbow defined by start tangent and point
	Creates elbow and straight segment to intersect selected axis

	Locate at intersection of two selected axes
	Pipe/wire segment endpoint at current location

Clicking the inner part of the tri-axes tool, you can obtain more options for corresponding axis than for locations of solids:

Icon	Use
	Dynamic rotation around selected axis, reference point at end of X axis
	Dynamic rotation around selected axis, reference point at end of Y axis
	Dynamic rotation around selected axis, reference point at end of Z axis
	Bend pipe around a corner, start at direction of selected axis
	Bend pipe around another elbow or axes intersection, start at direction of selected axis

While defining a path of a pipe or wire, Enter or right-click has different meanings according to the given situation. If a segment is inserted and no new location of tri-axes is defined, press Enter (or right-click) causes the pipe creation is finished. All previously defined segments are merged into a single pipe or wire. If a segment was created and a new tri-axes location is already defined, press Enter (or right-click) defines new segment's endpoint. VariCAD uses different cursors to distinguish each situation:

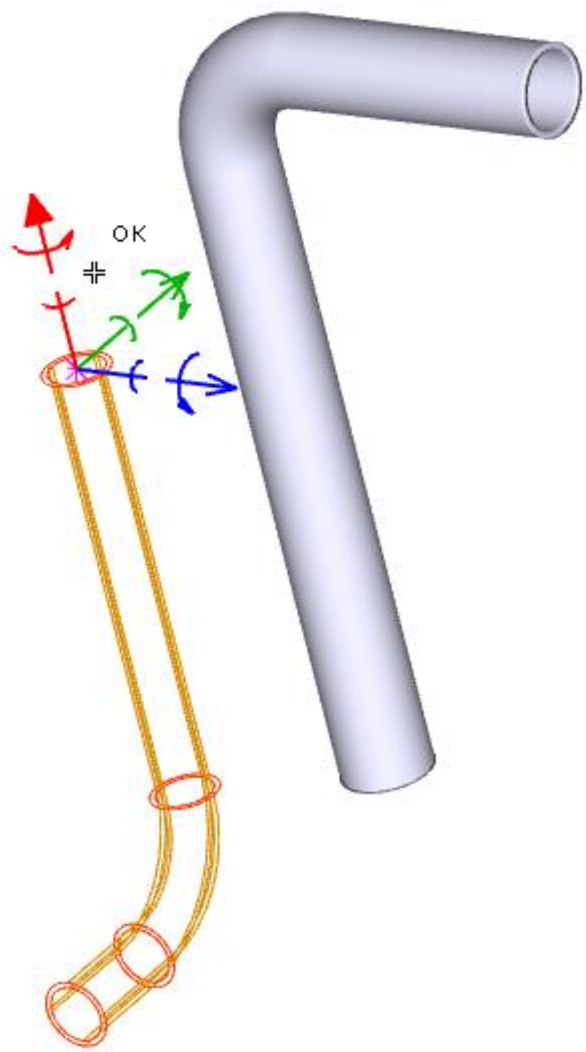
Cursor	Use of Enter	Use of Step Back
	End point of segment is defined	If a segment was created before, tri-axes are located back at the endpoint of this segment. Otherwise back to geometry confirmation
OK	Pipe or wire is finished	Last segment is removed

Pipes - PIPES

Creates pipes in space. Radial intersection is ring – defined by outer and inner diameter.

Wires - WIRES

Creates wires in space. Radial intersection is circle – defined by diameter.



Creation of pipe

Sweeping of 2D Profiles

Sweeping of 2D profiles is similar as creation of pipes or wires, except that you sweep a closed contour created as a 2D profile. At first step, create a 2D profile in sketching plane – similarly as a profile for extrusion, for example – see *Sketching of a 2D Solid Profile (page 171)*. Then, define a 3D path the same way as for pipes or wires.

 **Sweep 2D Profile along 3D Path - SWP**

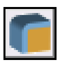
Offset Patches – Thick Shells

Shell is created as the offset patches are connected to the selected patches at a given thickness. Select patches on a solid first. Then define a thickness. You can select whether the resulting shell is created as outer layer to the selected patches (creation in a direction of the normal), or the selected patches are outer layer of resulting shell (creation against a direction of the normal). You can select also if the original solid remains in space or is removed after the shell creation. In both cases, copy of the shell pattern is stored in the shell object and remains available for further shell shape editing.

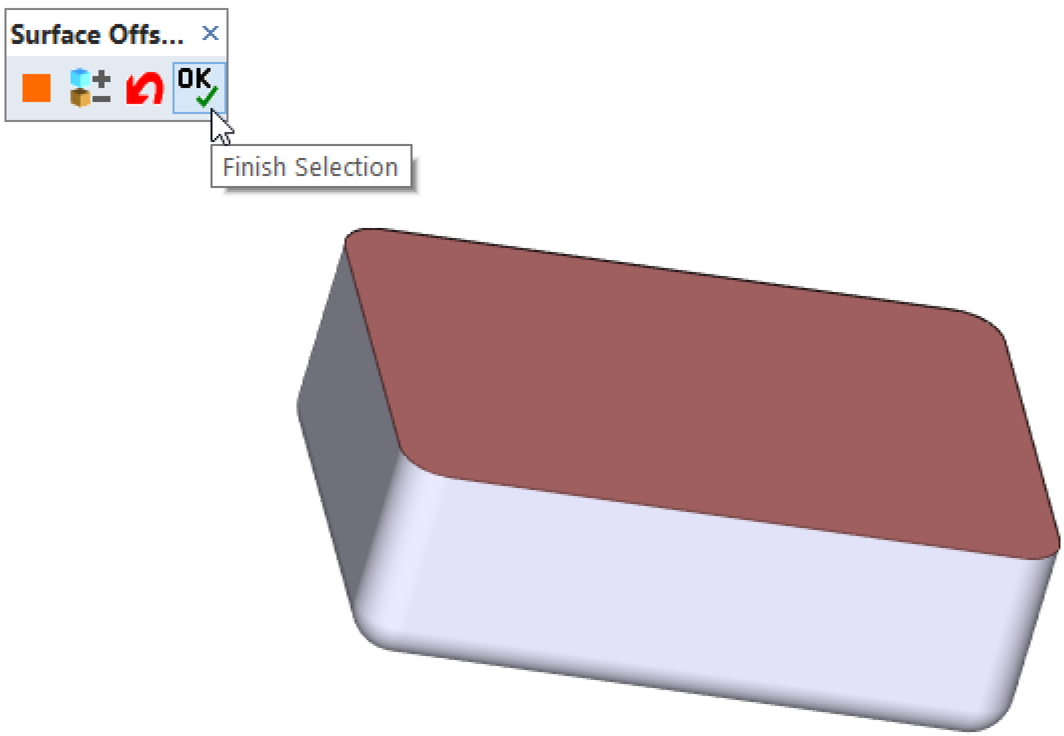
Selecting patches for shell definition, you can select patch by patch or use the following option:


All solid’s patches are selected. Then you can some of them deselect.

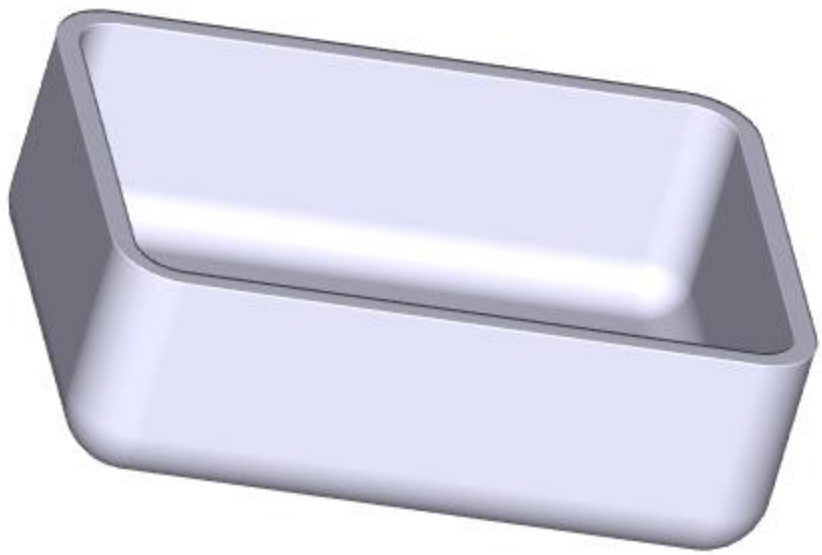
Shell objects can be used as sheet metal parts.


Offset Patches (Shells) - OFP

This function creates shell objects.



Creation of shell



Shell defined in previous example

Threads in 3D

Threads can be created on an existing outer cylindrical surface, as a threaded hole or by inserting a threaded cylinder (a threaded end of shaft, for instance). Once created, the threads are properly exported into 2D drawing area or into STEP files. Checking interferences, the threads are correctly distinguished not only according to their diameters, but also according to their pitches and types. If a solid which contains threads is to be rescaled, scaling values are limited to available standard thread diameters.

Parts inserted from mechanical parts libraries are correctly fitted with threads, too. The threads are present at screws or nuts inserted from the libraries. Screws and nuts created in versions prior to VariCAD 2008 are not automatically changed to objects fitted with threads.

If a threaded surface is exported into 2D drawing area and if you perform dimensioning, dimensions of threads contain corresponding texts automatically – see *Thread dimensions (page 84) (page 27)*.

If you create a threaded hole or outer threaded cylinder (a screw), you can select a thread from a list of threads:

- Metric Threads, for work with ISO units (millimeters)
- Unified Screw Threads, for work with imperial units (inches)
- Pipe Threads, ISO 228

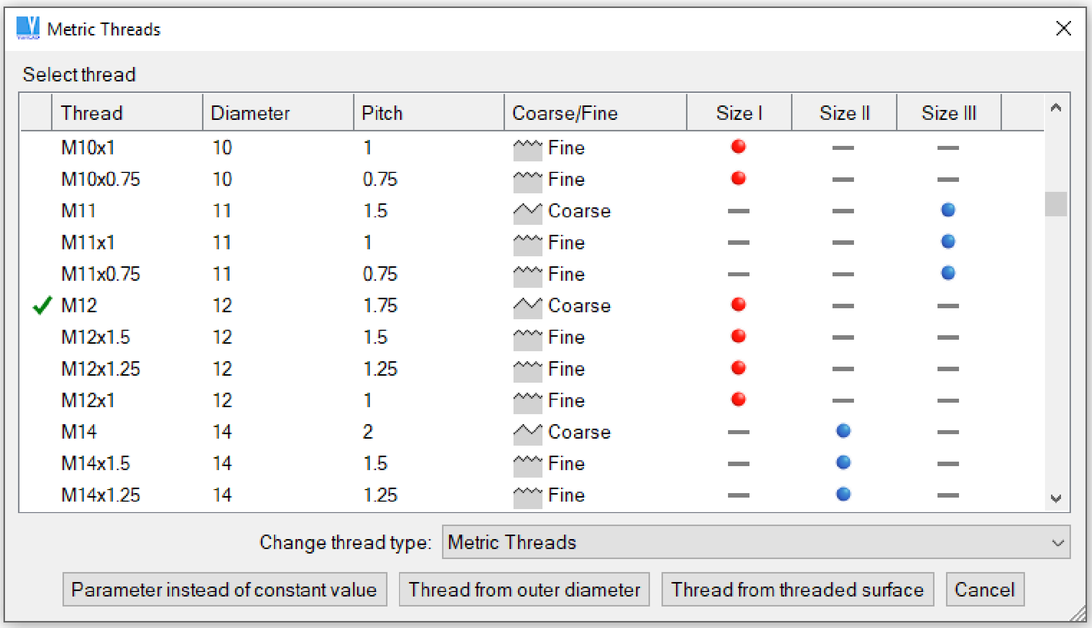
Functions available for thread creation:

 **Threaded Hole - THH**

Threaded Cylinder (Screw) - THS

Outer Thread Cutting Tool - OTC

Outer thread can be cut only at a diameter equal to a standard thread diameter.



Selecting a thread from a list of threads

Checking Functions and Calculations

Units of Calculation Results

Results units' settings can be done from command "CFG". You can select, whether the results of volume, mass, moment of inertia and surface area calculation will be displayed in:

- ISO units (meters, kilograms)
- Other units (inches, feet, pounds)
- Both unit systems.

Volume, Mass, Surface and Moment of Inertia Calculations

Volume, Mass Center of Gravity - VOL

Select the objects. The calculation provides object volume and coordinates of the center of gravity. You can then select the mass density from the list of materials, or enter the mass density manually, to recalculate the mass. If multiple objects are selected, the calculation incorporates all objects into the results.

Surface Area - SAR

Select the objects for surface area calculation. If multiple objects are selected, the calculation incorporates all objects into the results.

Surface Area of Selected Patches - SELSAR

The command calculates surface area of selected patches.

Moment of Inertia - MIN

Define the axis about which the inertia is calculated and select the objects. The moment of inertia is calculated, and you can select or enter the mass density to change calculated result. If multiple objects are selected, the calculation incorporates all objects into the results.

Checking and Measuring Geometry

Some of these functions require location input. For more information, see *Defining 3D Locations (page 241)*.

3D Coordinates - 3DCO

Displays 3D coordinates of a selected point.

3D Distance - 3DD

Displays the distance between two selected points and DX, DY, DZ between the points.

Distance Point Plane - DPP

Measures the shortest distance between a point and a plane.

Distance Point Cylinder - DPC

Measures the shortest distance between a point and a cylindrical surface, as well as the radius of the cylinder and distance to cylinder axis.

Cylinder Dimensions - SCY

Displays dimensions of a cylindrical surface.

 **Angle between Planes - APL**

Measures the angle between two planes.

 **3D Object Information - ODT3**

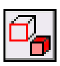
Displays information about a selected 3D object, including name and attributes, membership in solid groups and identical copy groups, and definition of any sections.

 **3D Space Information - STAT**

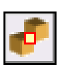
Displays information about all objects in the 3D space, including number of solids, number of blanked solids, defined sections, assembly links and defined groups.

Interferences among Solids

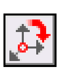
When inserting parts into an assembly, you may insert some parts incorrectly. This could cause solids to overlap other solids. Interference checking enables you to check for situations like this. After each test, the result of interference checking is displayed. Intersection curves of colliding objects are always displayed. The following options enable you to display more information:

 **Highlight all interferences together**

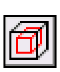
This function is useful at the beginning of interferences solving. All objects are displayed as wire-framed and in no expressive color. Intersection curves, identical objects or objects engulfing smaller ones are highlighted. Thus, you can easily see any interference even within large assembly.

 **Highlight solids in selected interference**

If the intersection curve is selected, corresponding solids are highlighted.

 **View rotation center to interference**

Function allows you to move the view rotation center to the center of gravity of the selected intersection curve. This is useful especially when the particular interference is solved and you need to change views.

 **Solid engulfing smaller solids**

If any small solid is completely engulfed by a larger solid, the larger object is highlighted and smaller object is displayed as wire-framed within the larger one.

Identical solids

This option highlights two or more identical objects at the same location. Such identical objects can occur especially during blanking/unblanking.

Finish interference displaying

You can check interferences using these functions:

Interference between Groups - CRT

The interference check is done between two groups of solids. The groups remain defined, enabling you to repeat the check after editing.

Repeat Interference Check - CRTR

Repeats the interference check between the previously defined groups.

All Interferences - ASCH

Checks for interference between all solids.

Interference without Selected - ASCHN

Checks for interference between all solids except for those selected.

Interference Selected vs. Rest - ASCHS

Checks for interference between selected solids and the rest of the solids.

Interference within Selected - ASCHB

Checks for interference only within selected solids.

Display Interferences - CHRD

Displays previously calculated intersections among solids. You can calculate these intersections once, and display the results repeatedly. If the objects in interferences are removed or blanked, corresponding interference is not displayed. If they are unblanked or if removing is undone, then interference is displayed again. On the other side, no edit changes affect displaying the interference. It is recommended to recalculate interferences often, if the objects are edited.

3D Assemblies

VariCAD enables you to define connections between part files, sub-assembly files and assembly files. Parts can be created and edited in their own files and then inserted into assemblies. If a part is edited, the change is transferred to all assemblies that contain the part. In addition, you can change a part within the assembly file and update the original part file, as well as other assemblies that contain the part. It is generally faster and more efficient to edit parts in their own files, since there is much less data in these files. But editing within the assembly can be handy if you need to edit only a small detail, especially a detail that affects other parts.

If a current file contains any parts inserted from other files, the file becomes an assembly file, and "Assembly" is displayed on the right side of the Status Bar. If the file contains a part used in an assembly, or an object identified as a part, "Part" appears on the Status Bar. If the current file is a sub-assembly, “Subassembly” is displayed. If the current file is assembly and contains also defined sub-assembly group, “Assembly+Sub.” is displayed.

Using links between parts and assemblies provides many advantages. However, you do not have to use links when working with multiple solids. Unlike other CAD systems, VariCAD provides freedom and flexibility when working with assemblies.

Assembly status is displayed in file-dialog and in recent files (history) panel, above the file preview image. This information is corresponding to highlighted item in list of files.

Creating Part Files, Assembly Files and Assembly Links

There are several ways to create assembly-part links:

- In the current file, select an object and use the Create Link from Part function. Define the filename for the part, and the object is saved to this file. The current file becomes the assembly file
- In the current file, select an object and use the Create Link to Assembly (Assemblies) function to "mark" the part to be used later in an assembly. Only one object in a file can be identified as a part.
- Make the current file as assembly by using File / Insert Objects from File and insert a file containing the defined part.

Saving and Loading the Assembly Files

While editing an assembly file, changes can be saved to part files as well as to the current assembly file. When opening an assembly file, the parts are loaded as they were last saved then the parts are updated from the part files. Therefore, if parts are changed after the assembly was saved, the changed parts will be used.

If part files cannot be located, a list of broken assembly links is displayed and you can use the following methods to resolve them:

- Leave the part as it was saved in assembly, and you can resolve links later. If you know the part file has not changed since the assembly was closed, this is the best option.
- Break the link permanently.
- Search for another file or directory. If there are parts in another directory, you can select one of them as a replacement. The new directory is identified, and you can change all links to use this path.

If you choose not to resolve any links, the parts will remain as they were saved last in the assembly.

Sub-assemblies

Instead of a single part inserted into assemblies, you can also define a group of solids as a sub-assembly. The sub-assembly objects are single separate objects in a sub-assembly file. When inserted into an assembly, they behave as one compact object.

The group of sub-assembly objects may contain also parts inserted from part files, or other sub-assemblies inserted from sub-assembly files. On the other side, in assembly file the sub-assembly may be also a member of sub-assembly group ready for insertion into higher level. This allows you to create a hierarchical structure of entire product.

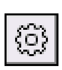
The structure of assembly and sub-assemblies is displayed in BOM, see *BOM Objects (page 305) (page 298)*.

In many ways, sub-assemblies behave similarly as parts inserted into assemblies. But there are also significant differences:

- Unlike part inserted from a part-file, the sub-assemblies cannot be modified in assemblies. To modify a sub-assembly, right-click it and from menu, select “Open as part file”. Then perform changes at the separate sub-assembly file.
- Sub-assemblies are created from multiple objects. You can add objects into a sub-assembly group, delete objects from the group or change their mutual positions.
- When a part is inserted into assemblies, transformation uses the part’s insertion axes. However, a sub-assembly contains multiple objects. The insertion axes are defined separately for entire sub-assembly, in an extra step.
- A part may have defined its own attributes, like name, material etc. These attributes are defined for a single solid, both in part file and assembly file. Contrary to this, attributes of a sub-assembly in assembly file are data defined in the corresponding sub-assembly file as file attributes – see *Assembly, Sub-assembly or Part Attributes, Fill Title Blocks (page 321) (page 298).*

Relative Paths in Assembly Links

By default, VariCAD uses absolute paths for all assembly to part or assembly to sub-assembly links. You can change the usage of absolute paths – in following command:

 **Complete System Settings - CFG**

Here, in section 3D, “Assembly link settings”, you can define how to solve changed paths, if an assembly is open:

- Changed paths are not solved – it means that only absolute paths are used. In case the part or sub-assembly is not found, you must solve the problem individually.
- Change of path is solved for the beginning of the path. This option can be used, if parts or sub-assemblies are stored at a server and if computers have differently mapped access to the server. For instance, the same path may begin with “J:\” at one computer, with “Z:\” at another computer and with “/mnt/” at third computer, using Linux.

- Complete changed path. This option is intended for situations when files are copied among different computers often, and different paths are used at each computer. In this case, parts or sub-assemblies must be saved in path under assembly or at the same level. If parts are used in multiple assemblies or sub-assemblies, they should be in one directory, “under” the path where assemblies are.

Paths are separated by “\” in Windows, and by “/” in UNIX. In assembly links used in VariCAD, separators are automatically handled according to current operating system. By other words – if assemblies and their parts are copied from Linux to Windows, path separators are changed automatically. If some computers in network use Windows and some use Linux, VariCAD assembly links are solved automatically, too.

Simultaneously Open Assembly and Part Files

Whenever you open and edit a part (detail) or sub-assembly, the file must be saved before you activate another file. Otherwise, any changes may be lost if you open simultaneously files linked within the current assembly structure.

Changed files can be saved automatically. By default, file save must be confirmed. If you do not wat to preserve changes, you have to close the file without save. Run command “CFG” to manage settings related to open assembly files.

 **Open Part File or Sub-assembly from Assembly - EDE**

Select a part or sub-assembly in assembly file, and its file is open.

More convenient method to open a part file or sub-assembly file is to right-click the solid and select “file open” from pop-up menu.

Definition of Assembly-Part links

 **Create Part, Save It into New Part File - DIA**

Defines a selected object as a part and exports it to its own file. The current file becomes the assembly, and the part-assembly link is established.

 **Define Part to be Inserted into Assembly - DEE**

Command defines the selected object as a part. The current document becomes a part file.

 **Change Definition of Part to be Inserted into Assembly - ROI**

If a file has a defined part, use this function to define a different part. Corresponding parts in all assemblies are changed according to this selection.

Definition of Sub-assembly - Part links

Create Sub-Assembly, Save It into New Sub-Assembly File -SBA

Defines selected objects as a sub-assembly and exports them into their own file. The current file becomes an assembly, and the sub-assembly - assembly link is established.

Define or Change Sub-Assembly to be Inserted into Assembly - SBE

Command defines the selected objects as a sub-assembly. The current document becomes a sub-assembly file.

Breaking Links between Part or Sub-assembly and Assembly

Break Link from Part or Sub-assembly - CSI

In current assembly, breaks the assembly link for one group defined by the selected solid.

Break Definition of Part to be Inserted into Assembly - CDE

In current part file, breaks the part definition. The file is no longer a part file.

Break Definition of Sub-Assembly to be Inserted into Assembly - CSB

In current sub-assembly file, break the sub-assembly definition. The file is no longer a sub-assembly file.

Break All Links from Parts or Sub-assemblies - CAI

Break all links in the current assembly. The current file loses its assembly status.

Editing Sub-assemblies or Parts in Assembly Environment

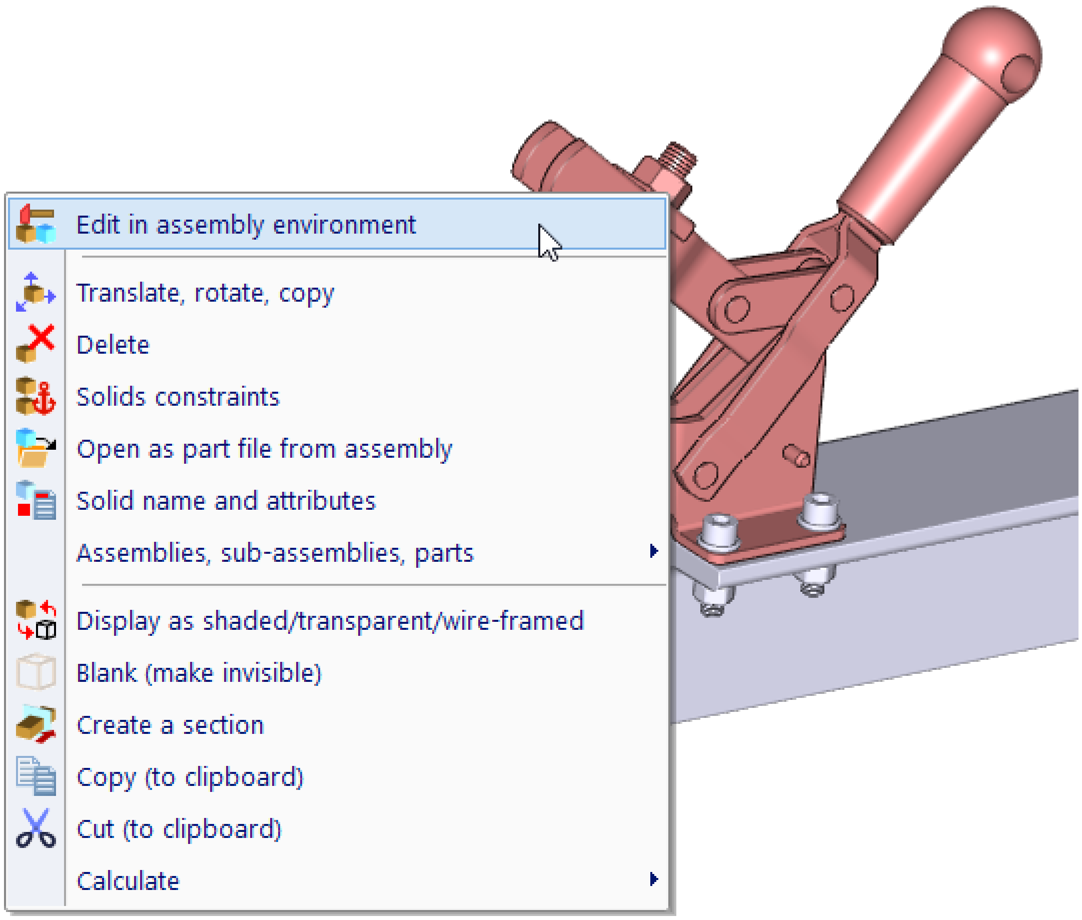
Parts or sub-assemblies can be edited separately in own files. However, you may often need to adjust shape of a part according to other parts in assembly, or rearrange locations of sub-assembly elements according to higher assembly. For these purposes, use editing in assembly environment.

When part or sub-assembly is edited, all assembly objects are copied into 3D space, where the part is edited. You may use the rest of assembly as tools for part modifying, you can measure values from them and you can modify them. Because these objects are mere copies, their changes are not transferred into actual assembly.

To distinguish copied parts from the edited part or sub-assembly solids, these parts are displayed in different pre-defined color.

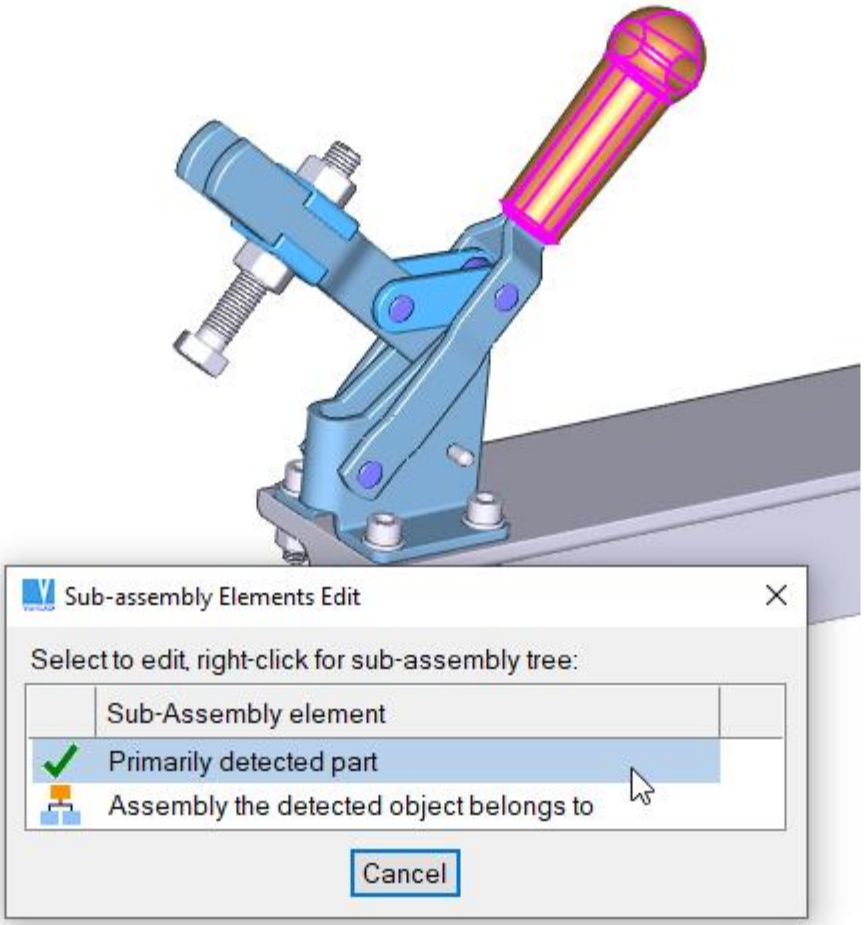
Selecting Parts or Sub-assemblies for Editing

To select a part or sub-assembly for edit, right-click it and select editing in assembly environment from menu. This method selects only parts or sub-assemblies inserted into highest assembly level.

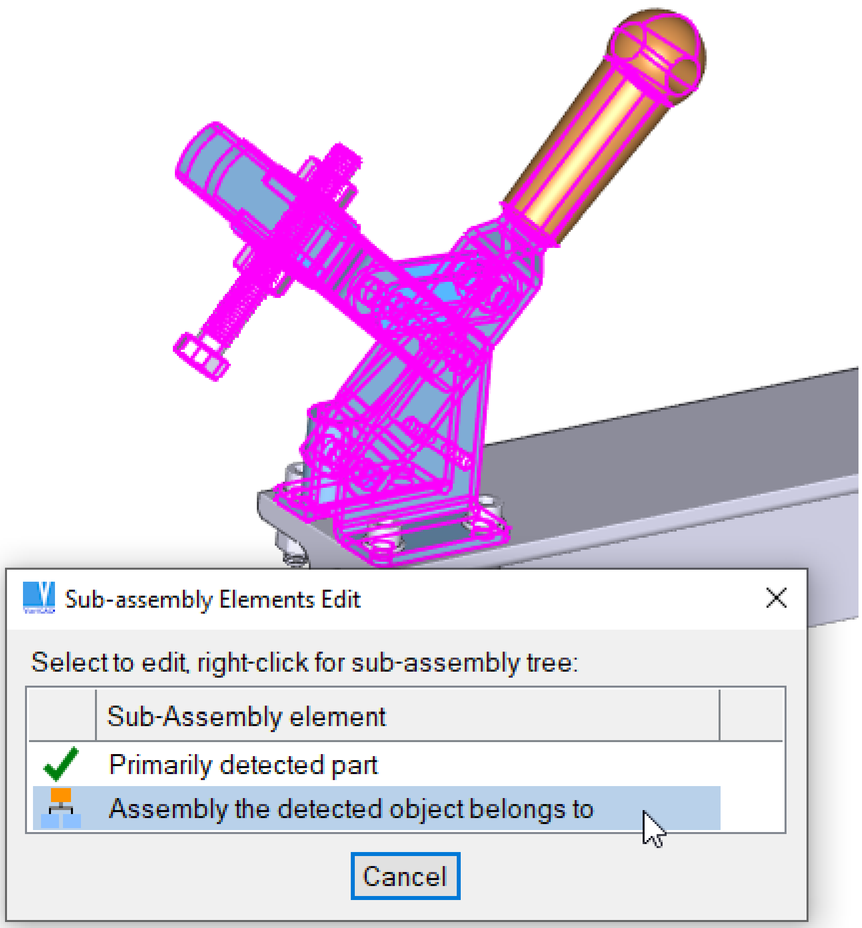


Pop-up displayed after clicking a sub-assembly or part

If you need to select objects from lower levels, press and hold Ctrl key and move cursor over parts in sub-assembly. Single parts from any level are detected. After right-click, a menu appears and you can select level the edited objects are from.



Selecting primarily detected single part



Selecting higher level, a sub-assembly the detected part belongs to.

Parts or sub-assemblies can be selected for editing in assembly environment, if you run assembly tree scheme. See *Part Editing or Attributes Definition Mode (page 291)*. The object is selected from list.

Edit Part or Sub-Assembly in Assembly Environment - EDIA

Calling this command is the last method of part selection. Edited object is selected by cursor.

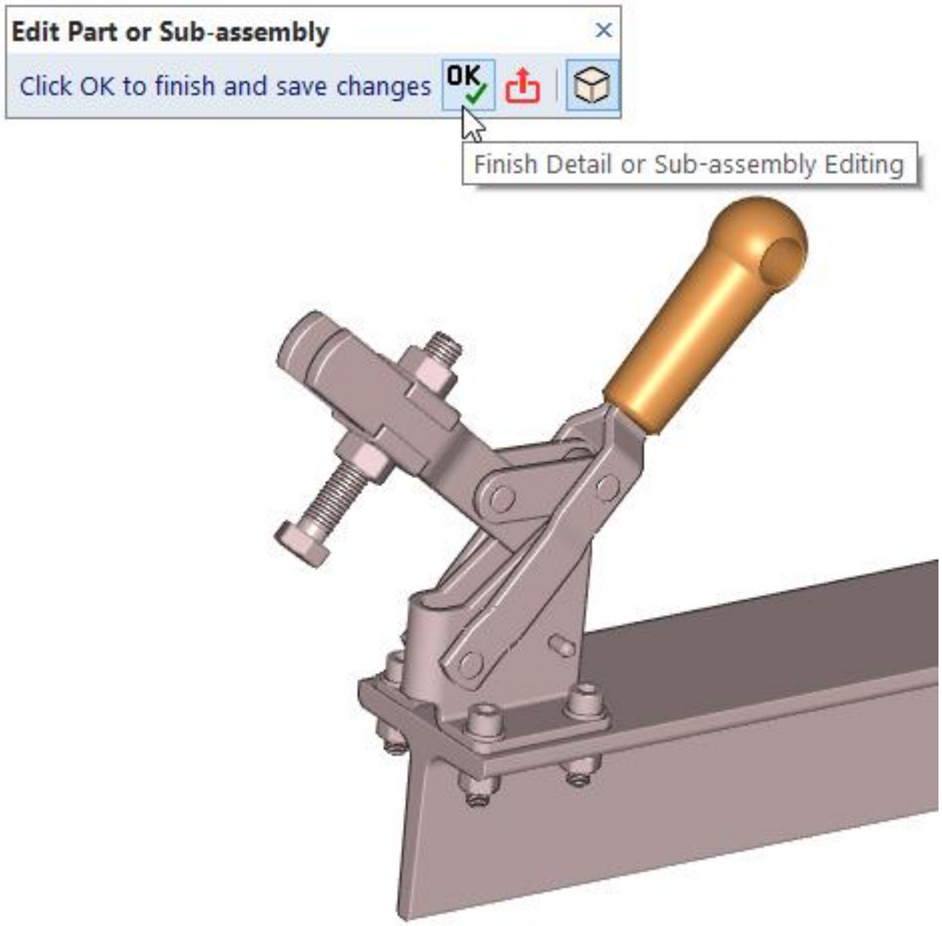
Editing Parts or Sub-assemblies

While editing parts or sub-assemblies, a toolbar is displayed. Following options are available:

Finish editing and return back to assembly. Edited object is saved and updated in assembly.

Return back to assembly. If objects were changed, all changes are discarded.

Change color of the rest of assembly objects. They can be displayed in predefined color or in their actual colors. We recommend to use display in predefined color.

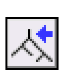


Editing part - the rest of assembly is displayed in distinctive color.

3D Assembly Tree Scheme

3D Assembly Tree Scheme Window

The assembly tree scheme allows you to select objects for standard commands, or to select single members from any level of assembly tree to change their names or attributes. Assembly tree window is open at a secondary monitor, if VariCAD runs at two monitor settings.



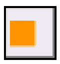
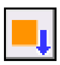
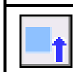
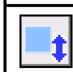
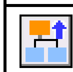
Assembly Tree Scheme - ASTR

This command opens the assembly tree window. It can be called by right-click an empty area and then from pop-up, or you can click a corresponding icon in tool-bar.

Each solid in 3D space is displayed in assembly tree scheme. The scheme contains a list sorted by solids' names and by assembly tree structures. If a solid has no name defined, a temporary name is assigned (like Part 1 – numbered part).

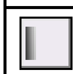
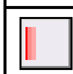
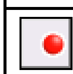
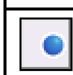
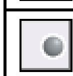
If you move cursor over lines in the scheme and a line is highlighted, corresponding solids in 3D are highlighted too.

First column contains the tree scheme. If you move cursor over the icons in this column, tool-tip containing corresponding file appears. Icons distinguish solid's position within assembly tree:

Icon	Type of object within assembly tree
	Current assembly
	Plain part, no assembly links
	Supplementary object (paint, oil…)
	Sub-assembly
	Part imported from part-file
	Part, member of sub-assembly group
	Part, imported from part-file and member of sub-assembly group
	Sub-assembly, member of sub-assembly group

The sub-assembly group is a group of objects defining the sub-assembly in the current file. This group is sub-assembly available for import into higher assemblies.

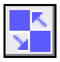
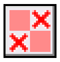
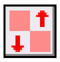
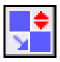
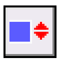
Second column contains file related information:

Icon	Corresponding file or solid information
	Corresponding file is open
	Corresponding file not found, cannot be open
	Solid is selected
	Attributes of solid at current level are changed
	Part or sub-assembly attributes are changed, will be saved

These symbols may be combined together.

Third column contains names of solids. If cursor hovers above a solid name, tool-tip with all solid attributes appears.

The last column contains number of solids and also icons related to same names or identical groups:

Icon	Name or solid group related information
	Identical copies
	Multiple occurrence of the same name for plain copies
	Library parts
	Plain solid has the same name as adjacent inserted parts
	Plain solid has the same name as adjacent identical copy
	Identical copy has the same name as adjacent plain solid
	Identical copies, part or sub-assembly group

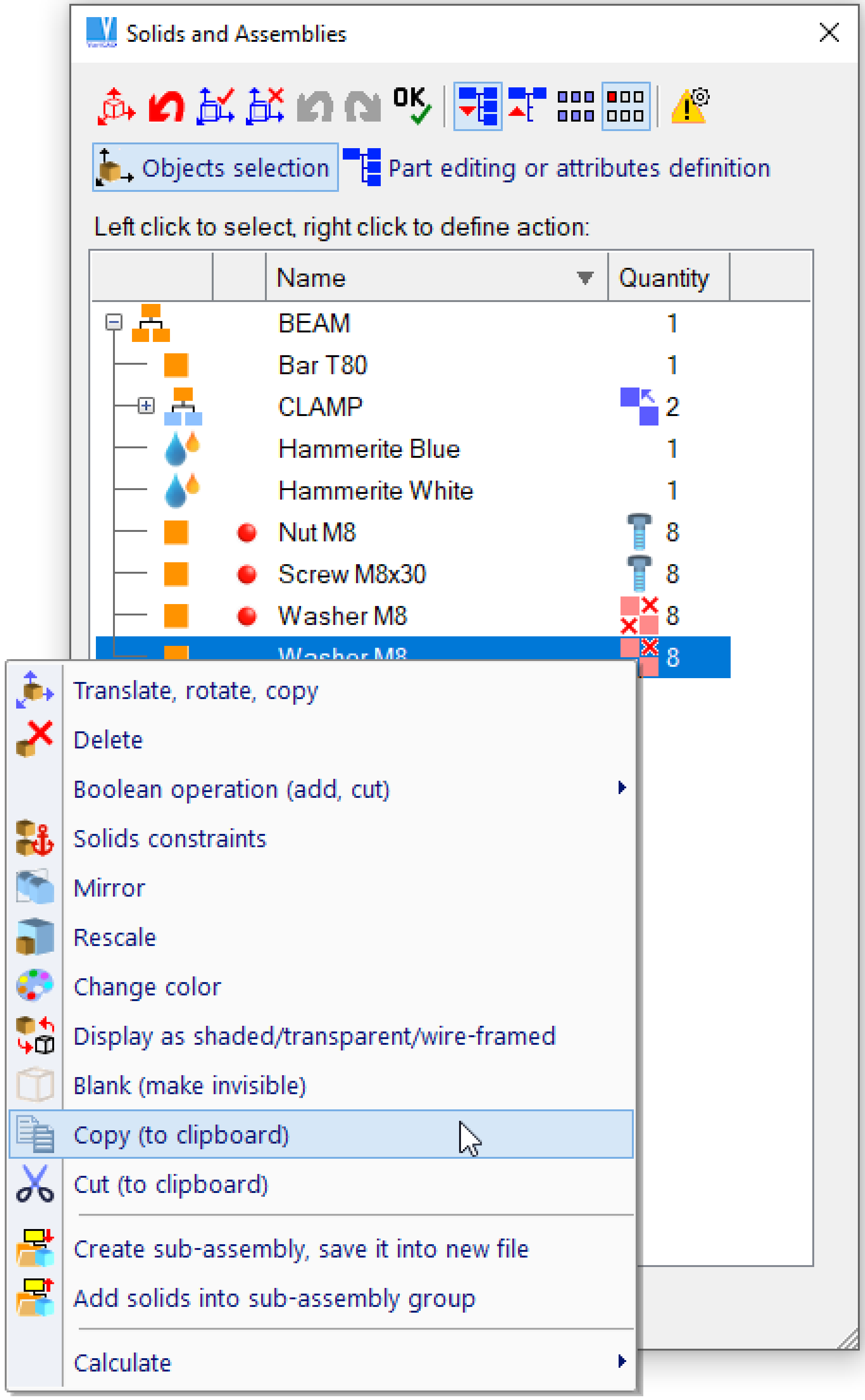
If solids have the same name and if they are not part of an identical group, BOM may contain erroneous data. Solids are sorted and counted according to their names. If they are not in identical group, they may be changed individually. Consequently, parts with different shapes may occur under the same name.

You may create identical group from selected group of solids – see *Add Solids to Identical Copies (page 240)*

Objects Select Mode

Objects select mode allows you to select objects for further edit commands, similarly as if you click objects in 3D space and after right-click, select a command from pop-up menu. You can select solids only at the highest assembly level for editing.

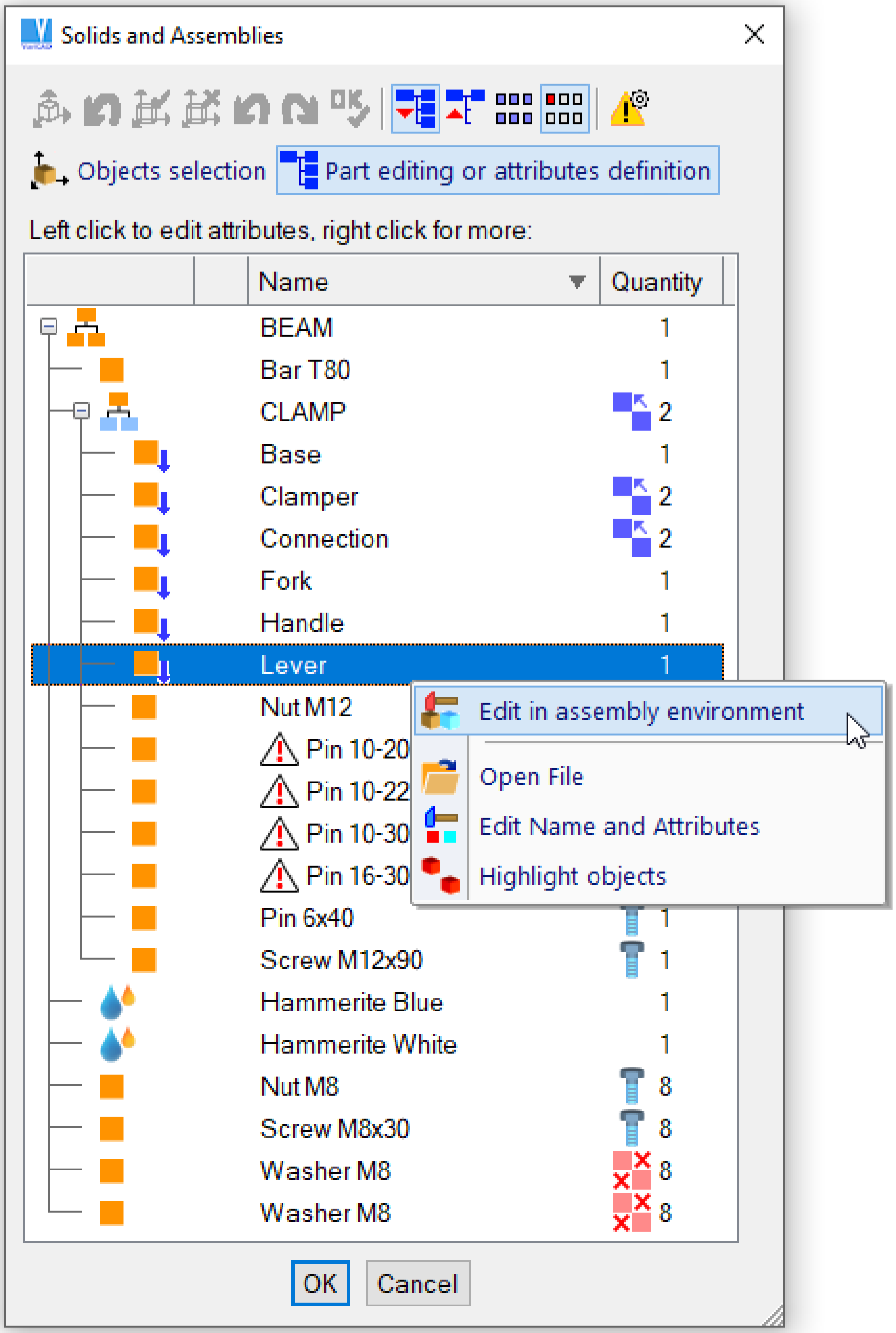
You may left-click multiple lines and finish selection by right-click, or you may right-click a single line for editing a single corresponding item.



Assembly tree structure, objects select mode

Part Editing or Attributes Definition Mode

Left-click starts definition of name or attribute of selected object. Right-click opens a pop-up menu, see image below. In this example, next step would be editing of the corresponding part.



Assembly tree structure, part editing or attributes select mode.

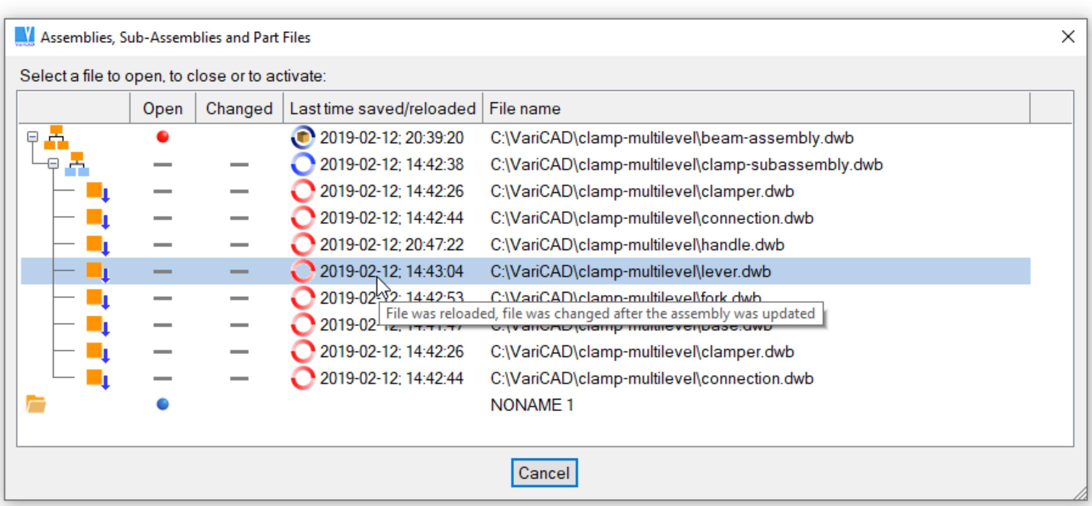
Managing All Assembly Tree Files

Assembly Tree Files - OATR

This command opens a window containing assembly structure of all files related to currently open files. You can open any sub-assembly or a part file if you click a corresponding line in the table. Mainly, the file structure listing contains dates of last changes of all related parts or sub-assemblies. The command is available from pull-down menu Window.

If you move cursor over icons in the scheme, a tool-tip with additional information appears.

While open an assembly or sub-assembly, files from lower levels are reloaded. If date of their change is older than date of change of the assembly, reloading may be optionally skipped.



Assembly tree files

Surface Development (Unbending)

Surface Development enables you to create an unbent (flattened) version of a 3D surface and transfer it to a 2D drawing. This enables you to represent parts created from sheet metal. Moreover, you can create these parts as filled objects, if you only need to obtain a developed surface. Each bent surface can be created as an outer or inner shell.

You can develop only surfaces through which lines can be laid, such as cylindrical or conical surfaces. Planes can also be developed, but they are displayed with no change in 2D. You can select more than one surface to develop. For multiple surfaces, the function resolves connections of developed patches.

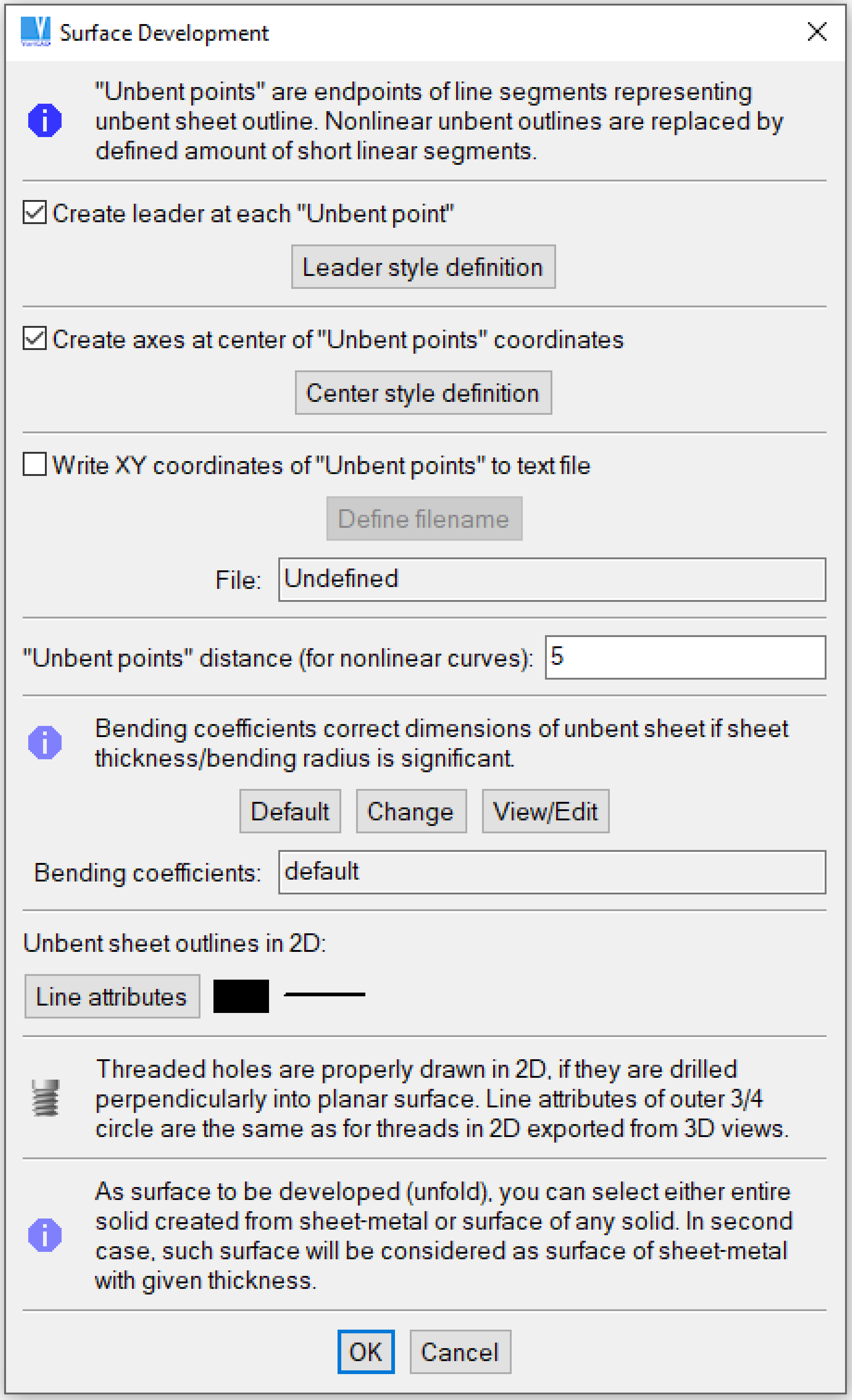
These unbent objects are created in the 2D drawing at a 1:1 scale, and can therefore be used as template, if 2D drawings are plotted. Surface outlines are lines or curves approximated into short line segments. The XY coordinates of outline points can be saved to a text file. Leaders can also be placed at outline points.

If the sheet is thin enough, you can ignore its thickness. Otherwise, the thickness must be defined and its value is incorporated while calculating the surface development.

Surface Development - SDE

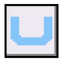
The following properties must be defined before selecting the surfaces to develop:

- Whether the text file with outline points will be created
- Whether the origin and leaders will be created in 2D development
- Material thickness (if undefined, zero is used)
- Line attributes used in the 2D drawing



Surface development window

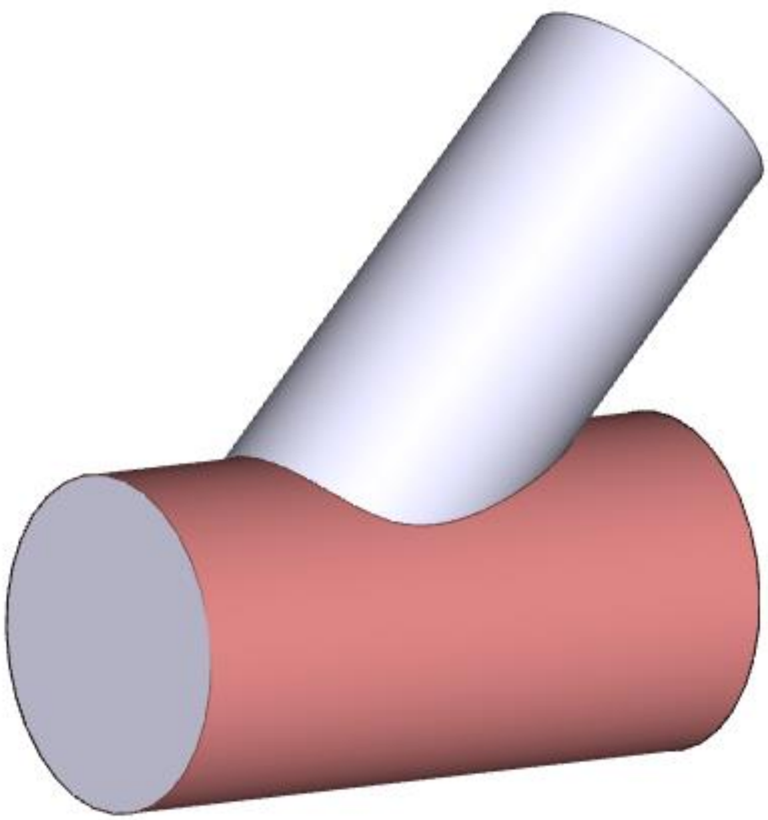
After defining these properties, select the surfaces to be developed. Press “Enter” or right-click to finish surface selection. The following are additional options:


Select the entire solid created from a sheet metal. This option is available only if no other patches are selected. If the entire solid is selected, you can deselect some of its patches, if necessary.

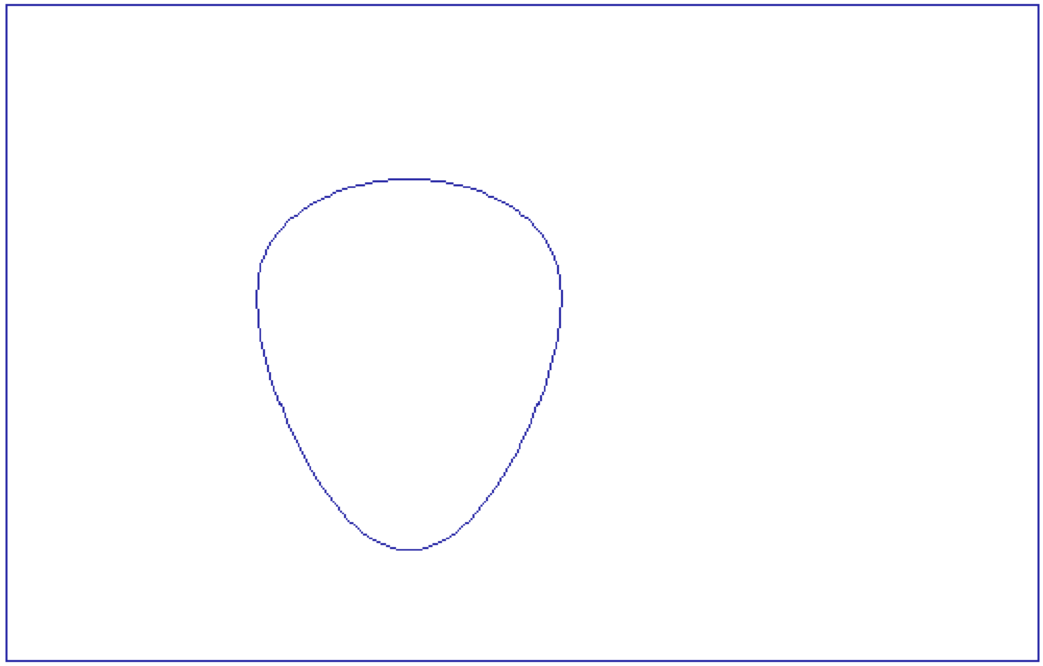
Other available options allow you to switch between select and deselect mode, undo previous selection or finish selection and perform unbending.

After selecting surfaces, define the material thickness. If the selected surfaces are surfaces of a sheet metal, the thickness is calculated automatically and you can confirm its value. Otherwise, define whether the thickness is significant. If yes, then define thickness value and select if the surface is an outer surface of the sheet metal or if the sheet metal is an additional layer of the surface.

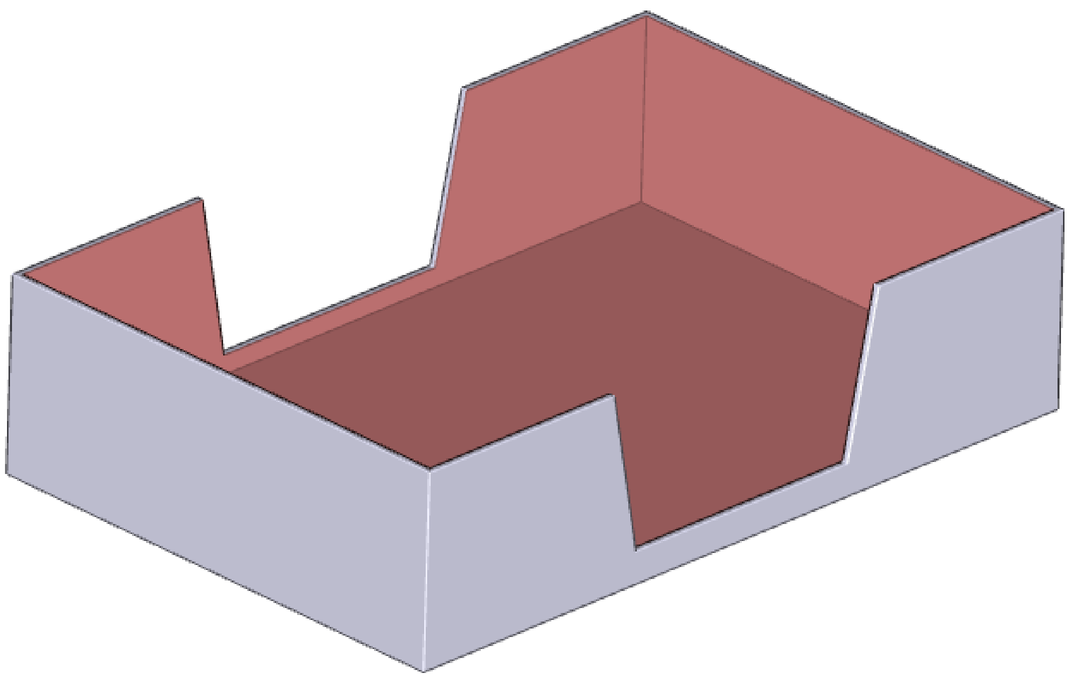
The final step is to drag to insert the unbent surface in the 2D drawing. If necessary, define the surface origin and leader positions.



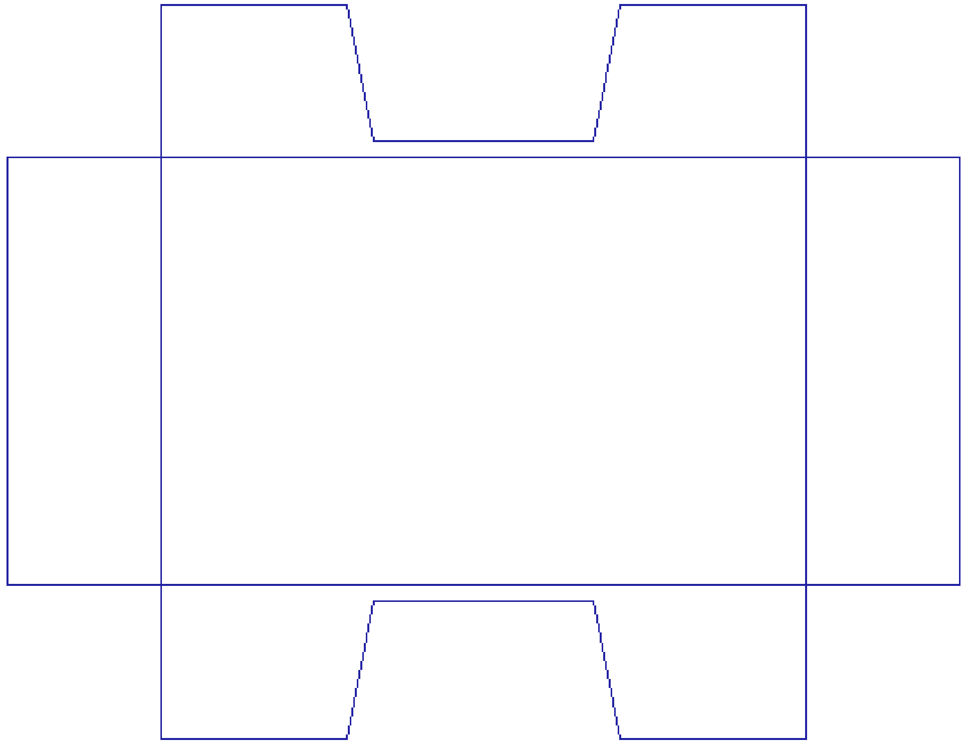
Selection of surface to be developed (unbent)



2D display of developed surface



Selection of multiple surfaces



2D display of developed surfaces

Chapter 14. Bill of Material, Object Attributes and Title Blocks

This section describes how to work with BOM's, attributes of 3D objects and assemblies, and methods of how to create a list of parts, automatically fill title blocks and how to manage other non-graphical data.

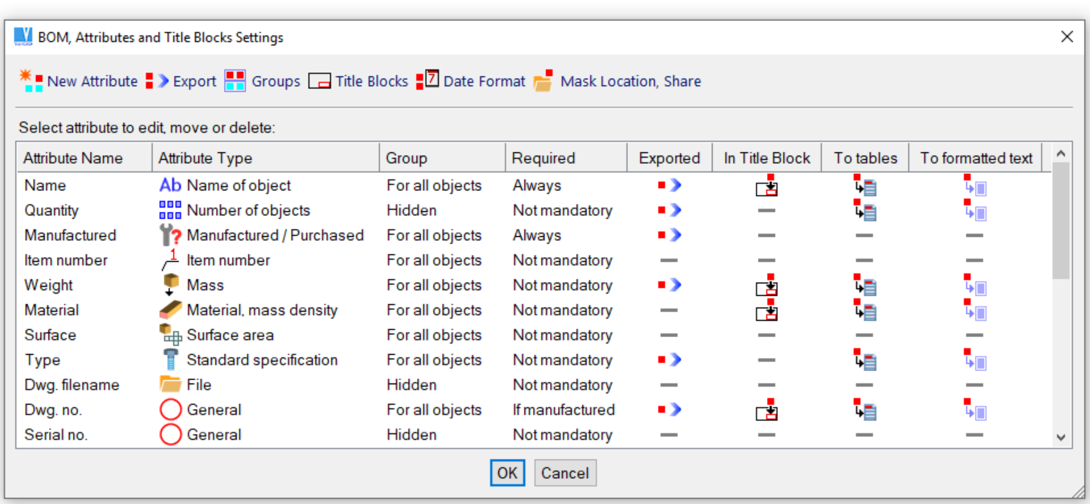
Object Attributes

Each solid or assembly can have a defined name and attributes. Mechanical parts such as screws, bearings, and rolled profiles have predefined attributes and names. Attributes and names can be used as a method of selecting solids (selecting solids according to their attributes). Attributes can be inserted into title blocks in 2D area, can be listed in files containing the list of assembly parts and can be exported into files suitable for other systems, like spreadsheets. The object name is, in fact, a sort of attribute.

BOM, Attributes and Title Blocks Mask

BOM, Attributes and Title Blocks Settings

This function allows you to define new attributes, edit or delete existing attributes, manage attribute groups, title blocks and create files suitable for other systems. Attributes and their properties are described below. All settings are stored in file called BOM mask.



BOM, Attributes and Title Blocks Settings (Mask)

Sharing BOM, Attributes and Title Blocks Settings (Mask)

Sharing of BOM mask (BOM settings) is available, if you click icon “Mask Location, Share” in upper tool-bar of BOM mask window. When you create a BOM mask suitable for all other users, then save it into selected network path. All users need to set the same network path for loading of the BOM mask.

Attribute Definition

Attribute Name

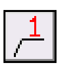
Each attribute has its own name. The name must be unique among all defined attributes. The name should represent the meaning of the attribute value. For instance, if the attribute defines material of a solid, then the attribute should be named “material” or similarly.

If the attributes are defined in different configurations, their names are used for recognition of compatible values – see *Compatibility of Defined Attributes and Attribute Groups (page 304)*

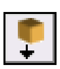
Attribute Type and Attribute Value

The type of attribute determines the method of attribute definition or behavior. Although each attribute may be defined as an attribute of “General” type, it is always better to consider the best type of given attribute. You may use the following types:

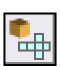
Name of Object – defines name for object recognition (for instance “Shaft 32-150” etc.). Defined names are usually demanded for the list of parts. If the name is not defined, the solid cannot be processed in BOM. Only one attribute of the type “Name of Object” can be defined among all available attributes.



Item Number – can be assigned automatically in BOM. If 2D drawing is created, item numbers of solids are automatically used as leader texts.

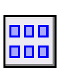


Mass – value is mass (weight) of the solid. If defined, VariCAD allows you to calculate mass of the solid using the same method as in the function “Volume, Mass, and Center of Gravity”. After calculation, you may select the result in various units (kg, g, lb., oz. etc...).



Surface Area – value is the surface area of the solid. If defined, VariCAD allows you to calculate surface area of solid using the same method as in the function “Surface Area”.

Manufactured / Purchased – value defines if the part is manufactured (documentation is created) or if it is purchased. According to this attribute, other attributes may be required as mandatory or may not be required at all. Manufactured objects may require other different attributes (like, for instance, drawing number) than the purchased ones (like, for instance, purchase code).



Number of Objects – for a single solid, the value is always equal to one. In BOM, the value is automatically counted as a sum of the same objects in the assembly. For the assembly, you may obtain the value as a sum of the number of objects of all parts.

Date – value contains a day of the month, month and year. You may configure the format of date representation, using the function *BOM, Attributes and Title Blocks Settings (page 298)*. You can select the same representation as in the operating system or you may define your own.

File – value contains an existing file, usually the file containing the corresponding part or the assembly file. You may select the file name from directory listing or you may choose it from the current file or file of a part defined in the assembly link.

Material – value contains material of the corresponding part. If defined, you may select from previously used values or you may copy the material from another solid.

Standard specification – value usually contains specification of standard, like DIN, ANSI etc. This attribute is automatically defined for solids inserted from mechanical parts libraries, like for screws, nuts, bearings etc...

Attribute of a general type – can be used always if the previous types are not the best option. For the general attribute, you can also define more methods of value definition - see below.

Attribute values can be:

- Text value – can contain any sequence of letters and digits (used for name, file name, description...)
- Integer value – contains whole numbers (used for a number of objects...)
- Real value – contains numbers with the decimal point (used for mass, surface area ...)
- Date value - contains a date

Value from 3D Solids or 2D Area

If the attribute is defined as attribute of a general type, you may select an additional method of its value definition. You can obtain value also as:

- Length measured in 3D
- Sheet thickness measured in 3D
- Cylinder diameter measured in 3D
- Any value measurable in 3D
- Scale of 2D area
- Format of 2D area

Other Definitions of Attribute

For each attribute, you can define also:

- When the attribute is required. If defined as required and value is missing, the warning sign appears in the corresponding line during attribute definition. You may check for missing attributes using the function "Check Attributes".
- Copy value from assembly. Value of the attribute can be copied from the attribute of the assembly the object belongs to (for instance Assembly Number).
- Sort criteria.
- If value is countable and how to create sum of objects.

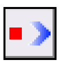
- Additional definition for type "file" - whether the attribute of the file type means a file containing the corresponding part.

Group membership, output to formatted text, title blocks or output to export files is described in following paragraphs.

If the attribute is listed in the Solid Attributes definition window or in BOM, Attributes and Title Blocks Settings window, you can always see usage of the attribute:

 The attribute is used in the list of parts (in formatted text files, suitable also for insertion into 2D area)

 The attribute is inserted into title block (or more title blocks).

 The attribute is used in text file suitable for import into other systems.

Groups of Attributes

You may create a new attribute group, or rename or delete an existing one. For an attribute, you can select or deselect a group the attribute belongs to. In the function "Solid Attributes", you can assign (or detach) the selected attribute group for the corresponding solid. The same can be done for an assembly or file in the function "Assembly Attributes, Title Block Filling".

Attribute group allows you to define different attributes for different objects. If the attribute group is defined for an object, extra attributes from this group are demanded.

For instance, you can define a group named "Sheet metals" and an attribute named "Sheet metal thickness". If you assign the group "Sheet metals" for a solid created as sheet metal, then during attribute definitions for this solid the attribute "Sheet metal thickness" is required. On the other side, the attribute "Sheet metal thickness" will not be required for a shaft.

Output to Formatted Text (List of Parts)

From BOM, you can create a formatted text file that contains a list of parts. Formatting is correct if the text file uses fixed-width font. If the file is inserted into 2D VariCAD area, columns are formatted properly for the fixed-width fonts. In 2D area, you can insert such a file into predefined tables using the function "Insert Text File - TXI".

Name and attributes of each part in the list can be divided into more lines.

For an attribute, you can define:

- If the attribute is printed into a formatted text file
- Width of the column in characters
- If the text is left or right aligned
- If a new line is created after the attributes value
- Optional insertion of spaces before the attribute value

Order of attributes written into such a formatted text file is the same as order in BOM, Attributes and Title Blocks Settings.

Title Blocks

You can define one or more title blocks. For a title block, you can define the corresponding 2D file with the title block itself. For an attribute, you can define its insertion into the title block.

Title Block Definition

Define a title block name. The name must be unique among other defined if you use more than one title block.

For title block automatic insertion into 2D area, define:

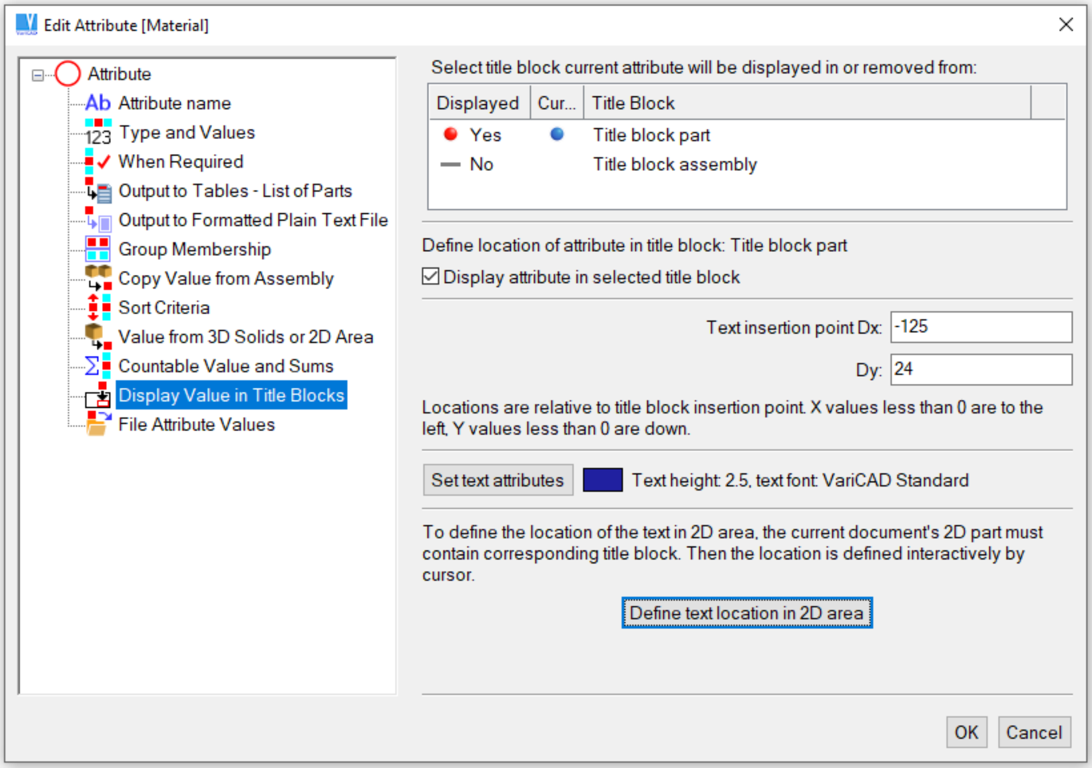
- 2D file with objects representing the title block. Prepare the file first. You can select the corresponding file from the list of files. In this case, the title block must be created with its insertion point at coordinates x=zero, y=zero. You can also select objects of the title block from the current file and then define the insertion point. In this case, open the file with objects first and then define the title block.
- Title block's insertion point relative to one of four corners of the drawing area.
- Title block for assemblies, parts or documents having no defined assembly-part status. Before automatic title block insertion, you have to select a title block from the list. If the current document has a defined assembly-part status and the title blocks are predefined differently for each such status, the selection is narrowed or skipped (in case of the only one option).

Once properly defined, VariCAD inserts the title block automatically into the desired 2D location whenever necessary.

Attribute Insertion into Title Block

For an attribute, you can select the title block the attribute is displayed in. One attribute may be displayed in more title blocks. Then define:

- Text properties like text height, slant, font or color (color can be mapped into line width for output to printer).
- Text location relative to title blocks insertion point. You can easily locate text using the cursor in 2D area, if the current file contains the corresponding title block.



Attributes to title blocks

Export to Other Systems

From BOM, you can create a text file suitable for other systems, like spreadsheets. Order of attributes written into such a text file may be different than the order in BOM, Attributes and Title Blocks Settings. From the list of attributes, you can define if the selected attribute is exported or define its order among the other exported attributes.

The name and attributes of each part in the file are written into one line. For export, you can also select an extra object - a level of insertion. In such case, the value determines whether the object in the line represents a part or assembly.

You can define a format of the text file:

- If the attributes are written into fixed-length records or if they are separated by selected characters (usually by ";" or by "|")
- How the value of the attribute "Manufactured - purchased" is represented
- How the text file is encoded – ANSI, Utf-8 or Unicode
- How ends of lines are created – according to the current operating system, or Windows (CR-LF), or according to UNIX (LF).
- Optionally, if a header is exported.

Compatibility of Defined Attributes and Attribute Groups

Before you will use the attributes, BOM and title blocks permanently, you should consider their proper configuration and change the settings according to your customs. The file containing the settings is a part of VariCAD distribution, but you should take it rather as an example. The settings are initialized when they are used for first time. From then, the attributes are recognized automatically. You may change their names (not only their values) and VariCAD always accepts them properly.

To allow such behavior for more users in one company, you must use the only one configuration file. The best option is to save the settings into location accessible via local area network. In the function "BOM, Attributes and Title Blocks Settings" select the option:

Change Path. You can load the settings from the selected directory (or LAN site). Next time you work with BOM, attributes or their settings the configuration is loaded from or saved to this directory. You can also redefine the configuration site saving the configuration into a selected path.

If you work with files created according to other attributes settings, the attributes match your settings only if they have identical attribute names (lower/upper cases are ignored).

If the solid attributes in files are defined according to an old attribute mask (in VariCAD versions older than 2007-3.00), they are recognized properly, too.

Compatibility of defined attribute groups follows the same rules as compatibility of attributes.

Working with BOM

BOM contains a legible list of assembly parts, their names and attributes. BOM allows you:

- To edit solids' attributes easily within one function
- To list, open or activate files associated with selected part
- To create files containing the list of parts, files suitable for other systems or to copy attributes into part files to be later used for title block filling

You can create BOM using three following methods – at a basic level, containing assembly or from 3D groups.

BOM at Basic Level

Create BOM at Basic Level – BOM, Ctrl + E

Each BOM object is a part belonging to the current assembly (to the current file). All objects are displayed at one level. All displayed objects are exported to files.

If the current assembly contains inserted sub-assemblies, you can select from following options:

BOM is created from highest level only. Objects creating sub-assemblies are not displayed.

BOM is created from all levels, including sub-assembly items. All objects are displayed at a single level. This method is useful, if you need a list of all parts and sub-assemblies from the current highest assembly.

BOM is created from all levels, only from single parts. All objects are displayed at a single level. This method is useful, if you need a complete list of all single parts necessary for manufacturing of the current assembly. This list of parts contains, of course, parts used directly in the highest assembly and also parts used in all sub-assemblies. It is a summary list of parts.

BOM Containing structure of Assembly

Create BOM Containing Assembly – DSS3

The first object in BOM is the current assembly; the next objects are the parts belonging to this assembly.

If the current assembly contains inserted sub-assemblies, you can select from following options:

BOM is created from all sub-assembly structures. BOM displays legible structure of the entire assembly and sub-assemblies, down to the lowest level.

BOM is created from the highest assembly level only. BOM displays the highest current assembly and all objects at the highest level.

BOM from 3D Group

Create BOM from 3D Assembly Group – BOMG

Each BOM object is a part from the current assembly (from the current file), same as the BOM creation at basic level. Only parts belonging to the selected solids group are inserted into BOM.

3D groups were used as a substitution of sub-assemblies, so the selection of one particular group can be used for BOM created for corresponding sub-assembly. However, this method is obsolete and is meaningful only for VariCAD files created in old versions.

BOM Objects

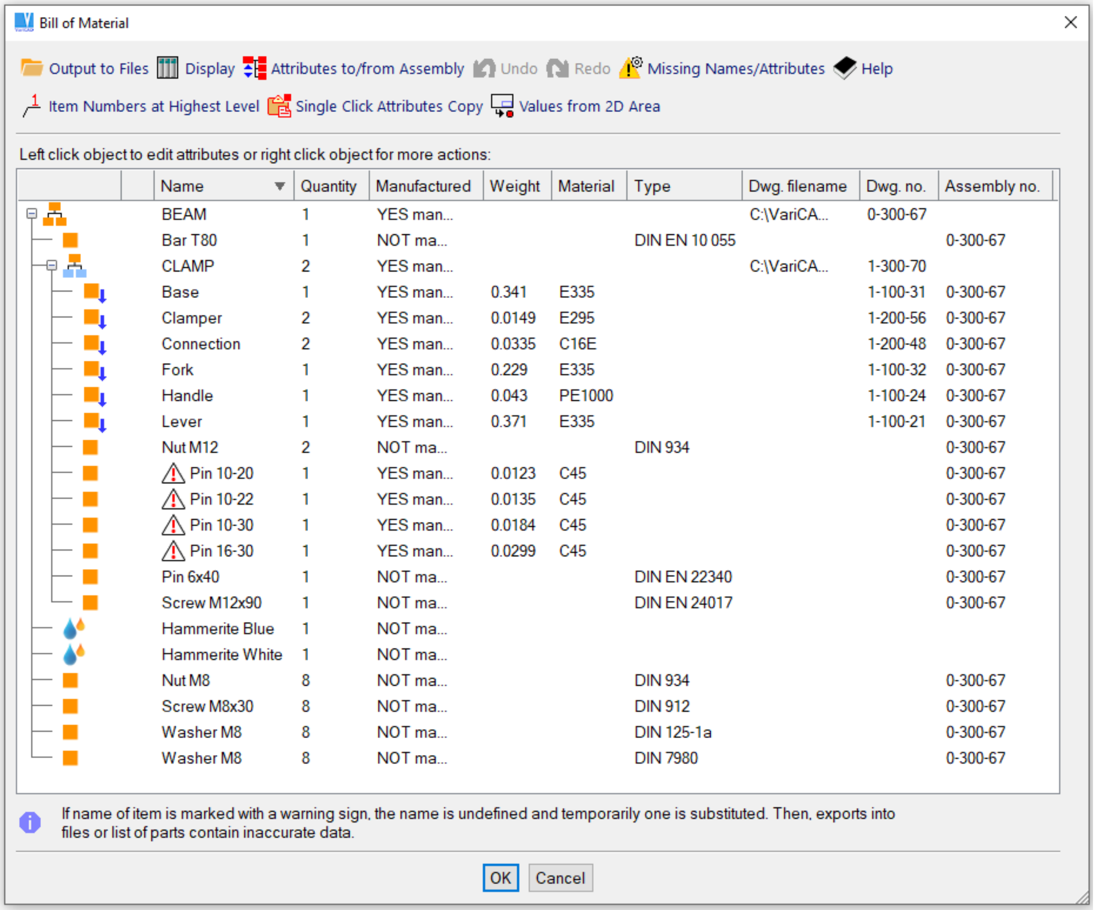
BOM object is either the current assembly or a part of the current assembly. It is listed in one BOM line. Object's name and attributes are arranged in the corresponding columns. Right clicking an object, you can:

- Edit its attributes
- List the corresponding file (if defined)
- Open the corresponding file
- Activate the corresponding file (if already open)
- Highlight the corresponding part or parts

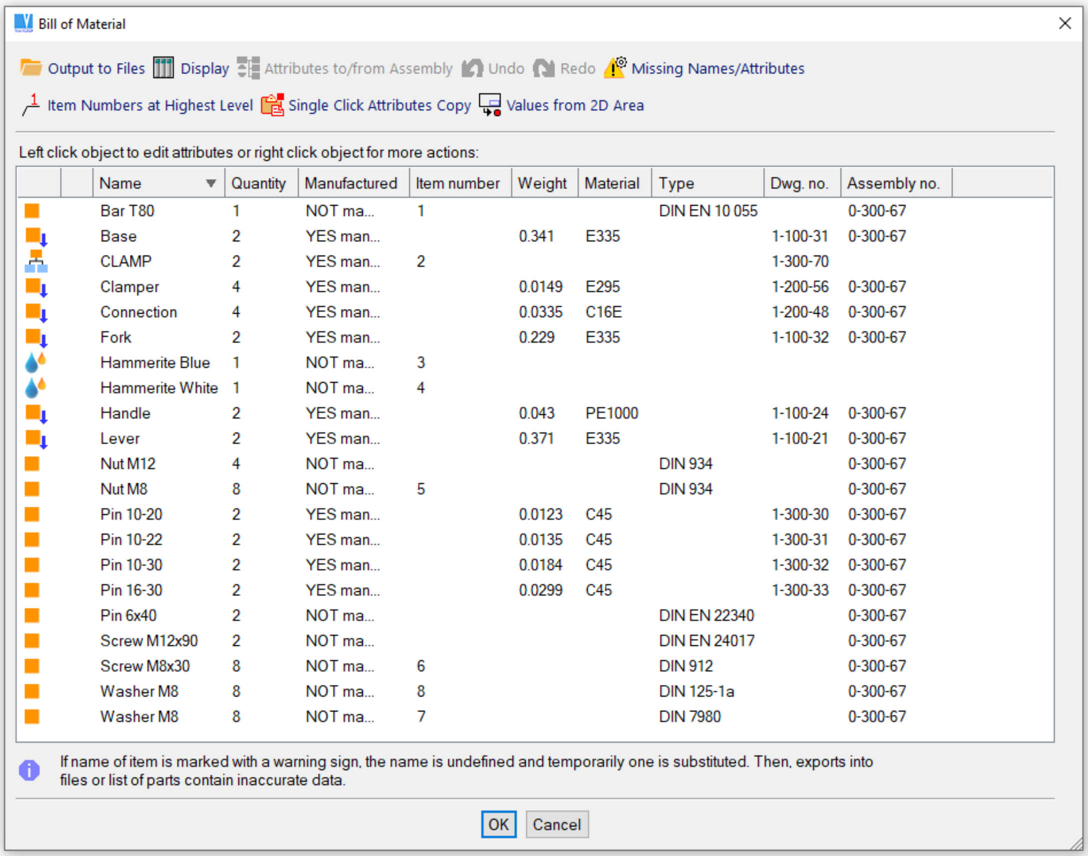
Left clicking an object, you can edit its attributes.

BOM objects are distinguished by different icons. These icons are identical with icons used in assembly tree structure window, see *3D Assembly Tree Scheme Window (page 287) (page 154)*.

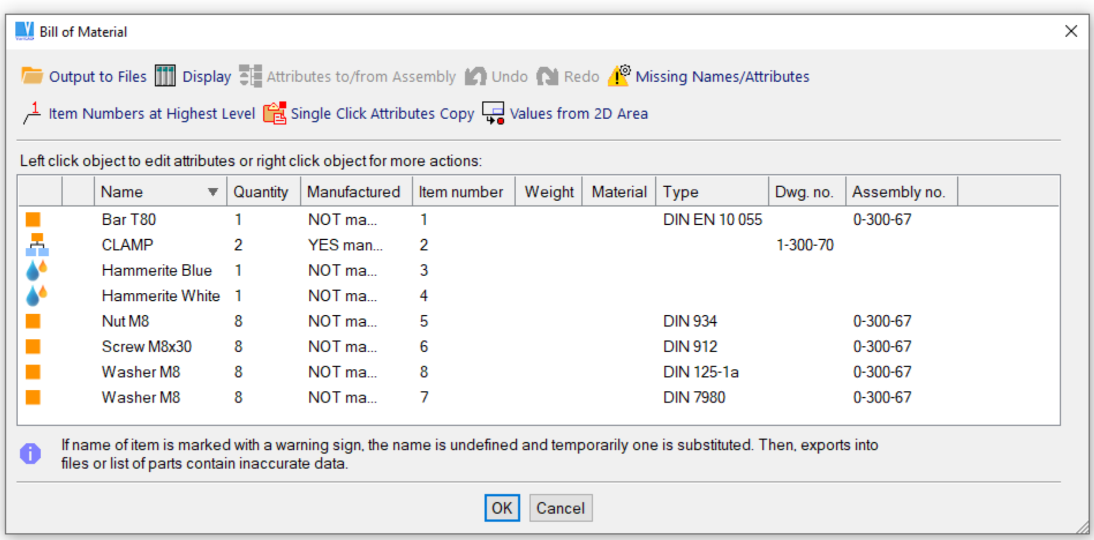
Following examples contain BOM created from the same assembly, by different methods:



Bill of materials, created as complete assembly structure



Bill of materials, created as complete list of single parts at one level

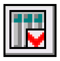


Bill of materials, created as highest assembly only

Displaying, Filtering and Sorting BOM Objects

To sort objects according to selected attribute values (or alphabetically according to names) left click the header of the attribute column.

You can use more options to define how the objects are displayed:

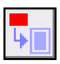


Display Columns. This function can be selected from the pop-up menu, or performed after right click the list header. You can select an attribute to display values in the column or deselect an attribute from displaying values in the column. You can also select the attribute the entire list is sorted according to.

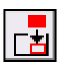
Filtering Objects to Display. It is possible to define filters for displaying objects. Filter allows you to display objects with a certain range of attribute values or to display objects with attribute values containing defined character sequences. For instance, if you want to create a list of all screws in the assembly, define a filter for object's name containing sub-text "screw".

Creating Files from BOM

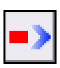
You can create output to following files:



Output to Formatted text, see *Output to Formatted Text (List of Parts) (page 301)*.



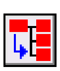
Attributes Output to Part Files. Data are suitable for the title block filling. To fill part title blocks after attributes export, perform function *Assembly/Part Attributes*, *Fill Title Blocks (page 321)* for each file containing the corresponding part.



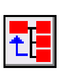
Output to Text Files. The output is suitable for other systems, like spreadsheets. See *Export to Other Systems (page 303)*. If the BOM is created from assembly and contains sub-assemblies, you can select between export from the highest level or from all levels.

Copying Data from Assembly into Parts and Vice Versa

If the BOM is created as BOM containing an assembly, extra options are available:



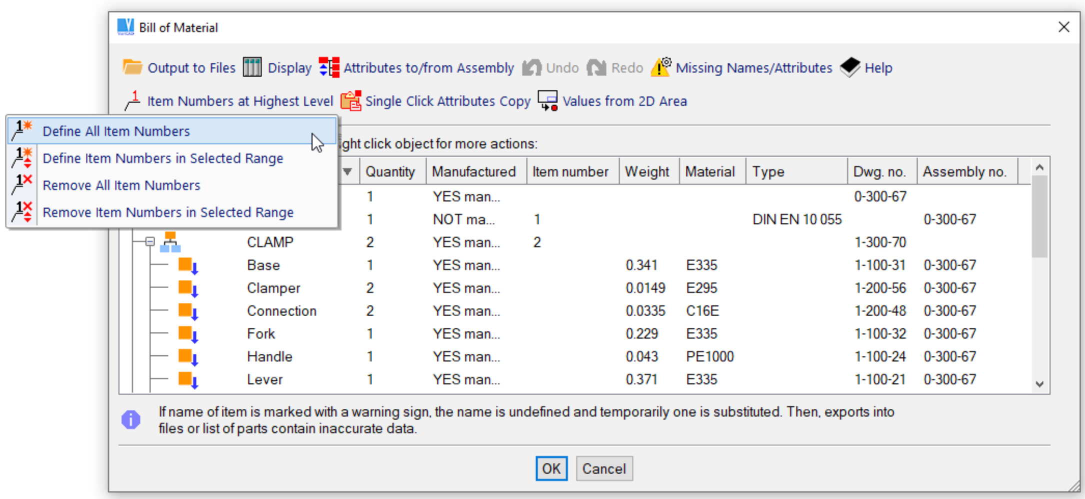
Attribute Values from Assembly. Some attributes may have defined option "Copy Value from Assembly". For instance, a value of the attribute "drawing number" from the assembly is a value of the attribute "assembly number" in the part. This function copies the attribute value from the assembly into the corresponding attribute of each part. If the BOM is created from assembly and contains sub-assemblies, there are two options. You can select export from the highest assembly into all lower objects, or export from each sub-assembly to one level down.



Sum of Attribute Values. For attributes with the corresponding option, value in the assembly is sum of values from all parts. Weight of the assembly is obtained as sum of all weights of parts. This function defines attribute values of the assembly as sum of corresponding attributes of the parts.

Defining Item Numbers in BOM

To define item numbers in BOM, your BOM mask must contain attribute of type “Item Number”. Then, you can assign item numbers to objects in assembly, at the highest level. After creation of 2D, item numbers are automatically used as leader texts.

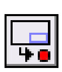


BOM menu containing item numbers definitions

Single Click Attributes Copy

This command opens attributes window. Then, define attribute values. After definition, click BOM line by line. Defined attributes are added to each selected line. Optionally, all BOM lines can be modified at one step. Some attributes or names cannot be defined this way. This method is convenient for definition of attributes common for multiple solids, like Manufactured-Purchased, Date of creation etc.

Attributes from 2D Area



This option allows to copy attributes from 2D area of each part drawing. As an attribute, you can copy 2D drawing format and/or 2D drawing scale. Attributes can be updated line by line, or for all objects at one step. 2D drawing format or 2D drawing scale as an attribute must be enabled in BOM mask. Both values can be further used for copying into 2D drawing title blocks of parts (details).

Supplementary Objects

Supplementary Objects - SPO

You can define supplementary objects for the current document. Supplementary objects are all objects not created as 3D solids. Typical example is oil, paint, welding electrodes etc. In BOM, all supplementary objects are at the same level as the other 3D solids. They are correctly listed in the BOM exports, a list of parts or wherever assembly parts appear.

Working with supplementary objects interface is similar to working with BOM. You can add, delete or edit a selected object.

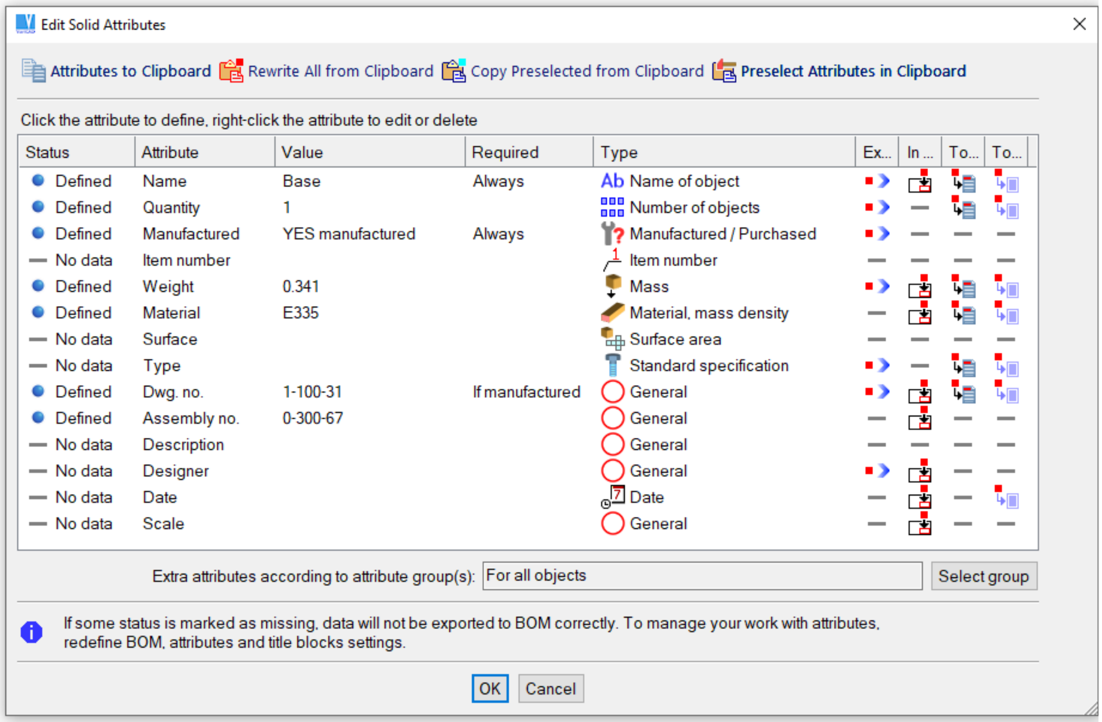
Solid and Assembly Attributes

Solid Attributes

Solid Attributes - SAT

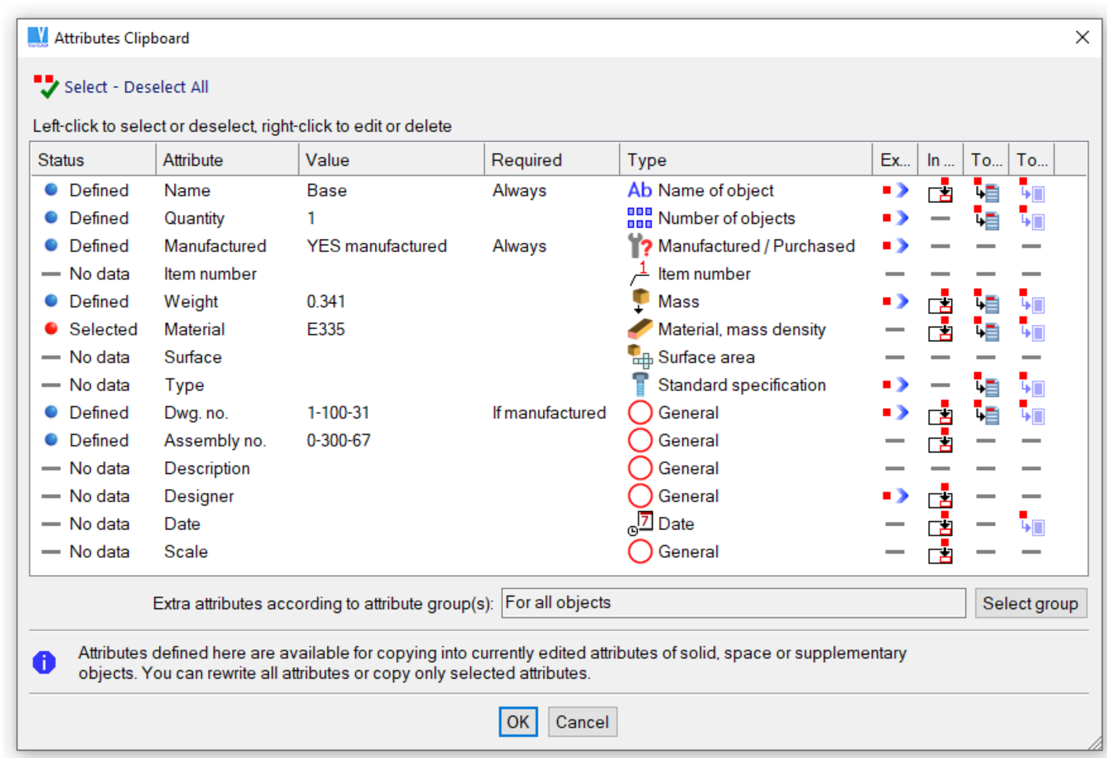
Define new attributes, change existing attributes or delete attributes of a selected solid. Attributes are defined according to the *BOM, Attributes and Title Blocks Settings (page 298)*. If you need to define mass or the surface area, there are geometric calculation functions available. Using the “Number of Items” attribute is not relevant when creating a BOM from 3D. In 3D, the number of items is defined automatically as the exact number of currently existing parts.

You can also define an attribute group or groups for the selected solid. Then you may define extra attributes belonging to the selected attribute group.



Solid attributes definition

Attributes can be put into a clipboard. Then, these attributes can be copied into another solid. You can rewrite all existing attributes, but more likely, you may need to rewrite only some attributes. Typically, it may be date of creation, author of project or similar attributes, usually common for multiple solids.



Solid attributes clipboard

Attributes from List - ATL

Define attributes from the displayed list of solid names or attributes. The solids that use a selected value are highlighted. From the highlighted group, select the object whose attribute you want to change. This function changes attributes of each object individually.

Change Identical Attributes - MTC

Changes one attribute value to a new value for all objects. First, select the attribute from the attributes list. Then select the old value, enter the new value, and all old values will be replaced with the new ones. For example, you can select the attribute “material” and replace all instances of one material with another.

Check Attributes - ATC

You can choose to check for missing names, missing attributes, or missing attributes according to their definition – see *BOM, Attributes and Title Blocks Settings (page 298)*. If objects with missing desired values are found, attribute definition is performed. You can exclude objects from attribute checking, so that these objects are ignored during next checking.

List of Materials

For definition of solid attributes and BOM data, material of solids can be selected from a list of materials. The list of materials is distributed as a part of VariCAD package. However, the distributed list is rather schematic and short. Each user can add materials used in his company, or edit existing data. Materials in list are sorted into material groups.

Together with material designation, each material contains definition of mass density and other optional data. By other words, if a material is once defined for a solid, you don't need to define mass density later. Mass density must be known if you want to calculate mass of solids.

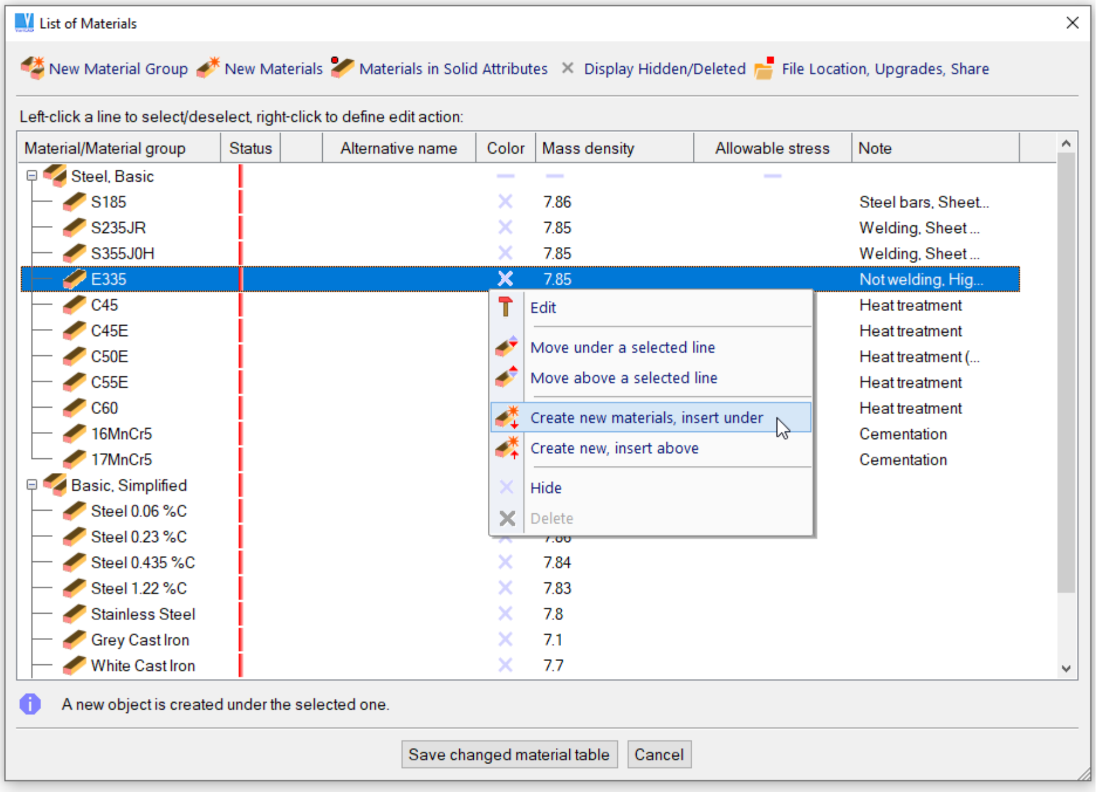
Optionally, materials contain definition of colors. In settings of display of solid colors and wires, you may turn on a display mode, when solid colors correspond to solid materials. For solids without material definition or materials without color definition, a specific common color is used.

Creating and Editing List of Material

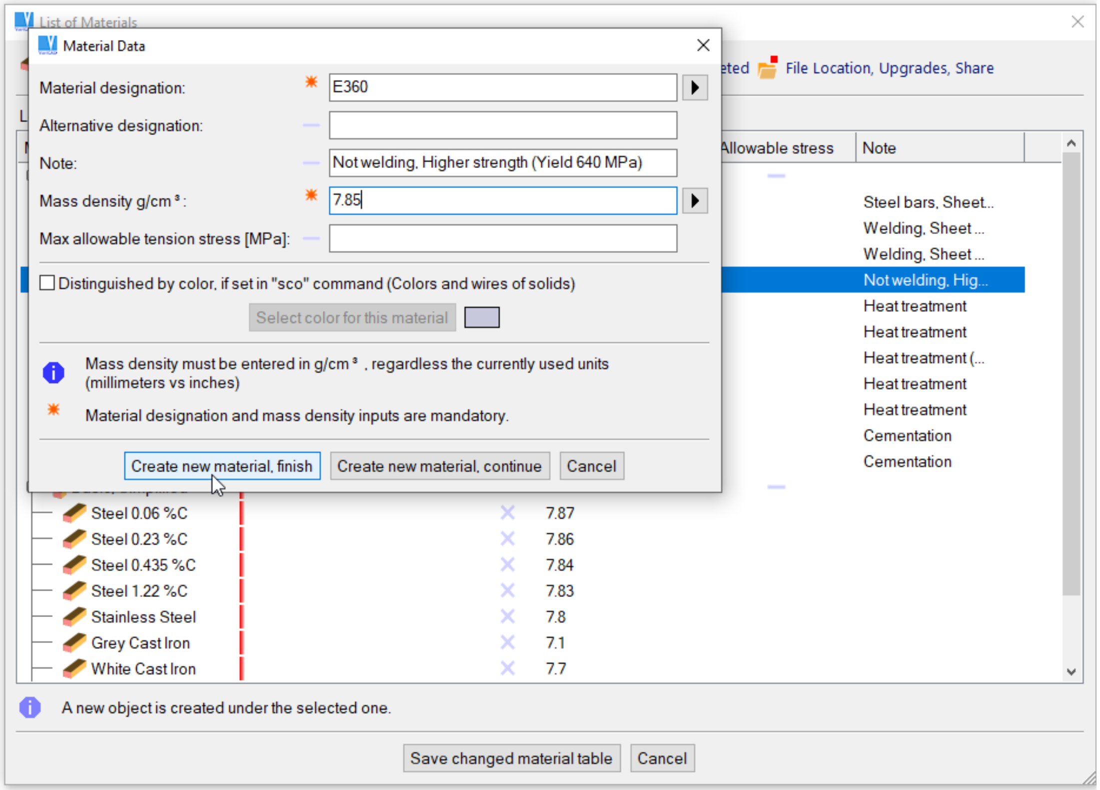
 **Define or Edit List of Materials - DMAT**

This command manages list of materials. You can run it also from system settings – command "CFG", from General section, "List of Materials Settings". Here, you can delete user-defined list of materials. If the user-defined list is deleted, a new one is created as a copy of list distributed in VariCAD package.

To create a new material, right-click a line and select, whether the new material is inserted above or under it. You can also click a corresponding icon at upper row of icons. To create a group of materials, or to edit or delete material or group, click a line and select a step from pop-up menu.



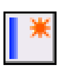
New material will be inserted under a selected line

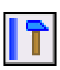


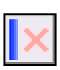
Definition of a new material

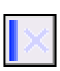
Status of Materials or Groups

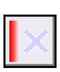
Status is displayed in second column of the list. Icons can contain combination of multiple situations.

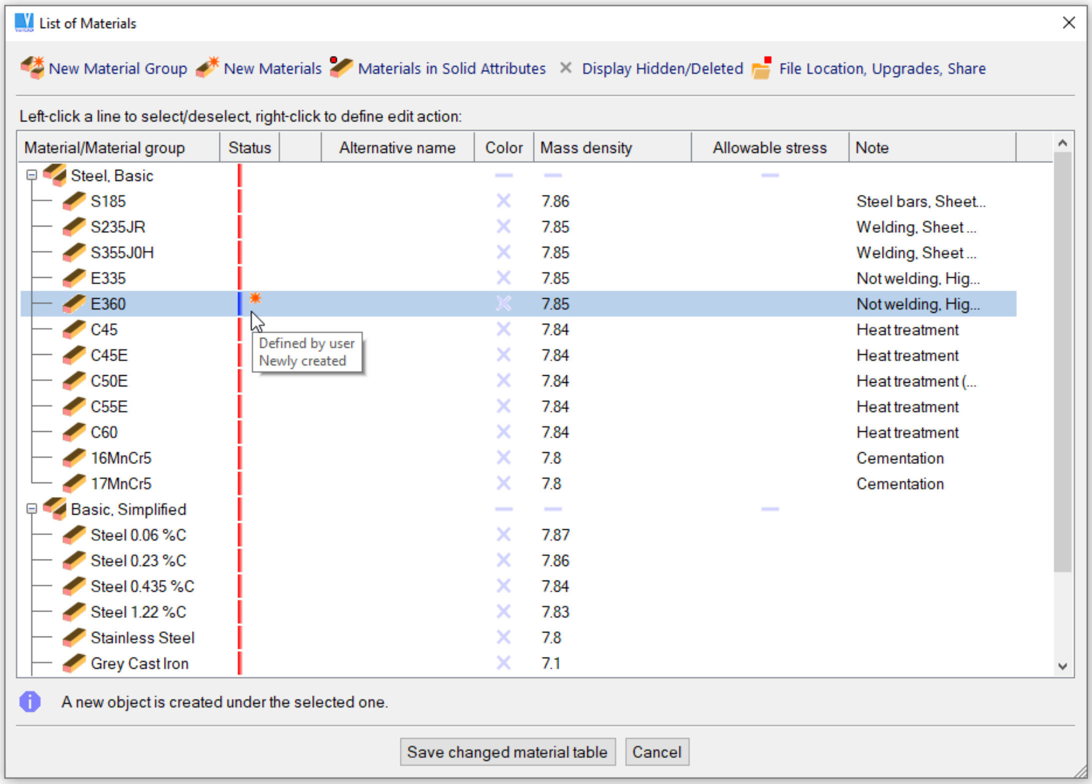
 Newly created

 Edited

 Scheduled for delete. It is shown only, if you turn on “Display Hidden/Deleted”

 Hidden. It is shown only, if you turn on “Display Hidden/Deleted”

 Hidden, original data. It is shown only, if you turn on “Display Hidden/Deleted”

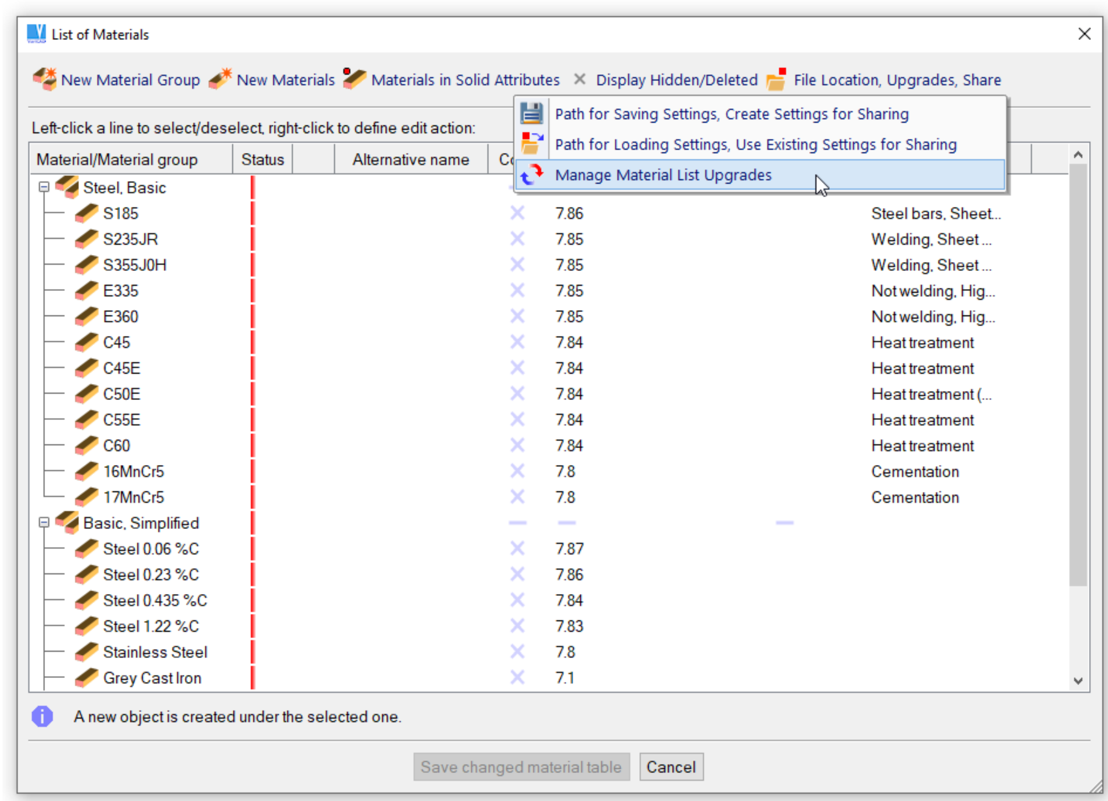


Example of status of material

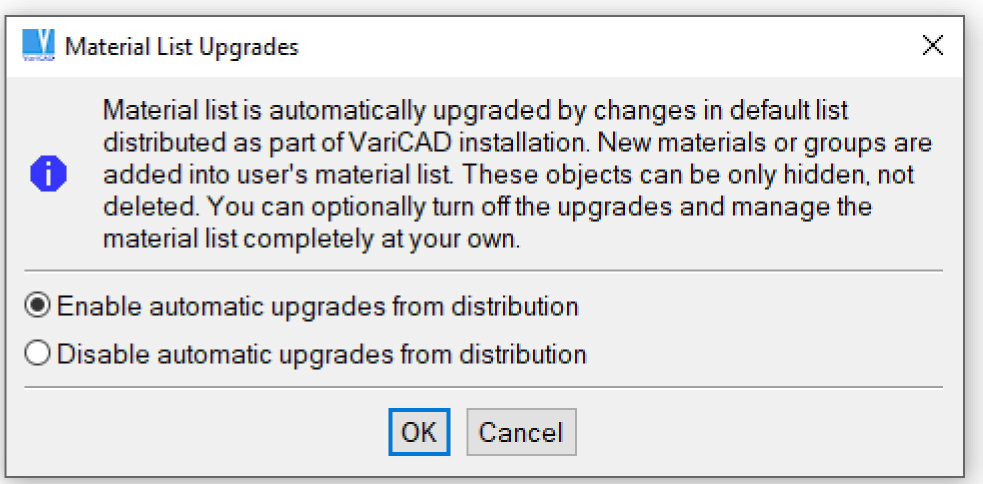
Original vs. User Defined Data, Update of Table

Data (materials or groups of materials) in table may be original or user-created. Original data are updated from VariCAD distribution. They cannot be deleted or renamed. They can be hidden, or partially edited.

Optionally, you can stop updating of list of materials and to manage it completely at your own. In such case, all data have status of user-defined.



Selection of update options of list of materials



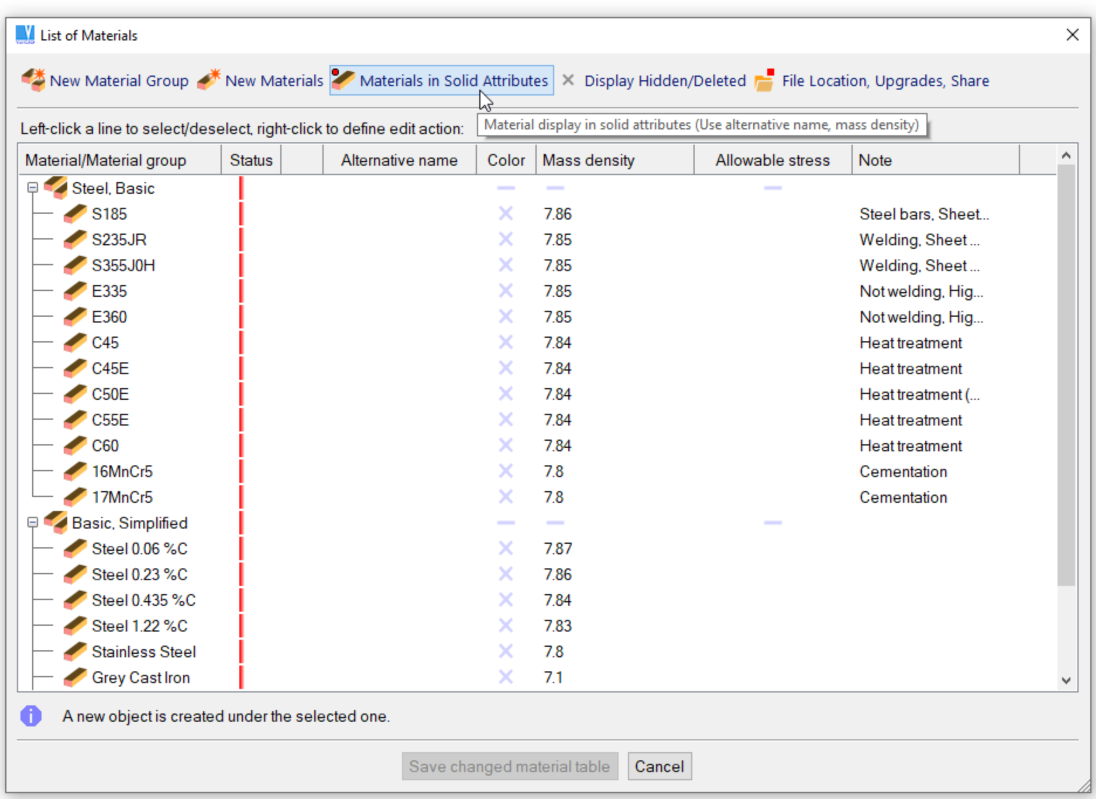
Update options of list of materials

Sharing List of Materials

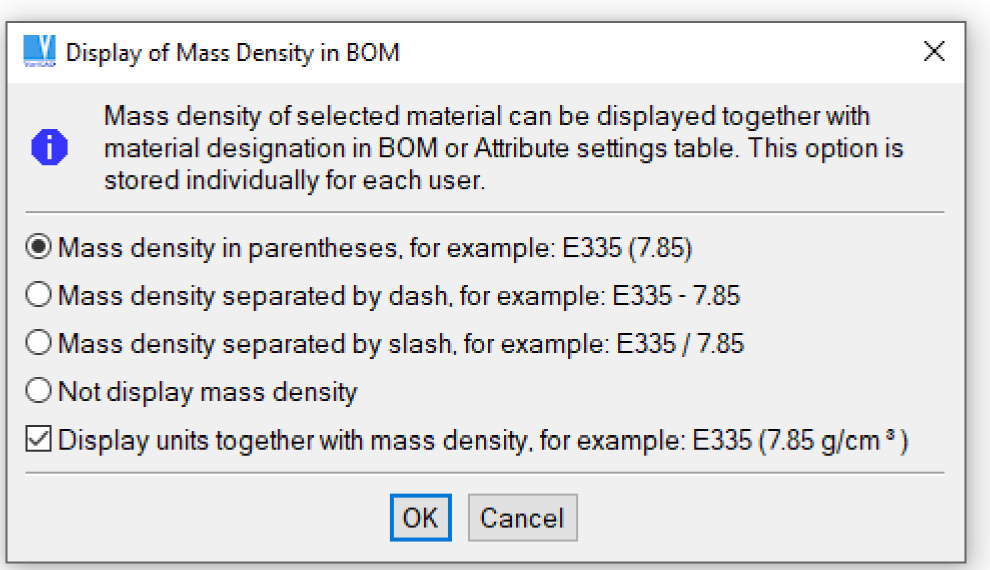
Sharing of list of materials is available from the same menu as you can select update options (see above). If you create a material table suitable for all other users, then save it into selected network path. All users need to set the same network path for loading of list of materials.

Material as a Solid Attribute

It is possible to select, how materials are displayed in list of solid attributes.



Selection of attribute display options

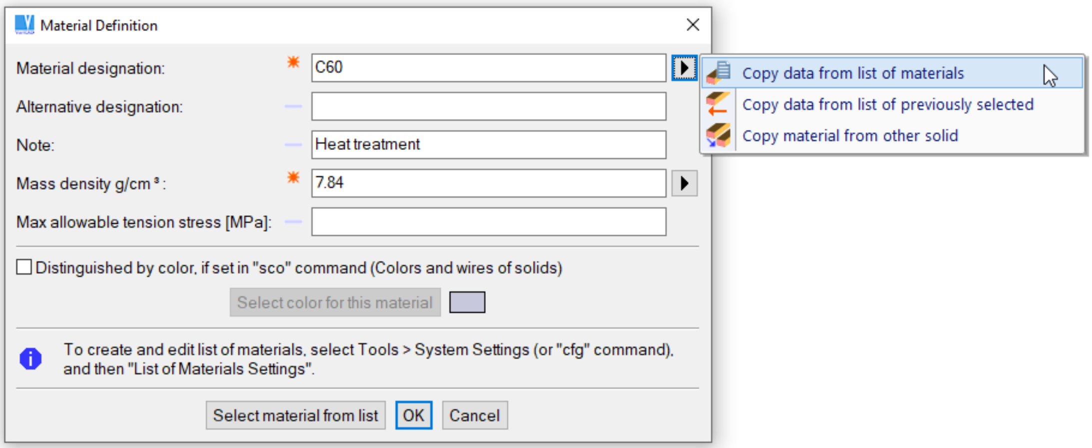


Options of material display in list of solid attributes

To define material as a solid attribute, you can:

- Select a material from a list of materials, then optionally change data before the definition is completed
- Select a material from a list of materials (click the corresponding button at bottom row) and finish immediately
- Use a material from a list of previously used materials
- Or, to define a material manually

It is always mandatory to define material designation and mass density.



Definition of material as a solid attribute

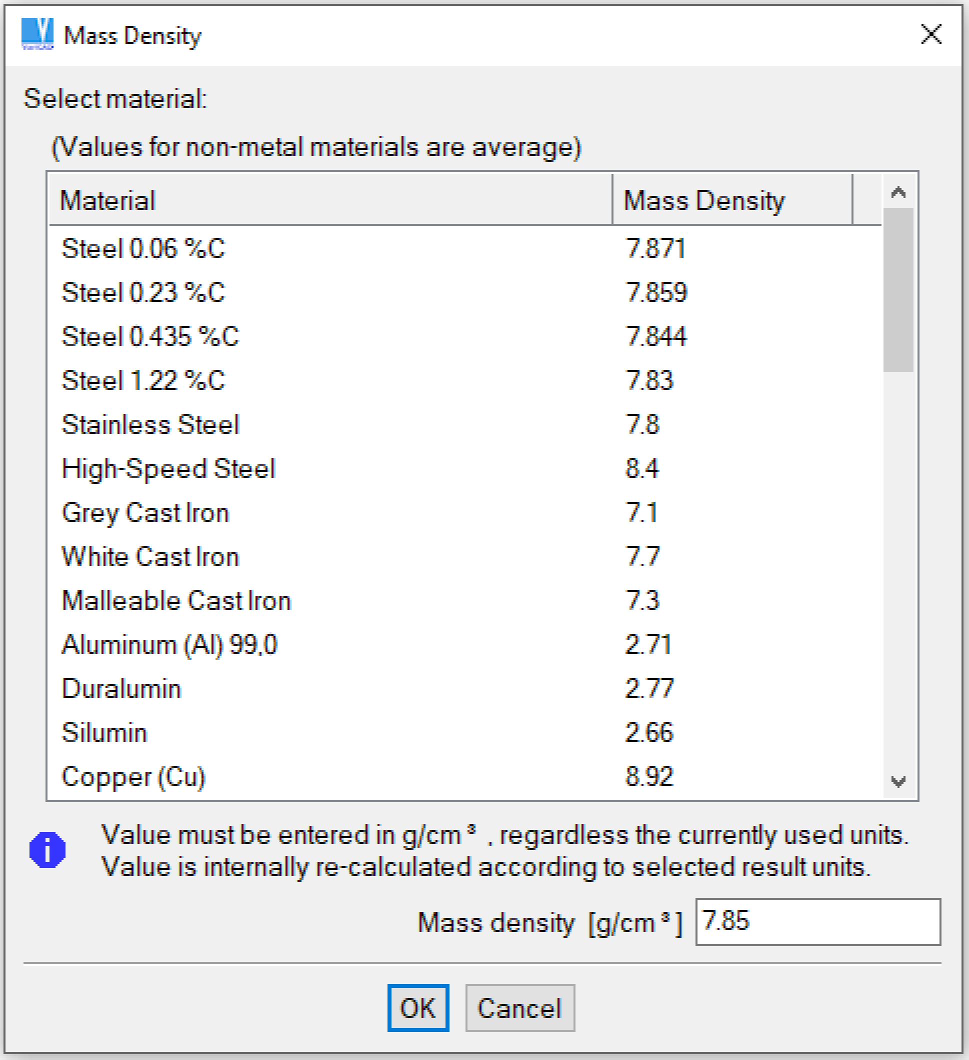


Table of mass densities

Define Mass Density or Material - MAT

This command allows you to define mass density and material for a group of selected solids. Although the material can be defined as a solid attribute, you may define it separately from attributes management.

Mass density is stored for each solid together with material. If a mass of an assembly is calculated and different materials of different parts are used, the calculation copes this correctly. If some or all solids have no material defined, mass calculation asks for mass density. Then, entered value is used only temporarily for the calculation.

Assembly Attributes, Title Block Filling

Assembly, Sub-assembly or Part Attributes, Fill Title Blocks - AAT

Define new attributes, change existing attributes or delete attributes of the current file – assembly, sub-assembly or part file. Attributes are defined according to the *BOM, Attributes and Title Blocks Settings (page 298)*, the same way as in the function "Solid Attributes", described above. If you define attributes of a part file containing a part from assembly links, the attributes are defined exactly for this part. Otherwise the attributes are connected to the current file instead to a solid. This function also allows you to fill title blocks.

If the current file is a sub-assembly, the file attributes are solid attributes of integrated sub-assembly, whenever the sub-assembly is inserted into an assembly.

Fill Title Block, Define Attributes - AAT2

Similar function as the previous - available in 2D.

Filling Title Blocks

Select a title block first. If you have defined only one title block, then the selection is skipped. If the title block has no corresponding file defined or if the corresponding file was not found, then you cannot insert the title block into 2D. You can only insert 2D text objects representing attribute values (name, date, material etc.) relative to title block's insertion point.

Automatic insertion of the title block is possible, if you need the title block always at the same location relative to one of four corners of the drawing area. Insertion of attributes uses 2D text properties defined in *BOM, Attributes and Title Blocks Settings (page 298)*. The title block location and location of texts are defined in the same function.

Fill Title Block – only fills the selected title block. If started in 3D, VariCAD switches itself into 2D. Then select a title block's insertion point. All defined attributes are inserted into predefined locations in the title block area – 2D text objects are created.

Update Title Block – fills or updates the selected title block. If the title block was filled previously, the old objects are automatically removed first. The other is the same as in the previous option.

Insert Title Block - if the selected title block does not exist, then it is inserted automatically. All defined attributes are inserted also automatically into predefined locations relative to the title block's insertion point. If the title block is already filled, the old objects are removed first.

Chapter 15. Tips and Tricks

Although most of the interactions within VariCAD are obvious, the system provides some additional conveniences you may not know about.

Display Changes

- To pan, press Ctrl and the left mouse button while moving the cursor. You can also press and hold the middle mouse button (usually the wheel) and move the cursor.
- To zoom, use the mouse wheel. You can configure the mouse wheel direction of rotation.
- To zoom, press Shift and the left mouse button while moving the cursor.
- To rotate the view around the center of rotation, press Shift, Ctrl, and the left mouse button while moving the cursor. You can also press and hold right mouse button while moving the cursor.

Selecting Objects between Commands

- Right-click after selection of 3D or 2D objects starts pop-up menu containing edit functions. Pop-up menu may contain different possibilities, if you right-click a single highlighted object.
- Right-click a single highlighted object starts pop-up menu with edit functions performed at the highlighted object.
- If no objects are selected and you right-click an empty area, pop-up menu offers features related to entire drawing.
- To select a part of solid (like a hole, fillet...), press Ctrl while moving the cursor.
- To select an edge or edges for blending, press Shift while moving the cursor.
- To select a drawing plane for sketching in space, press Ctrl + Shift while moving the cursor.
- To select entire 3D view in 2D drawing area, including connected dimensions, axes or hatches, press Ctrl while moving the cursor.
- To start a selection window for 2D stretching, press Ctrl + Shift while moving the cursor.
- To select corner, fillet or chamfer function in 2D or sketching, press Shift while moving the cursor. Right-click for options, when a corner or intersection of 2D objects is detected.
- To disable automatic objects detection temporarily during 2D or 3D dragging, press and hold F1 or press and hold left mouse button while moving the cursor. To disable automatic detection temporarily during location input, press and hold F2. Sometimes, automatic objects detection may interfere with dragging – especially if cursor increment movement is used.

Basic Tips

- If you need to enter any value, like a distance or angle, you can enter a mathematical expression instead of a single value. If supported in 3D, the expression can contain new or existing parameters.
- Consider right-click behavior during 2D/3D location input, objects selection etc... By default, right-click is the same as Enter, while Ctrl + right-click opens pop-up menu with currently available options. The same pop-up menu can be opened by right-click and left-click simultaneously.

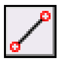
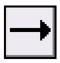
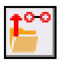
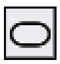
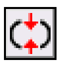
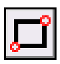
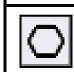
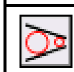
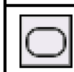
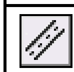
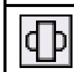
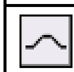
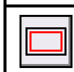
- If you right-click a highlighted 2D object during 2D location or a highlighted 3D object during 3D location, pop-up menu is displayed. You can select a location at end-point, mid-point, circle center or other options directly from menu.
- It is convenient to use cursor step movement, especially for dragging in 3D or in 2D. During dragging, you can right-click an empty area and select dragging increment or turn off dragging in increments.
- Use command "CFG" to change the default settings.

More Tips

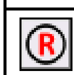
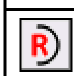
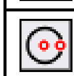
- To go one step back in a function, press the middle mouse button (usually a wheel) or Ctrl + Backspace.
- To go one step back in a function, press additional mouse button, if you have 5-button mouse.
- If current interactive input does not wait inside a window (panel), right-click opens a pop-up or is the same as pressing Enter, according to situation.
- If the system waits for input in a window (panel), right-click while the cursor is inside the window is the same as clicking OK.

Chapter 16. List of All VariCAD Functions

Drawing Lines and Curves

Icon	Command	Hotkey	Description
	LIN	Ctrl + L	Line
	ARR	N/A	Arrow
	2DFF	N/A	2D Objects, XY Coordinates from File
	CPL	N/A	Draw Polyline
	PLL	N/A	Join Objects into Polyline
	RECT	N/A	Rectangle
	POL	N/A	Polygon
	TAN	N/A	Tangent Line
	ELL	N/A	Ellipse
	MLL	N/A	Multi Line
	SHA	N/A	Shaft
	SPL	N/A	Spline
	BOR	N/A	Sheet Border

Drawing Circles and Arcs

Icon	Command	Hotkey	Description
	CCR	N/A	Circle Center Radius
	ACR	N/A	Arc Center Radius
	CCP	N/A	Circle Center Point
	ACP	N/A	Arc Center Point

	CR2	N/A	Circle 2 Points
	AR2	N/A	Arc 2 Points
	C3P	N/A	Circle 3 Points
	A3P	N/A	Arc 3 Points
	APT	N/A	Arc Point Tangent
	AT2	N/A	Arc Tangent to 2 Objects
	CT2	N/A	Circle Tangent to 2 Objects
	TG3	N/A	Circle Tangent to 3 Objects
	HOL2	N/A	Group of Holes

Creating 2D Text

Icon	Command	Hotkey	Description
	NOTE	N/A	Note (Multiple Lines)
	TEX	N/A	Single Text Line
	TXI	N/A	Insert Text File

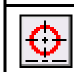
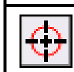
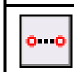
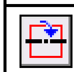
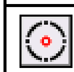
Creating Points

Icon	Command	Hotkey	Description
	POI	N/A	Point
	POC	N/A	Points on Arc
	PLN	N/A	Points on Line, Number
	PLD	N/A	Points on Line, Distance

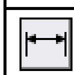
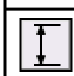
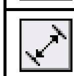
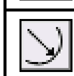
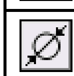
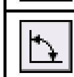
Hatching

Icon	Command	Hotkey	Description
	HAT	N/A	Hatch, Select Boundary
	AHB	N/A	Hatch, Detect Boundaries Automatically
	CHH	N/A	Change Hatch Area or Style
	CHHP	N/A	Change Hatch Style
	CHP	N/A	Create Pattern

Creating Axes

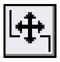
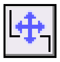
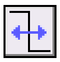
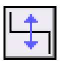
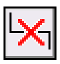
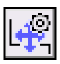
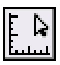
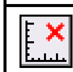
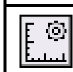
Icon	Command	Hotkey	Description
	AXI	N/A	Axes
	CAX	N/A	Circle or Arc Axis
	AX2P	N/A	Axis by 2 Points
	LAX	N/A	Axis of Rotation Surface
	AXPC	N/A	Create Pitch Circle

Dimensioning

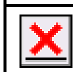
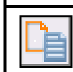
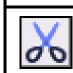
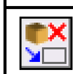
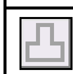
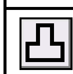
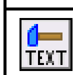
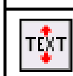
Icon	Command	Hotkey	Description
	HDI	N/A	Horizontal Dimension
	VDI	N/A	Vertical Dimension
	SDI	N/A	Diagonal Dimension
	RDI	N/A	Radius Dimension
	DDI	N/A	Diameter Dimension
	ADI	N/A	Angular Dimension

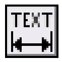
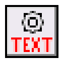
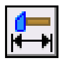
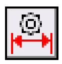
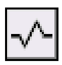
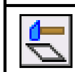
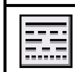
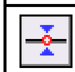
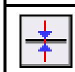
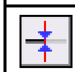
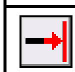
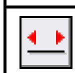
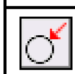
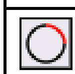
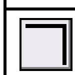
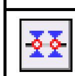
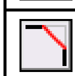
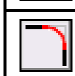
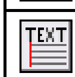
	HPD	N/A	Horizontal Baseline Dimensions
	HSD	N/A	Horizontal Serial Dimensions
	HDD	N/A	Horizontal Datum Dimensions
	VPD	N/A	Vertical Baseline Dimensions
	VSD	N/A	Vertical Serial Dimensions
	VDD	N/A	Vertical Datum Dimensions
	SPD	N/A	Diagonal Baseline Dimensions
	SSD	N/A	Diagonal Serial Dimensions
	SDD	N/A	Diagonal Datum Dimensions
	HDM	N/A	Horizontal Diameter Dimension
	VDM	N/A	Vertical Diameter Dimension
	SDM	N/A	Diagonal Diameter Dimension
	HTH	N/A	Horizontal Thread Dimension
	VTH	N/A	Vertical Thread Dimension
	STH	N/A	Diagonal Thread Dimension
	THR	N/A	Thread Dimensions
	STXA	N/A	Single Text Arrow
	MTXA	N/A	Multiple Text Arrow
	LDR	N/A	Leader
	CHLDR	N/A	Check and Update Leaders
	FSY	N/A	Finish Symbols
	WSY	N/A	Welding Symbols
	TSY	N/A	Tolerance Symbols

2D Drawing Tools

	DCC	N/A	Displayed Cursor Coordinates
	ORTA	N/A	Ortho Modes, Leading Lines
	ORT	F11	Drawing in Ortho Mode
	ORTC	N/A	Ortho, if Close to Vertical/Horizontal
	ORTH	N/A	Ortho, Next Horizontal
	ORTV	N/A	Ortho, Next Vertical
	OMO	Ctrl + F11	Turn off Ortho Mode
	ORTS	N/A	Ortho, Set Close Angle
	STP	F9	Drawing in Increment Mode
	STO	N/A	Increment Mode Off
	STS	N/A	Set Increments of Cursor Movement

Editing 2D Objects

Icon	Command	Hotkey	Description
	DOB	Ctrl + D	Delete 2D Objects
	CPY	Ctrl + C	Copy to Clipboard
	PAS	Ctrl + V	Paste
	CCUT	Ctrl + X	Delete - Cut to Clipboard
	ROL	N/A	Remove Previous View Export
	BLA	Ctrl + B	Blank 2D Objects
	UBL	Ctrl + U	Unblank 2D Objects
	ETX	N/A	Edit Text
	MTL	N/A	Move Text Vertically

	TWD	N/A	Text Width
	TAC	N/A	Change Text Style
	EDM	N/A	Edit Dimension
	EDI	N/A	Edit Dimension Text
	MDT	N/A	Move Dimension Text
	EDS	N/A	Change Dimension Style
	EXP	N/A	Explode
	BLN	N/A	Break Line
	MLA	N/A	Change Layer
	MPE	N/A	Change Color
	MLT	N/A	Change Line Type
	BPO	N/A	Divide by Point
	BBO	N/A	Divide by Curve
	TBO	N/A	Trim
	EBO	N/A	Extend
	CHLL	N/A	Change Line Length
	CHAR	N/A	Change Arc Radius
	CEC	N/A	Circle from Arc
	CCO	N/A	Corner
	RSG	N/A	Remove Segment
	CHM	Ctrl + R	Chamfer 2D Corner
	RND	Ctrl + F	Fillet 2D Corner
	JTX	N/A	Align Text

	ESP	N/A	Edit Spline
	BTF	N/A	Explode Font

Transforming and Copying 2D Objects

Icon	Command	Hotkey	Description
	MOV	Ctrl + T	Translate or Copy 2D Objects
	DRG	N/A	Dynamic Translation
	ROT	N/A	Rotate or Copy 2D objects
	DRO	N/A	Dynamic Rotation
	SCA	N/A	Scale
	DSC	N/A	Dynamic Scaling
	DTR	N/A	Translate and Rotate
	MIR	N/A	Mirror
	OFFS	N/A	Offset
	SOB	N/A	Stretch
	STRVECT	N/A	Stretch by Vector
	CTA	N/A	Array Copy
	DST	N/A	Stretch to Direction

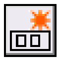
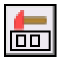
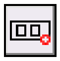
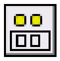
Grid, Construction Lines

Icon	Command	Hotkey	Description
	GRI	Ctrl + G	Grid
	CCL	N/A	Create Multiple Construction Lines
	HCL	N/A	Horizontal C.L.
	VCL	N/A	Vertical C.L.

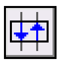
	HVCO	N/A	Horizontal/Vertical C.L. Offset
	HVCP	N/A	Horizontal/Vertical C.L. from Point
	HCT	N/A	Horizontal C.L. Tangent
	VCT	N/A	Vertical C.L. Tangent
	1CL	N/A	Angle 1 C.L.
	1CI	N/A	Angle 1 C.L. Offset
	1CF	N/A	Angle 1 C.L. from Point
	1CT	N/A	Angle 1 C.L. Tangent
	1CS	N/A	Define Angle 1
	2CL	N/A	Angle 2 C.L.
	2CI	N/A	Angle 2 C.L. Offset
	2CF	N/A	Angle 2 C.L. from Point
	2CT	N/A	Angle 2 C.L. Tangent
	2CS	N/A	Define Angle 2
	DCL	N/A	Delete Selected Construction Line
	DAH	N/A	Delete All Horizontal Construction Lines
	DAV	N/A	Delete All Vertical Construction Lines
	DA1	N/A	Delete All Angular Construction Lines
	DACL	N/A	Delete All Construction Lines

2D Blocks

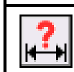
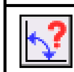
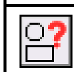
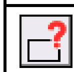
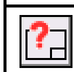
Icon	Command	Hotkey	Description
	BLS	N/A	Save Block
	BLI	Ctrl + K	Insert Block

	BLC	N/A	Create Block
	BLE	N/A	Edit Block
	BIE	N/A	Change Insertion Point
	BAE	N/A	Edit Block Attributes

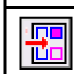
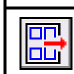
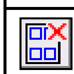
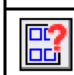
Symbols

Icon	Command	Hotkey	Description
	SYM	N/A	Symbols

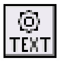
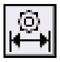
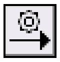
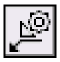
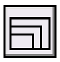
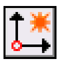
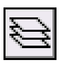
2D Check Functions

Icon	Command	Hotkey	Description
	COO	N/A	2D Coordinates
	DIS	N/A	2D Distance
	ANG	N/A	Angle
	ODT	N/A	2D Object Information
	CHL	N/A	Highlight Layer
	2DA	N/A	Calculate 2D Section Properties
	2DP	N/A	2D Drawing Area Properties

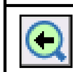
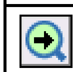
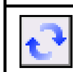
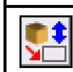
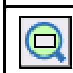
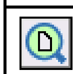
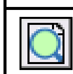
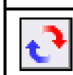
2D Work Sets

Icon	Command	Hotkey	Description
	ATW	N/A	Add to Work Set
	RFW	N/A	Delete from Work Set
	CLW	N/A	Clear Work Set
	CHW	N/A	Highlight Work Set

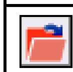
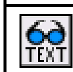
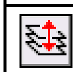
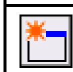
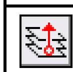
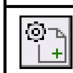
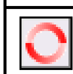
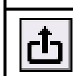
2D Settings

Icon	Command	Hotkey	Description
	TXA	N/A	Text Style
	DMA	N/A	Dimension Style
	ARA	N/A	Arrow Style
	SWS	N/A	Weld Symbol Settings
	FMT	N/A	Change Drawing Format
	SCH	N/A	Change Drawing Scale
	UCO	N/A	Change Center of Coordinates
	SBD	N/A	Sheet Borders Definition
	LAY	N/A	Create, Edit or Delete Layers

2D Views

Icon	Command	Hotkey	Description
	ZPR	N/A	Undo View
	ZRD	N/A	Redo View
	RDR	F6	Redraw
	SON	N/A	Old/New View Export, Updated 2D
	ZWI	F5	Zoom Window
	ZALL	N/A	Zoom All
	ZFO	N/A	Zoom Drawing Format
	REG	N/A	Regenerate

Files and Windows

Icon	Command	Hotkey	Description
	DOP	Ctrl + N	New
	DAD	Ctrl + O	Open
	RCFA	N/A	Open Recent Files - History
	CLO	Ctrl + F4	Close
	DSV	Ctrl + S	Save
	SVA	N/A	Save As
	DPS	N/A	Save Selected
	DPO	N/A	Insert Objects from File
	SVALL	N/A	Save All Changed
	REOPEN	N/A	Reopen Unchanged Current File
	TXV	N/A	List Text Files
	WIN	Ctrl + 3	Windows
	NDW	N/A	New Document from Current Document
	SWD	Ctrl + TAB	Activate Previous Window
	FCO	N/A	Batch File Conversion
	DEF	N/A	Current File Attributes as Default
	INFF	N/A	Information about Current File Changes
	RESET	Ctrl + Alt + D	Restart of VariCAD
	EXT	N/A	Exit

System Settings

Icon	Command	Hotkey	Description
	CFG	N/A	Complete System Settings
	TLBS	N/A	Toolbar Settings
	CBS	N/A	Create Backup of All Settings
	RBS	N/A	Restore Settings from Backup
	CHU	N/A	Change Current Units – mm, inch

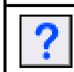
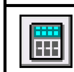
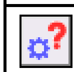
Printing

Icon	Command	Hotkey	Description
	BMP	N/A	Create Bitmap File from 3D
	WPR	Ctrl + P	Print
	WPS	N/A	Print to System Printer
	BPRP	N/A	Batch Print, from Predefined List
	BPRW	N/A	Batch Print, or Define List to Print

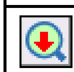
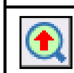
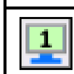
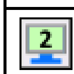
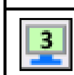
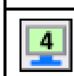
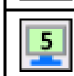
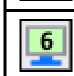
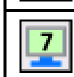
Help

Icon	Command	Hotkey	Description
	QDM	N/A	Tips and Tricks
	HFU	F1	Context Help
	MAN	N/A	Reference Manual
	DMAN	N/A	Display View Functions Related Help
	TIPS	N/A	Tips and Tricks from Manual

Other Functions

Icon	Command	Hotkey	Description
	2D	Alt + 2	Switch to 2D
	3D	Alt + 3	Switch to 3D
	DRP	Alt + S	Sketching
	RED	Ctrl + Y	Redo
	UND	Ctrl + Z	Undo
	INFO	Shift + F3	List of VariCAD Directories
	INFS	N/A	About
	CAL	Shift + F9	Calculator
	HWTEST	N/A	Hardware Test

Saving and Restoring 2D or 3D Views

Icon	Command	Hotkey	Description
	ZSV	N/A	Save View
	ZRE	N/A	Restore View
	RS1	N/A	Restore View 1
	RS2	N/A	Restore View 2
	RS3	N/A	Restore View 3
	RS4	N/A	Restore View 4
	RS5	N/A	Restore View 5
	RS6	N/A	Restore View 6
	RS7	N/A	Restore View 7

	RS8	N/A	Restore View 8
	SV1	N/A	Save as View 1
	SV2	N/A	Save as View 2
	SV3	N/A	Save as View 3
	SV4	N/A	Save as View 4
	SV5	N/A	Save as View 5
	SV6	N/A	Save as View 6
	SV7	N/A	Save as View 7
	SV8	N/A	Save as View 8

Sketching

Icon	Command	Hotkey	Description
	SXDP	N/A	2D Object as Solid X Sketching Plane
	PXDP	N/A	2D Object as Patch X Sketching Plane
	THL	N/A	Thick/Thin 2D Outlines
	ZALLP	N/A	Zoom All in Drawing Plane
	DPV	N/A	Drawing Perpendicular to View
	ZWD	N/A	Zoom Window in Sketching Plane
	VCNI2	N/A	Define View Rotation Center in Sketching Plane

3D Objects Shape Representation

Icon	Command	Hotkey	Description
	OSHELL	N/A	Converts a Solid to Open Shell
	PTCHS	N/A	Converts a Solid to Set of Patches
	PTCHM	N/A	Converts a Solid to Object with Missing Patches

	DOS	N/A	Displays All Open Solids
	DHOS	N/A	Displays All Holes in Open Solids
	DEHOS	N/A	Displays Holes in Open Solids around Deleted Patch

Solving Problems in 3D

Icon	Command	Hotkey	Description
	REGALL	N/A	Regenerate All 3D Objects
	REGTRAN	N/A	Regenerate Transformations
	RECOVERY	N/A	File Recovery
	TOIMP	N/A	Converts to Imported Object, Destroys Creation History
	INSIDEOUT	N/A	Reverse All Normals of Solid
	INSOUTSEL	N/A	Reverse Normals of Selected Patches

Creating Basic Solids

Icon	Command	Hotkey	Description
	PRS	N/A	Box
	CYL	N/A	Cylinder
	PEL	N/A	Pipe Elbow
	TPY	N/A	Pyramid
	PIP	N/A	Pipe
	CPI	N/A	Cone Pipe
	CON	N/A	Cone
	ELW	N/A	Solid Elbow
	SPH	N/A	Sphere

Creating Solids from 2D Profiles

Icon	Command	Hotkey	Description
	RSO	N/A	Full Rotation
	ESO	N/A	Extrude
	RSOP	N/A	Partial Rotation
	MPL	N/A	Lofting, Multiple Profiles Lofting
	HLX	N/A	Helix
	MPLR	N/A	Rotate and Loft between Two Planes
	LB2P	N/A	Loft between Two Planes' Outlines

Editing Solids

Icon	Command	Hotkey	Description
	STC	N/A	Translate, Rotate, Copy Solids
	CHAX	N/A	Permanent Change of Imported Solid Axes
	PAR	N/A	Parameters
	CST	N/A	Geometric Constraints among Solid Elements
	CSTS	N/A	Geometric Constraints among Entire Solids
	RMS	Ctrl + D	Delete Solids or Blending
	CPY	Ctrl + C	Copy to Clipboard
	PAS	Ctrl + V	Paste
	CCUT	Ctrl + X	Delete - Cut to Clipboard
	CS3	N/A	Change Color
	BL3	Ctrl + B	Blank
	UB3	Ctrl + U	Unblank

	MSO	N/A	Edit Solid or Blending
	SHC	N/A	Shade/Wireframe/Transparent Selected Solids
	MIRR3	N/A	Mirror Solid
	RSSO	N/A	Scale Solid

3D Comprehensive Shapes

Icon	Command	Hotkey	Description
	TXT3D	N/A	Create 3D Text
	OFP	N/A	Create Offset Patches (Shells)
	PIPES	N/A	Create Pipes in Space
	WIRES	N/A	Create Wires in Space
	SWP	N/A	Sweep 2D Profile along 3D Path
	THH	N/A	Threaded Hole
	THS	N/A	Threaded Cylinder (Screw)
	OTC	N/A	Outer Thread Cutting Tool

Boolean Operations

Icon	Command	Hotkey	Description
	CUT	N/A	Cut, Delete Cutting Solid
	ADD	Ctrl + A	Add Solid
	CUTS	N/A	Cut, Keep Cutting Solid
	TRX	N/A	Explode Boolean Tree
	CUTPS	N/A	Selective Cut, Delete Cutting Solid
	ADDPC	N/A	Selective Add
	CPSS	N/A	Selective Cut, Keep Cutting Solid

	SIN	N/A	Solid Intersection
	NADD	N/A	Add Solid, Perform No Intersection
	TREE	N/A	Boolean Tree Structure Editing

Common Boolean Operations, Blending

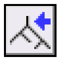
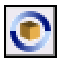
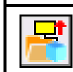
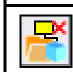
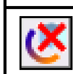
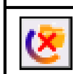
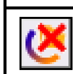
Icon	Command	Hotkey	Description
	HOL	N/A	Hole
	MIL	N/A	Cut by a Box (Mill)
	MILX	N/A	Cut by an Extruded Solid (Mill)
	GRV	N/A	Groove
	RN3	Ctrl + F	3D Fillet
	CH3	Ctrl + R	3D Chamfer

Interferences among Solids

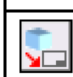
Icon	Command	Hotkey	Description
	CRT	N/A	Interference between Two Groups
	CRTR	N/A	Repeat Interference Check between Two Groups
	CHRD	N/A	Display Interferences
	ASCH	N/A	All Interferences
	ASCHN	N/A	Interference without Selected
	ASCHS	N/A	Interference Selected vs. Rest
	ASCHB	N/A	Interference within Selected

Assemblies and Identical Copies

Icon	Command	Hotkey	Description
	EXV	N/A	Exploded View of Assembly

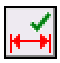
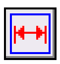
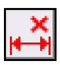
	ASTR	N/A	Assembly Tree Scheme
	OATR	N/A	Assembly Tree Files
	EDE	N/A	Open Part File from Assembly
	EDIA	N/A	Edit Part or Sub-Assembly in Assembly Environment
	RAI	N/A	Regenerate Assembly
	DIA	N/A	Create Part, Save It into New Part File
	DEE	N/A	Define Part to be Inserted into Assembly
	CDE	N/A	Break Definition of Part to be Inserted into Assembly
	ROI	N/A	Change Definition of Part to be Inserted into Assembly
	SBA	N/A	Create Sub-Assembly, Save It into New Sub-Assembly File
	SBE	N/A	Define or Change Sub-Assembly to be Inserted into Assembly
	CSB	N/A	Break Definition of Sub-Assembly to be Inserted into Assembly
	ASI	N/A	Add Solids to Identical Copies
	RSI	N/A	Break Identical Copy Link
	RIC	N/A	Break Identical Copy Group
	CSI	N/A	Break Link from Part
	CAI	N/A	Break All Links from Parts

Creating 2D Views from 3D

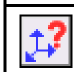
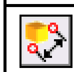
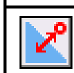
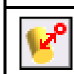
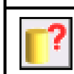
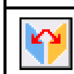
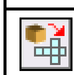
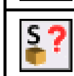
Icon	Command	Hotkey	Description
	32E	N/A	2D View from 3D

	32EN	N/A	Update 2D after 3D Changes
	32SET	N/A	Update 2D after 3D Changes Setting
	SEM	Ctrl + F2	3D Section Management

Checking of Automatically Updated 2D after Changes in 3D

Icon	Command	Hotkey	Description
	HOD	N/A	Highlight Objects Related to 3D View Exports
	OOD	N/A	Highlight Objects Off
	ZOD	N/A	Zoom in on Highlighted Dimensions
	ROD	N/A	Remove Objects Unable to be Updated by 3D View Export

3D Calculations and Check Functions

Icon	Command	Hotkey	Description
	3DCO	N/A	3D Coordinates
	3DD	N/A	3D Distance
	STAT	N/A	3D Space Information
	ODT3	N/A	3D Object Information
	DPP	N/A	Distance Point Plane
	DPC	N/A	Distance Point Cylinder
	SCY	N/A	Cylinder Dimensions
	APL	N/A	Angle between Planes
	SDE	N/A	Surface Development
	VOL	N/A	Volume, Mass, Center of Gravity
	SAR	N/A	Surface Area

	SELSAR	N/A	Surface Area of Selected Patches
	MIN	N/A	Moment of Inertia
	CPP	N/A	Check Pipes
	PATCHI	N/A	3D Patch Information
	CURVI	N/A	3D Curve Information

3D Groups

Icon	Command	Hotkey	Description
	3GR	Ctrl + F1	3D Groups Management

3D Views

Icon	Command	Hotkey	Description
	SRD	N/A	Enhanced Rendering
	ZPR	N/A	Undo View
	ZRD	N/A	Redo View
	VLE	N/A	Left View
	VRI	N/A	Right View
	VFR	N/A	Front View
	VBA	N/A	Back View
	VTO	N/A	Top View
	VBO	N/A	Bottom View
	ISO1	N/A	Isometric View 1
	ISO2	N/A	Isometric View 2
	PRV	N/A	Predefined View
	VCN	N/A	Auto View Rotation Center

	VCNI	N/A	Define View Rotation Center
	X90	N/A	Rotate View Around X 90 Degrees
	X180	N/A	Rotate View Around X 180 Degrees
	X270	N/A	Rotate View Around X 270 Degrees
	Y90	N/A	Rotate View Around Y 90 Degrees
	Y180	N/A	Rotate View Around Y 180 Degrees
	Y270	N/A	Rotate View Around Y 270 Degrees
	Z90	N/A	Rotate View Z 90 Degrees
	Z180	N/A	Rotate View Z 180 Degrees
	Z270	N/A	Rotate View Z 270 Degrees
	ZRD	N/A	Redo View
	SHW	N/A	Shade/Wireframe Entire Display
	RNP	N/A	View to Plane
	3DMR	N/A	3D Mouse Rotation On/Off
	3DMZ	N/A	3D Mouse Pan/Zoom On/Off
	3DMRON	N/A	3D Mouse Rotation On

Bill of Materials, Object Attributes, Title Blocks

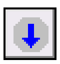
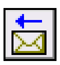
Icon	Command	Hotkey	Description
	BOM	Ctrl + E	Create BOM at Basic Level
	DSS3	N/A	Create BOM Containing Assembly
	BOMG	N/A	Create BOM from 3D Assembly Group
	SPO	N/A	Supplementary Objects
	SAT	N/A	Solid Attributes

	MAT	N/A	Define Mass Density and Material
	DMAT	N/A	Define or Edit Material Table
	MTC	N/A	Change Identical Attributes
	ATL	N/A	Attributes from List
	ATC	N/A	Check Attributes
	AAT	N/A	Assembly/Part Attributes, Fill Title Blocks
	AAT2	N/A	Fill Title Block, Define Attributes
	ATM	N/A	BOM, Attributes and Title Blocks Settings

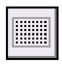
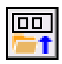
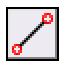
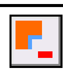
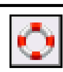
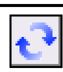
Mechanical Part Calculations

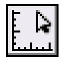
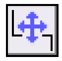
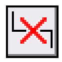
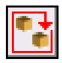
Icon	Command	Hotkey	Description
	TSP	N/A	Tension Spring Calculation
	CSP	N/A	Compression Spring Calculation
	SQK	N/A	Square Key Calculation
	SSC	N/A	Spline Shaft Calculation
	RPC	N/A	Round Pin Calculation
	BCC	N/A	Bolt Connection Calculation
	BEN	N/A	Shaft and Beam Calculation
	FDC	N/A	Spur Gear Calculation
	CDC	N/A	Straight Bevel Gear Calculation
	VBE	N/A	V-Belt Calculation
	RLC	N/A	Roller Chain Drive Calculation
	SKF	N/A	SKF Bearings Calculation

VariCAD on the Web

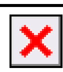
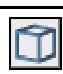
Icon	Command	Hotkey	Description
	INH	N/A	Home Page
	YOUTUBE	N/A	YouTube VariCAD Channel
	FACEBOOK	N/A	Facebook VariCAD Page
	ELCD	N/A	License Code
	TREG	N/A	Registration
	PCHS	N/A	Online Purchase
	INST	N/A	Web Browser Settings
	INN	N/A	What's New Page
	INI	N/A	Upgrade
	INF	N/A	Feedback
	FAQ	N/A	FAQ

Chapter 17. Hotkeys

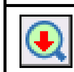
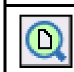
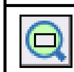
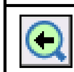
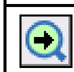
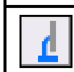
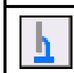
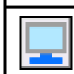
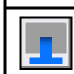
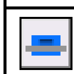
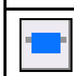
Hotkey	Icon	Command	Description
Ctrl + A		ADD	Add Solid
Ctrl + C		CPY	Select Objects to Clipboard
Ctrl + E		BOM	Bill of Material
Ctrl + G		GRI	Grid
Ctrl + K		BLI	Insert Block
Ctrl + L		LIN	Line
Ctrl + N		DOP	New
Ctrl + O		DAD	Open
Ctrl + P		WPR	Print
Ctrl + S		DSV	Save
Ctrl + T		MOV	Translate or Copy 2D Objects
Ctrl + V		PAS	Objects from Clipboard
Ctrl + W		CUT	Cut Solid
Ctrl + X		CCUT	Delete Objects, Put to Clipboard
Ctrl + Y		RED	Redo
Ctrl + Z		UND	Undo
F1		HFU	Context-Sensitive Help
F2		DSV	Save
F5		ZOOM	Zoom
F6		RDR	Redraw

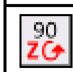
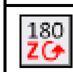
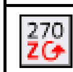
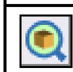
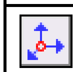
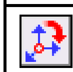
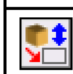
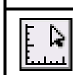
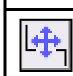
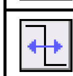
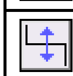
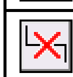
F9		STP	Drawing in Increment Mode
F11		ORT	Drawing in Ortho Mode
Shift + F3		INFO	Information
Shift + F9		CAL	Calculator
Shift + F11		OMO	Turn off Ortho Mode
Ctrl + F1		3GR	3D Groups Management
Ctrl + F2		SEC	3D Sections Management
Ctrl + F4		CLO	Close
Ctrl + 3		WIN	Windows
Ctrl + TAB		SWD	Activate Previous Window
Alt + 2		2D	Switch to 2D
Alt + 3		3D	Switch to 3D
Alt + S		DRP	Sketching

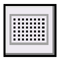
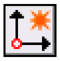
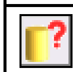
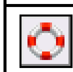
Hot Keys Common for both 3D and 2D Edit Functions

Hotkey	Icon	Command	Description
Ctrl + B		BLANK	Blank Objects
Ctrl + D		DELETE	Delete Objects
Ctrl + C		CPY	Copy
Ctrl + V		PAS	Paste
Ctrl + X		CCUT	Delete Objects, Put to Clipboard
Ctrl + F		FILLET	Fillet Edge/Corner
Ctrl + R		CHAMFER	Chamfer Edge/Corner
Ctrl + U		UNBLANK	Unblank Objects

Chapter 18. Embedded Functions

Icon	Command	Description
	RS1	Restore View 1
	RS2	Restore View 2
	RS3	Restore View 3
	RS4	Restore View 4
	RS5	Restore View 5
	RS6	Restore View 6
	RS7	Restore View 7
	RS8	Restore View 8
	ZSV	Save View
	ZALL	Zoom All
	ZWI	Zoom Window
	ZFO	Zoom Drawing Format
	ZPR	Undo View
	ZRD	Redo View
	VLE	Left View
	VRI	Right View
	VFR	Front View
	VBA	Back View
	VTO	Top View
	VBO	Bottom View

	ISO1	Isometric View 1
	ISO2	Isometric View 2
	X90	Rotate View X 90 Degrees
	X180	Rotate View X 180 Degrees
	X270	Rotate View X 270 Degrees
	Y90	Rotate View Y 90 Degrees
	Y180	Rotate View Y 180 Degrees
	Y270	Rotate View Y 270 Degrees
	Z90	Rotate View Z 90 Degrees
	Z180	Rotate View Z 180 Degrees
	Z270	Rotate View Z 270 Degrees
	RNP	View Perpendicular to Plane
	PRV	Predefined View
	VCN	Auto View Rotation Center
	VCNI	Define View Rotation Center
	SHW	Shade/Wireframe Entire Display
	ODT3	3D Object Information
	SON	Old/New View Export, Updated 2D
	STP	2D Drawing in Increment Mode
	ORT	2D Drawing in Ortho Mode
	ORTH	Ortho Alternating Horizontal/Vertical
	ORTV	Ortho Alternating Vertical/Horizontal
	OMO	Turn off Ortho Mode

	GRI	2D Grid
	UCO	2D User Origin
	STAT	3D Space Information
	3DD	3D Distance
	3DCO	3D Coordinates
	DPP	Distance Point Plane
	DPC	Distance Point Cylinder
	APL	Angle between Planes
	SCY	Cylinder Dimensions
	HFU	Context-Sensitive Help

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