VariCAD

VariCAD

Table of Contents

Chapter 1. A. Reference Manual

Chapter 2. 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 8)* or *Tips and Tricks (page 160).* We recommend that you read this manual while simultaneously using VariCAD.

To learn how to work with VariCAD, use the Tutorial section of this documentation. You can also work with Quick Demonstration, which is at the end of Tutorial. Both Tutorial and Quick Demonstration are available separately from VariCAD pull-down menu Help. The Quick Demonstration Flash movies are obviously not available in the printed version of documentation. If you are not able to run Flashes on your computer, select a version of Quick Demonstration with a VariCAD own animation viewer.

Chapter 3. 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
- 3D parameters
- Geometric constraints
- Mechanical part calculations
- Sheet metal unbending (surface development)
- Crash tests (interferences)
- Pipes
- Thick shells
- Import from and export to other CAD systems
- Support for 3D threads

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

Chapter 4. VariCAD Installation

VariCAD is distributed on CD's or as downloadable files from our FTP or website. To install VariCAD, follow the installation instructions. When installing on Windows, VariCAD uses InstallShield 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 81))*.

Installing Upgrades

Upgrades for one year are included in a maintenance package. After the upgrade period expires, you can purchase another one-year upgrade within another maintenance package. During the upgrade period, you can simply install the new version over the old one, and all system files will be upgraded properly. Your own 2D/3D model files will either remain unchanged or will be automatically upgraded when you start the upgraded version of VariCAD. If you attempt to upgrade after the upgrade period has expired, the installation routine will not work.

Hardware and Software Requirements

For Windows users, we recommend Windows Vista over older versions. For Linux users, we recommend Kubuntu or openSUSE.

The minimum recommended RAM is 128 MB for working in 2D, and 256 MB for working in 3D. Large 3D assemblies need at least 512 MB. Working with insufficient memory can cause swapping problems and a significant decrease in operation speed. For optimum performance, a graphics card that supports OpenGL is required, with a minimum RAM of 16MB. With insufficient graphics card RAM, the 3D display is forced to use software emulation and may be slow. For color settings, use true color mode; 256-color mode or high color mode is insufficient.

There are no special requirements for printers or plotters. Under Windows, VariCAD uses the device drivers. You can also use VariCAD print drivers. VariCAD supports output to all standard output devices - ink, pin, pen or laser. VariCAD drivers support PostScript, HPGL, PCL or EPSON. For Linux, VariCAD uses KDE drivers, or you can choose to use VariCAD drivers.

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 Vista).

The 64-bit version is recommended especially for work with extremely large assemblies.

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 5. Files and Directories Used by VariCAD

Running VariCAD the First Time

On Windows, VariCAD is installed by default to the Program Files directory. 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)
- *.sym - symbol libraries containing 2D symbols (obsolete)

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 149)*. For Linux, VariCAD file extensions must be lowercase.

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

VariCAD supports the following formats:

- *.STP - STEP 3D, import or export.
- *.STL – Stereolithography 3D, 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 2D, import or export.
- *.IGS - IGES 3D, export. If you save IGES file while VariCAD is in 2D mode, only 2D objects are converted. If VariCAD is in 3D mode, only 3D objects are converted.

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. The default IGES settings should be sufficient for most 2D IGES conversions. 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. The following commands enable you to set parameters for converting 3D objects to IGES or STEP:

STEP Export Settings - STPO

IGES Export Settings - IGSO

When exporting to IGES, you can also select the method of patch boundary description. You may want to change this setting if the input for the created IGES file works inefficiently.

Creation of STL format

STL Export Settings - STLO

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.

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.

Directories

 Directories - PTH

This function is used to set directories of VariCAD backup files. We recommend not to use 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.

Chapter 6. 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.

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 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.

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 recommended not to remove 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

Function Windows - Cancel and Back Buttons

Most VariCAD windows have buttons for "Cancel" and "Back." See *Finishing VariCAD Functions (page 9)* or *Stepping Back within a Function (page 9)*. Right-clicking while the cursor is inside the window is equivalent to clicking OK.

Mouse Buttons

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

• Left button - used for selecting objects or defining position

• Middle button - proceeds one step back within a function

• Right button - completes a selection, equivalent to pressing Enter or OK

Invoking and Running VariCAD Functions

To invoke a function, you can:

• Click the toolbar icon. To help you recall icon meanings, tools tips appear when you hold the cursor over an icon. A short tool tip appears next to the icon; a more detailed description appears in the Status Bar.

• Use the Menu Bar. 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 function ends and is replaced by the new one. The F-keys only interrupt the current function temporarily.

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 182)*

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:

• Invoke a new function

• Press Esc

• Click Cancel, if the function uses a window

If you exit a function by pressing Esc, the Status Bar displays "Ready." If the cursor is still within the drawing area, you can right-click to invoke the previous function.

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:

• Click the middle mouse button, or press Ctrl + Backspace

• If there is a window, click "Back"

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 14)*

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.

Example of standard 2D selection toolbar

Example of 2D selection toolbar with additional option icons

For a list of all functions and commands, see *List of All VariCAD Functions (page 161).*

For a list of all hotkeys, see *Hotkeys (page 180)*

Creating, Opening and Saving VariCAD 2D/3D Files
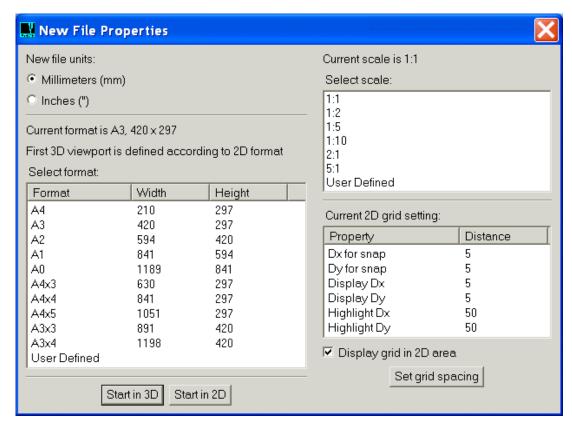
 Create a New File – DOP, Ctrl + N

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.

• Grid - spacing of the rectangular grid in 2D area.

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 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.

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.

Listing 2D/3D Files

When viewing a list of VariCAD files, a 2D or 3D component of the file is displayed in the preview window. You can select whether the 2D or 3D will be displayed. By default, the component that was active when the file was saved will be displayed.

List 2D/3D Files - DRV

This function lists file content in a window larger than standard preview window of the file dialog.

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 savea 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 29)*. Selection of 3D solids is described at *Selecting Solids (page 97)*. 2D objects from another file are inserted similarly as 2D blocks – see *Insert Block (page 66)*. 3D objects from another file are inserted the same way as the solids are transformed and copied – see *Transforming and Copying Solids (page 110)*.

Backing up Your Files

Backup - BAK

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.

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.

Settings of Work with Multiple Files - WMF

You can configure:

• 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.

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 131)*.

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 *Windows and Mouse Settings (page 16)* 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- or right-clicking 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-clicking or pressing Enter ends the dragging movement.

Listing System Information

 Information – INFO

Lists directories currently used by VariCAD.

2D Drawing vs. 3D Modeling

This section contains a comparison of 2D and 3D. For more detailed descriptions of specific functions, see *2D Drawing (page 18)* or *3D Modeling (page 82)*.

When starting a new design project, you need to decide whether to work in 2D or 3D. While there are advantages to both methods, 3D modeling is usually more efficient.

Working in 3D allows you to:

• Create 2D drawings by exporting 3D views

• Update 2D views after 3D changes

• Calculate volume, mass or surface area of solids

• Check for interference between solids

• Use common Boolean and blending functions, such as automatic drilling of holes for screws, and filleting

• Create an exact bill of materials by exporting solid attributes

• Enjoy a clearer display of your work

Working in 2D has advantages as well. Hardware requirements are smaller, and work is faster. If you only need to create a few small drawings that won't be updated or changed, working in 2D is more efficient.

Chapter 7. 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 directories, that do not take effect until the VariCAD session is finished. In these cases, you are informed of this by a message.
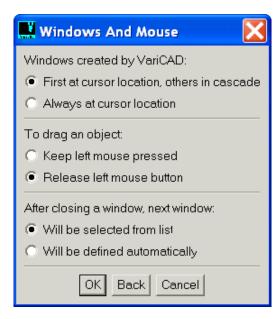
This section describes a few functions for general system settings. Functions used to manipulate settings are described in greater detail in other sections of this help system.

Windows and Mouse Settings

Windows and Mouse - WPST

Sets the position of windows and method of dragging.

Windows and Mouse Settings window

Sound

Sound - SOU

Turns on or off warning sounds. Warning sounds are used when you enter invalid data, if you attempt to close a modified file without saving, etc.

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.

Other Settings

Miscellaneous Settings - OST

Turns on or off the following warning messages:

- If the loaded file contains invisible objects

Chapter 8. 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 - Shift + left mouse button, or right then center mouse buttons. Moving the cursor up enlarges the objects; moving down shrinks them.

• Zoom – using mouse wheel

• 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 Display menu. Zoom functions can also be invoked by pressing F5.

Rebuilding Functions

 Redraw - F6

Quickly refreshes the 2D area.
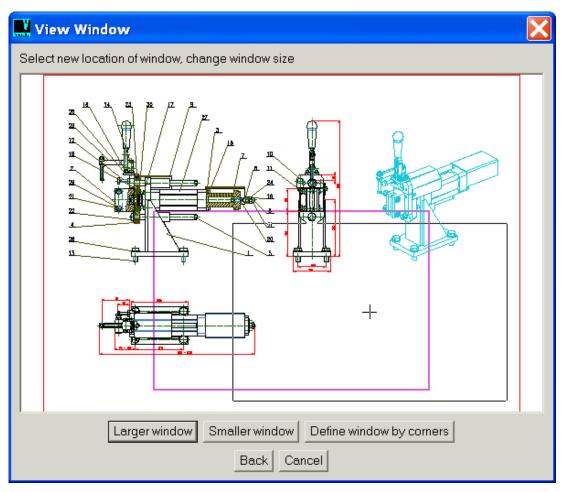
 Regenerate - REG

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.

• Move View Window - the entire 2D area is displayed in a small window and you can define the new view size and position.

• 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.

View Window

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 84)*.

2D Display Settings

Screen Settings

Circle Drawing - SCR
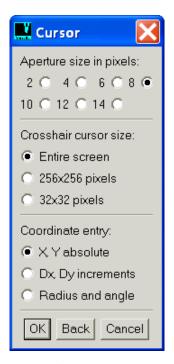
Enables you to increase the number of segments used to draw 2D circles. When the number of segments is high, circles will always appear smooth even when zoomed closely. Increasing the segment number is only available within the current session. A high number of segments can slow drawing speed.

Cursor Settings and Coordinate Listing

⊞ **Cursor – APE, F7**

Sets the type of crosshair cursor, aperture size, and the type of coordinates listed in the 2D Status Bar. The crosshair cursor is used for creating selection windows or when automatic detection of 2D objects is turned off. 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 window

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:

- Point - used mostly as construction aids
- Line - includes single lines defined by two points, multiple (chain) lines, and curves such as splines or ellipses
- Arc - includes arcs and circles
- 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

For more information about creating blocks, see *2D Blocks (page 66)*.

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 10) (page 8)*). 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
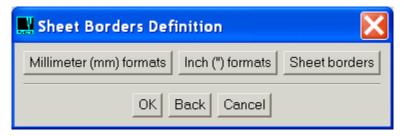
Drawing Format - FMT

Changes the current drawing format. The format controls the view area, sheet border, and print attributes.
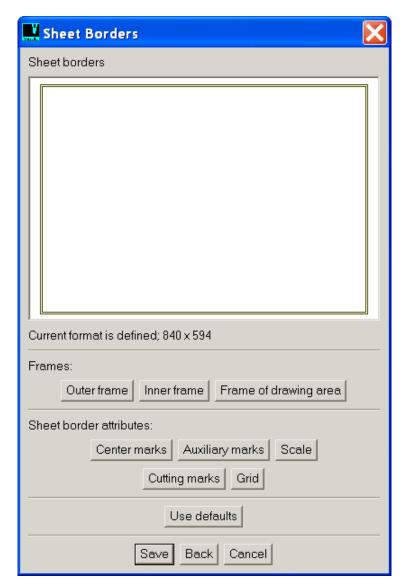
Sheet Borders Definitions - SBD

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.

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:
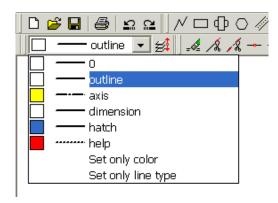
• Layer

• 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.
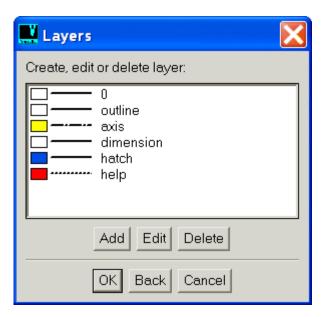
Setting the active layer

Layers - LAY

Creates new layers, and edits or deletes existing layers. You cannot delete the active layer, a layer containing objects, or Layer 0.

Layers window

Automatic Layer Changing - ALS

Automatic layer switching is useful for drawing 2D details. Layers are switched according to the executed functions. Drawing functions like Line or Arc tools create objects automatically placed in Layer "outline." Hatches are placed in "hatches;" dimensioning are placed in "dimension." Layer changing works when you use the default layer setting from VariCAD installation. Otherwise, you must turn Automatic Layer Changing off and select layers manually.

If Automatic Layer Changing is active, the following is the list of default layer names and their behavior:

• 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

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 and 3D objects, there are nine colors you can use. By default, the colors are set as follows:

Sample	Number	Color
	1	white
	2	red
	3	green
	4	blue
	5	cyan
	6	magenta
	7	yellow
	8	dark red
	9	dark cyan

Colors for each number can be redefined using the *Colors (page 17)* or Palettes functions. For printing, line thickness is set according to color number. For color printers, you can map colors to other colors. For traditional pen plotters, color numbers define pen numbers.

Change Color - MPE

Changes the color of selected objects.

Line Types

The following line types are available:

Sample	Number	Line style
	1	continuous
	2	dotted
	3	centerline
	4	short dashed
	5	dashed

—·—··—	6	dash-dot-dot
—·—·—	7	dash-dot

▦ **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.

Locates the user-defined origin while defining the locations of a 2D objects

User Origin - UCO

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

2D Drawing Aids

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

• Grid

• Construction lines

• Ortho mode

• Increment cursor movements

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 in subsequent files, 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 "helper," "auxiliary," or temporary lines, independent of any other objects. You can create individual constructions lines or a mesh, and objects or other construction lines can be placed at the intersection points of other constructions lines. Construction lines can be horizontal, vertical, or diagonal according to two user-defined angles.

The number of construction lines is limited to 100 for horizontal or vertical, and 30 for diagonal. You can define Angle 1 and Angle 2, but if any constructions lines using those angles exist, they must be deleted before the angle can be changed. Construction line functions are available in the Construction Lines toolbar, and from the Objects / Drawing Aids menu.
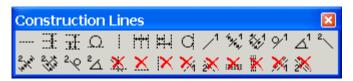
Creating Construction Lines

You can create construction lines as:
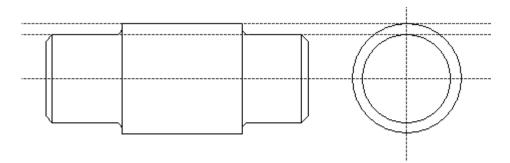
• Single lines that pass through a selected point

• 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 that are tangent to a selected circle or arc

Deleting Construction Lines

You can delete all construction lines, delete individually by type (horizontal, vertical, diagonal), or delete all lines by type.

Construction Lines toolbar

Example: construction lines used to create a side view

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.

Ortho Mode

In Ortho mode, lines are always horizontal or vertical. You can also use Alternating Ortho mode, in which lines alternate between horizontal and vertical, regardless of cursor position. Ortho mode is indicated in the Status Bar.

Ortho On - F11

Turns on Ortho mode. If Ortho mode is already on, Alternating Ortho will be used. Pressing F11 again will change the direction of the next line from horizontal to vertical or vice-versa.

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 Off - Shift + F11

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
	P	Selects a point
	A	Selects an arc or circle
	S	Selects a symbol
	Shift + 6 (^)	Selects an arrow
	C	Selects a hatch
	N	Selects text
	D	Selects a dimension
	1-8	Selects objects in a work set

Selecting Groups of Objects

Icon	Key	Use
	R	Selects objects completely inside the selection window
	I	Selects objects completely or partially inside the selection window
	I	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. You can select from the entire drawing, or use a selection window.

Using Selection Windows (Marquees)

Selection Windows - 2RG

Use this function to 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.

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 46)*.

Automatic Object Detection

With Automatic Detection, the cursor detects objects within the aperture. Press "+" to activate Automatic Detection, and press "-" to deactivate it.

Deselecting Objects

To delete objects from a selection set, active Deselect 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.

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
L	Line endpoint
M	Line or arc midpoint
A	Arc endpoint
X	Intersection
S	Connection point of a symbol or block
@	Center of circle or arc
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 use the keyboard, the desired point must already be within the cursor aperture. The Snap menu also enables you to permanently use certain snaps.

Automatic detection of a snap point

2D Snap Points

Icon	Key	Location
	L	Nearest endpoint of a line segment
	M	Midpoint of a line or arc

Icon	Key	Location
	A	Nearest endpoint of an arc
	5	Center of an arc or circle
	X	Intersection of two segments (intersection point must lie within cursor aperture)
	F	Intersection of two segments (select both segments, can also find intersection of segment extensions)
	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
	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
	O	Nearest point on an arc or line
	Spacebar	Nearest grid point
	C	Nearest intersection of construction lines

Combination Location Points

Icon	Key	Location
	G	Defined distance from the nearest line endpoint
	B	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
	E	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
N/A	I	Snaps to the center of the cursor aperture
N/A	+	Turns on automatic detection of objects
N/A	-	Turns off automatic detection of objects

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). Temporary icons and keys are provided to specify the method of angle definition. The available angle directions are:

Icon	Key	Angle definition
	L	Direction is relative to the line proceeding to the left
	R	Direction is relative to the line proceeding to the right
	D	Direction is relative to the line proceeding up
	U	Direction is relative to the line proceeding down

In some cases, the angle can be defined perpendicular to a selected line segment.

Writing Special Characters

Special characters can be used in dimension text. When setting the dimension text location, you can change the text itself.

Text Input - click this icon to write special characters directly from the window.

The following special characters are always available:

• %%c - diameter sign

• %%d - degree sign

• %%p - plus/minus sign

- %%% - converted to single %

If you use a US keyboard, you can use the following keys:

- ~1 - plus/minus sign

- ~2 - diameter sign

- ~3 - degree sign

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 the following methods:

• By line direction

• By two points

• Between two lines

• By three points

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. The displayed data depends on the type of object:

• For a point - X, Y coordinates

• For lines - endpoint coordinates, length and angle of each segment

• For circles and arcs - centerpoint coordinates, radius, start and end angles

• For text objects - font name and text attributes

• For symbols or blocks - connection points, insertion point and name

• For block - attributes

Drawing 2D Objects

This section describes the various 2D objects you can create. See *Selecting 2D Locations (page 32)* 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:

Endpoints - the default mode, in which each mouse click defines a segment endpoint.

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

Local Ortho - uses Ortho mode only for the current line. Local Ortho enables you to use a rotated coordinate system, as opposed to the standard Ortho mode invoked by pressing F11.

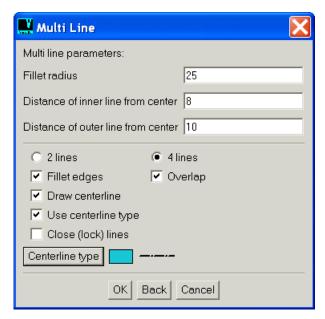Freehand - creates a freehand curve by holding the mouse button.

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

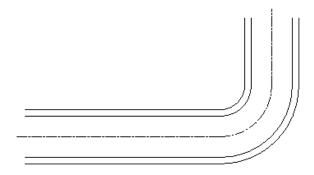One Object - creates one object that contains all the line segments.

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.

Multi Line window

Example of multi lines created within one function

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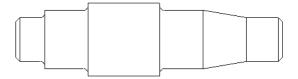
Creates shafts or other symmetrical 2D objects. Before creating shafts, you can define coordinate display, rounding, chamfering or creation of conical or cylindrical parts. When drawing shafts, coordinates are displayed as dX, dY. You can choose whether dX is measured from the first shaft point (total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

Shaft Attributes - change properties during shaft creation.

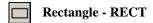Shaft window

Example of object created by Shaft

⊕ Axis - AXIS

An axis can be created in two ways:
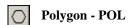
• Two points - the axis is defined as a line between two points, with an extension past the points.

• Arc or circle - a set of two axes will be created at the circle/arc centerpoint.
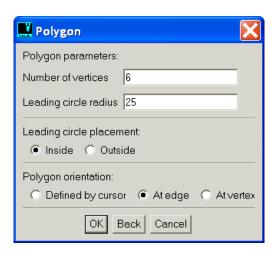
Axes created at center of circle, and between two points

▭ Rectangle - RECT

Creates a rectangle by defining two opposite corners.

⬡ 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 window

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

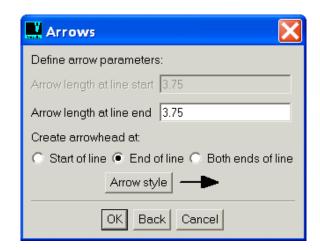An arrow is a single- or multi-segment line that has an arrowhead at the end of its last segment.

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

Arrows window

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

Drawing Curves
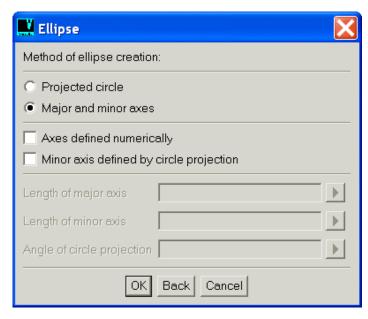
⊞ Graphs - GRA

A graph is a curve defined by x, y coordinates loaded from a text file. Coordinate values stored in a text file must be separated by spaces.
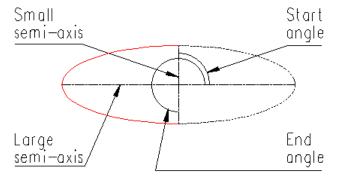
◯ Ellipse - ELL

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 once while entering the first angle, and again (without moving the cursor) while entering the second angle.

Ellipse window

Example of ellipse

Spline - SPL

A spline is a curve defined by a series of points. The spline passes through the defined points. You can choose to create an open or closed spline, and define the number of segments used for curve displaying. Closed curve are connected smoothly – gap between first and last point is spanned. Splines are created as B-spline 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 actually the same objects; a circle is a 360 degree arc. Angles of arcs are measured counterclockwise, with zero degrees along the positive X axis.

Circle Center Radius - CCR

Creates a circle defined by a centerpoint and radius.

Circle Center Point - CCP

Creates a circle by defining the centerpoint 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 centerpoint, 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 centerpoint, 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.
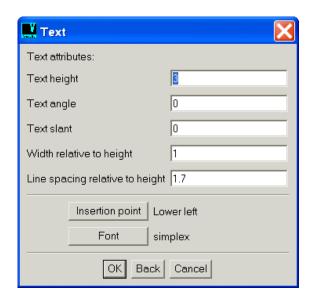
Creating Text Objects

There are several ways you can add text to your drawing: as single text lines, as notes of up to 10 lines, or inserted from a text file. Text can be horizontal or drawn at a specified angle. See *Editing Text (page 48)* 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 (most have fixed width, except for Simplex1)

Text Attributes window

Note (Multiple Lines) - NOTE

A note is a single object containing up to 10 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 the text start point, you can also:
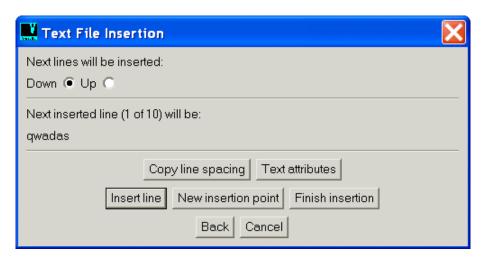
Copy Text Attributes - use attributes from a selected text object.

Insert Text File - TXI

Inserts text from a file into the drawing. Select the file, 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 29)*.

Deleting Objects

Delete 2D Objects – DOB, Ctrl + D

Deletes one or more selected 2D objects.

Changing Objects Geometry

Trim - TBO

Removes sections of lines and 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 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
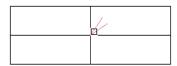
Creates a full circle from a selected arc.
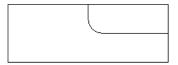
Edit Spline - ESP

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.

Creating Corners, Chamfers and Fillets

Each of these functions can be applied to line or arc 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 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.

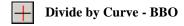## Breaking and Dividing 2D Objects

 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
- Arrows become lines
- Hatches becomes 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 or arc at the selected location, creating two separate objects.

Divide by Curve - BBO

Divides a line 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.
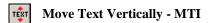
Text Attributes - TAC

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.
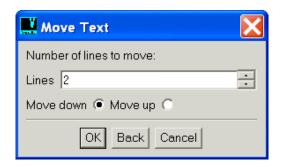
Align Text - JTX

Moves text lines horizontally, aligned to a centerpoint 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

![F icon] **Explode Font - BTF**

Explodes 2D text letters into line segments. This is useful when you want to create a profile from a single letter, to be used in Extrude, for example.

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 either define transformation parameters first and then apply them, or to perform the transformation dynamically. See *Dragging Objects (page 15)*.

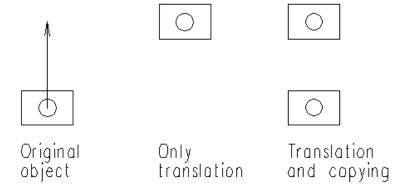## Translating, Rotating and Scaling

![icon] **Translate or Copy 2D Objects– MOV, Ctrl + T**

Moves or copies objects by first defining a 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.

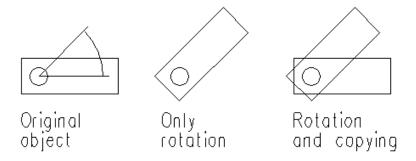Vector Translate window

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 a mirror image of selected objects. 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 use non-isotropic mirroring, the mirrored object will be created on the same side of the mirror axis, and will be scaled perpendicular to the mirror axis. Scaling is only possible for lines.
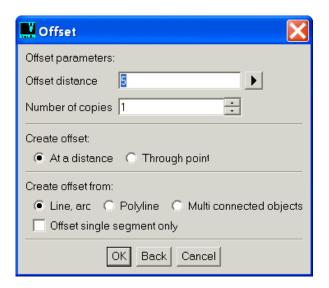
Mirror window

Offsetting Objects

Offset - OFFS

When offsetting objects, you can define an offset distance or create the offset so that it passes through a specified point. You can offset to individual lines, arcs, and polylines, or you can create a continuous offset curve by selecting a chain of connected objects. To do this, select the first line or arc in the chain close to the beginning of the chain. In other words, do not select the segment close to where the next segment is connected.

Offset window

Stretching Objects

Stretch - SOB

Uses the cursor to define the new shape of selected linear objects. Use a selection window to encompass the corners and 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 Ortho mode.
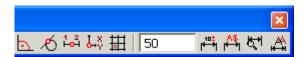
Stretch to Direction - DST

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 translated parts of the objects to be stretched, the objects, and the reference point. Move the cursor to move the reference point. 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 15)*). Dimensioning tools have several options for additional formatting and settings.

The figure above shows additional icons and the text change window in the toolbar.

Once created, the dimensions can be easily edited (see *Editing Dimensions (page 62)*). Format and style of dimensions and dimension text can be modified using *Dimension Attributes (page 57)*. Changing dimension text height affects the length of leader lines arrows.

Single Dimensions - Horizontal, Vertical and Diagonal
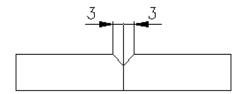
Horizontal Dimension - HDI

Vertical Dimension - VDI

Diagonal Dimension - SDI

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

Dimension Format - 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:

- According to defined input (default) - cancels all other previously selected options.

- Current Dimension Serial - select the dimension you want the current dimension to be aligned with. For example, if you select a horizontal dimension, the current dimension will be in series with it, at the same level. This creates a chain of aligned dimension at the same level.

- Current Dimension Baseline - similar to Current Dimension Serial, except that baseline dimensions are created.

- All Dimensions Serial - similar to Current Dimension Serial, the current and future dimensions are affected, until another option is selected.

- All Dimensions Baseline - similar to All Dimensions Serial, except that baseline dimensions are created.

- Toggle Points and Arrow - switches between points and arrows on the leader lines. This option is useful when space is limited.

- Center Dimension Text - for the current dimension, all texts will be automatically centered.

Dimension arrows changed

Dimension text is created automatically, and you can use *Dimension Attributes (page 57)* to define parameters. For the current dimension, you can change text directly in the text input window in the temporary toolbar. This toolbar contains the following additional options for text modification:

Tolerance - modify the dimension tolerances.

Text Input - add text and special characters. Characters such as the degree sign, diameter sign, or plus/minus sign can also be created directly from they keyboard - see *Writing Special Characters (page 34)*.

Dimension Text Attributes - controls the dimension text properties such as text height, angle, and font. Leader line arrow size is controlled by dimension text height.

To change the text scale, use *Dimension Attributes (page 57)*. This is useful when a rescaled detail is dimensioned.

For diagonal dimensions, the dimension angle is defined by the start and end points. To change this, you can use the following additional options:

Define Dimension Angle - enables you to change the dimension angle.

Automatic Dimension Angle - returns the dimension angle to the default.

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

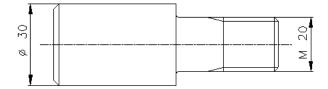**Vertical Diameter Dimension - VDM**

Diagonal Diameter Dimension - SDM

Horizontal Thread Dimension - HTH

Vertical Thread Dimension - VTH
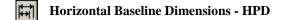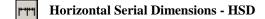
Diagonal Thread Dimension - STH

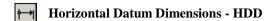Example of predefined dimensions

Serial, Baseline 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.
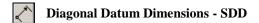
Baseline dimensions are a group of parallel dimensions that all start at the same point. The offset between parallel dimensions is defined in *Dimension Attributes (page 57)*. Datum dimensions consist of a line of points, at each of which the dimension defines the total distance from the start point. Serial dimensions are a chain of aligned dimensions that define the distance from point to point.
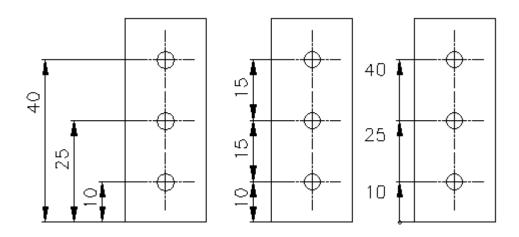
Horizontal Baseline Dimensions - HPD

Horizontal Serial Dimensions - HSD

Horizontal Datum Dimensions - HDD

Vertical Baseline Dimensions - VPD

Vertical Serial Dimensions - VSD

Vertical Datum Dimensions - VDD

Diagonal Baseline Dimensions - SPD

Diagonal Serial Dimensions - SSD

Diagonal Datum Dimensions - SDD

Example of baseline, serial and datum dimensions

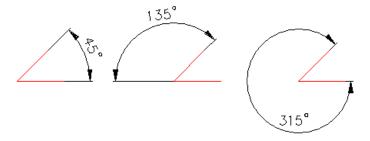# Angular Dimensions

 Angular Dimension - ADI

Angular dimensions define the angle between two lines. You have the same additional options as for the linear dimensions, as well as the following:

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

 Less Than 180 (default) - dimensions the angle less than 180 degrees.

 Complementary Angle On - dimensions the complementary angle.

 Complementary Angle Off - turns off the complementary angle dimensioning.

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 include:

 Mark Arc Center On - creates a mark at the arc center.

 Mark Arc Center Off - turns off marking.

 Thread dimensions - THR

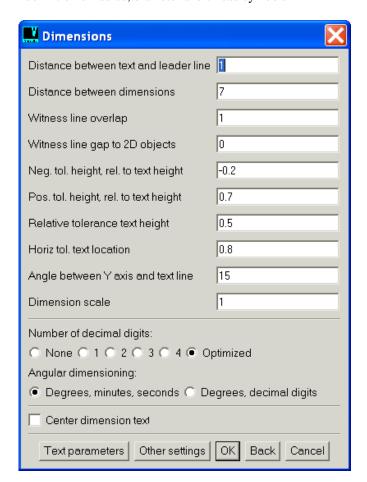Thread dimensions behave like diameter dimensions, except the predefined thread symbol is used.
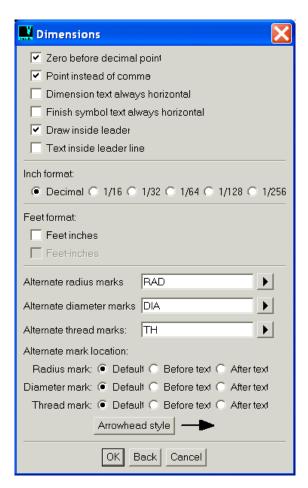
Dimension Attributes

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:
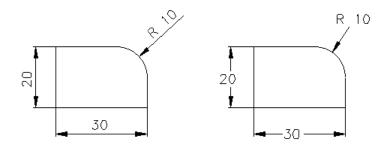
- Text attributes - height, slant and font. Text height controls the size of leader arrows.

- Dimension attributes - distance between parallel baseline dimensions, distance between text and leader lines, text centering, dimension scale, witness line creation, tolerance position, angle dimensioning and number of decimal digits

- Arrow style

- Dimension Style - text angle, position with respect to leader lines, inch and foot representation and definition of radius, diameter and thread symbols

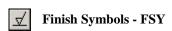Dimensions window; basic settings
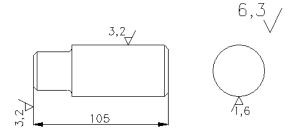
Dimensions window; setting dimension style

Example of dimension styles

Finish Symbols

 Finish Symbols - FSY

Finish symbols are created by entering a roughness value, or by selecting a symbol for unmachined 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.
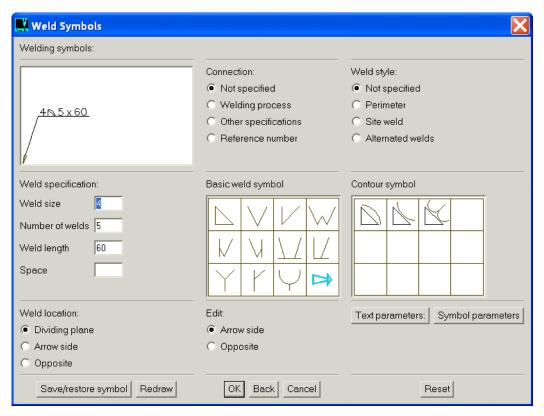
Finish symbols example
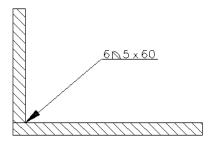
Weld and Tolerance Symbols

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 window

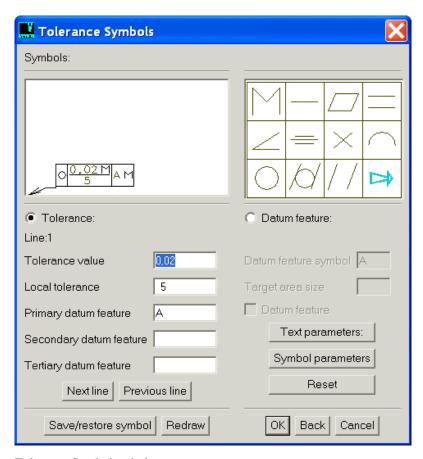Example of welding symbol; this symbol was created within one function.

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 window

Creating Leaders

Leaders - POS

Leaders are typically used in 2D assembly drawings. Leader attributes are defined first, then the location on a 2D object, and then the location of the leader text. Text positions must be defined for the first and second leaders. The dX and dY values of the two leader texts are used to locate texts for the next leader. After the first leader, you need only to define the 2D object for the next leader and confirm the text position, and the value of each leader is automatically increased. As with dimensions, the leader value can be modified in the text window of the temporary toolbar. While defining leader positions, the following options are available:
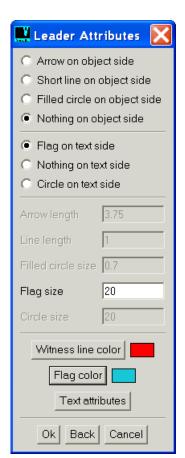
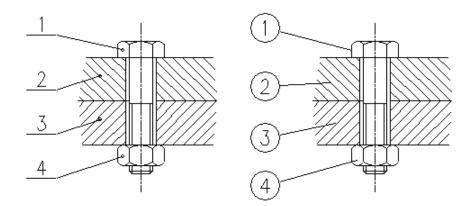Leader Attributes - modify leader attributes

Leader Distance - define distance between adjacent leaders. This setting is useful only when dragging requires the left mouse button to be pressed.

Leader Text Attributes - modify leader text and text properties

Leader Attributes window

Example of leaders

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 and the 2D object itself. One additional option is available:

Edit Multiple Dimensions - modify multiple dimensions, according to the properties of the first selected dimension.

For more information see *Dimensioning (page 52)*

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.

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

Simple Hatch - HAT

Uses a pattern of straight lines. You can define the distance between lines and the line angle. Select each segment of the hatch boundary. For segments that need to be trimmed for the hatch (such as overlapping lines), reselect the segments that are to remain in the boundary. (This trimming is used for hatching only, the original objects do not change.) The following additional options are available during boundary selection:

Create Temporary Boundary Part - creates temporary lines and arcs for hatch boundary segments. These objects will be removed after hatching. You can also select single segments from a multi-segmented line.
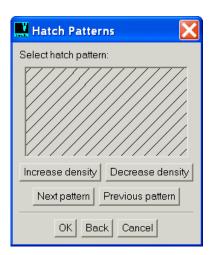
Change Previous Hatch - change previously created hatches.

Pattern Hatch - PHA

Pattern Hatch is similar to Simple Hatch, except that you can select a predefined hatch pattern and specify the density.

Hatch Patterns Window

Simple Hatch - Find Boundary - AHB

Like Simple Hatch, the hatch pattern consists of straight lines. Rather than select each boundary segment, click just one segment. Then select a point on the side of the segment within the desired boundary. You

can continue selecting more boundaries, or add islands. Press Enter or right-click to finish the boundary definition. The following additional options are available during boundary selection:

Set Boundary Detection - ignores any open segments crossing the area boundary, and ignores any small gaps.

Remove Objects Temporarily - ignores selected objects. Select these objects first. Before hatching, the objects will be removed temporarily, and will be reinserted after hatching is complete.

Pattern Hatch - Find Boundary - APH

Similarly to Simple Hatch, Find Boundary, you can select a pattern to fill a detected boundary.

Hatching 3D Sections

Sections from 3D objects that are exported as views to 2D can be hatched the same way as objects created in 2D. In addition, 3D section boundaries can be automatically defined.

Simple Hatch - 3D Section - HSE

The hatch pattern is the same as for Simple Hatch. Select a segment of the 3D section, and the entire boundary is automatically identified. If a 3D section is created from more than one plane, it may be necessary to remove some of the automatically detected lines inside the boundary.

Add/Remove - removes objects from the temporary set. Because the boundary is selected automatically, you can use this option to remove any restraining lines across the area.

For more information see *Selecting 2D Objects (page 29)*.

Pattern Hatch - 3D Section - HPSE

Similarly to Simple Hatch - 3D Section, you can select a pattern to fill a detected boundary.

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.
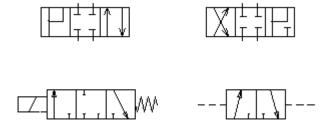
Symbols

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

• An insertion point

• Connection points - used after insertion for snap locations

• Name and comment

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

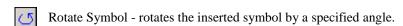Welding, tolerance and finish symbols are managed by other functions, see *Dimensioning (page 52)*.
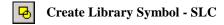
Example of symbols

Insert Symbol - SLI

Inserts symbols from the open library. If no library is open, select the library first. When inserting symbols from a library, you can use icons to select symbols. During insertion, the symbol automatically adjusts its dimension to the current drawing scale. The following additional options are available while inserting symbols:

Rotate Symbol - rotates the inserted symbol by a specified angle.

Scale Symbol - changes the scale of the symbol.

Open Symbol Library - SLA

If you want to create, edit or insert your own symbols, open the symbol library first.

Create Library Symbol - SLC

Create your own symbols and save them to the open library. Define the insertion point and connection points, select symbol objects, and assign a name and comment. You can also choose whether inserted symbols will use their original line attributes, or whether the attributes will depend on the current settings.

Edit Symbol Names - SYC

Edit symbol names and comments, or delete symbols from a library. To add or change the symbol itself, use Create Library Symbol.

2D Blocks

Objects created in 2D can be combined 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 (page 68)*.

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 with the extension *.bkb. 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. 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 29)*.

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.

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

• Rolled profiles (structural steel)

• SKF bearings

• Spline shafts (only for 2D)

• Threads (only for 2D)

The parts are created according to the following standards:

• ANSI

• DIN

• Other standards, including European standards and ISO

All parts have predefined default names and attributes. The names indicate the part definition and basic dimensions according to the chosen standard.

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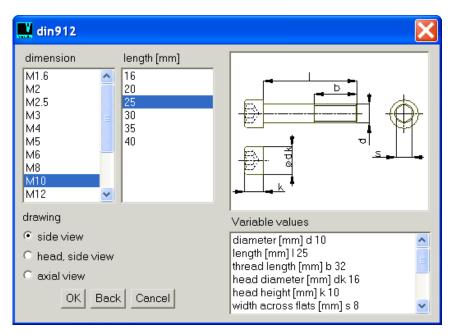
Select a part (screw, washer, etc.) from the Parts menu. The icon menu appears from which you can select a specific part, such as a hex bolt. To define the dimensions, you can select:

• 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.

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 defining a rolled profile 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 66)*.
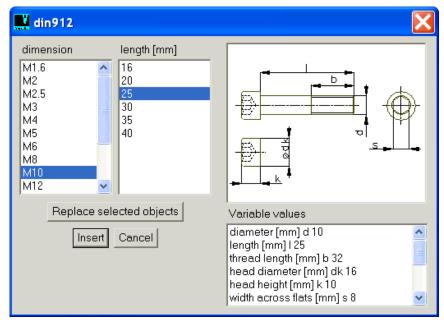
Mechanical Part window - 2D

Inserting Mechanical Parts into 3D

When defining part dimensions in 3D, you can:

• 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 97)*.
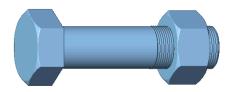

Mechanical Part window - 3D

Working with 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 can be edited without any warning.

Smart Functions using 3D Mechanical Parts

If possible, some mechanical parts have defined method of cutting or trimming other solids. For example, if you want to create hole for screw, you can use screw as "cutting tool" and create hole as exact negative copy of cylindrical part of screw. This method corresponds with creating hole with thread. If you select corresponding modification of counterpart, hole with diameter greater than screw diameter is created. Hole's diameter is created according to respective standard.
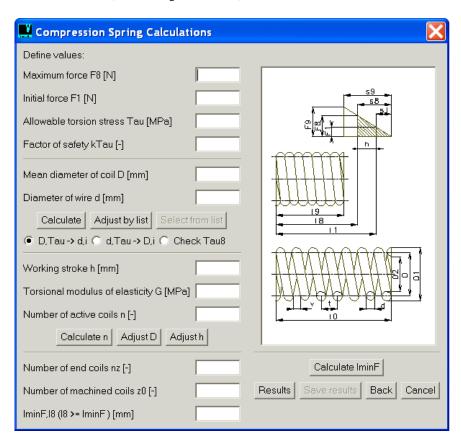
Example of 3D library parts.
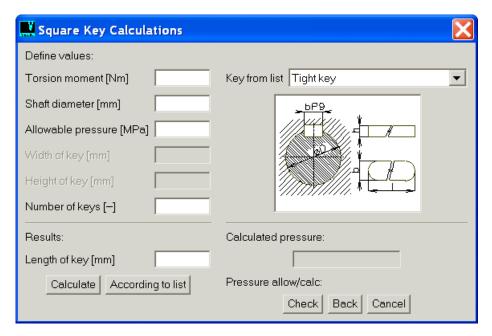
Chapter 10. Mechanical Part Calculations

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:
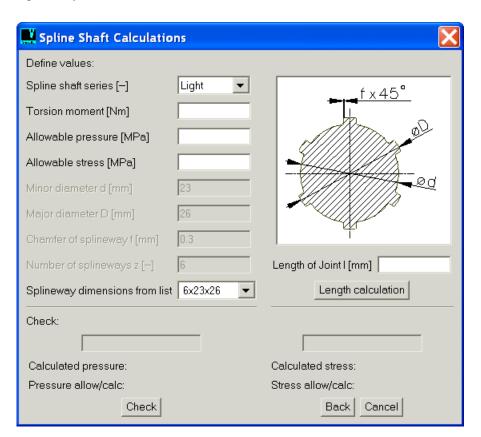
• Compression springs

• Tension springs

• Keys

• Spline shafts

• Screws

• Pins

• Spur gears

• Straight bevel gears

• V-belt drives

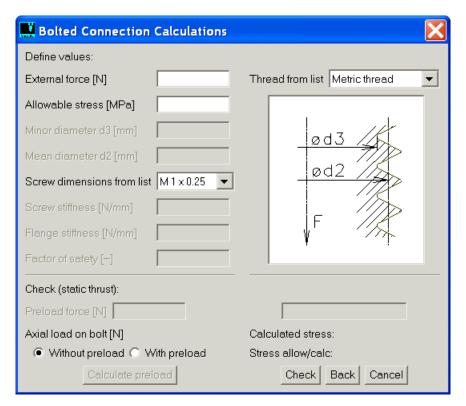• Roller chain drives

• Beams or shafts (for bending and torsion)
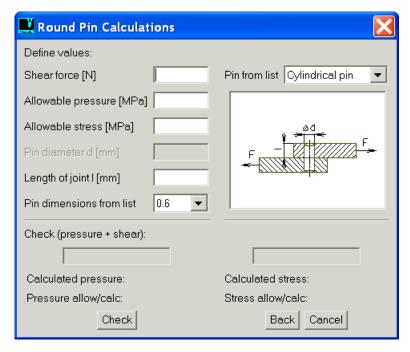

Compression Spring Calculations window

Square Key Calculations window
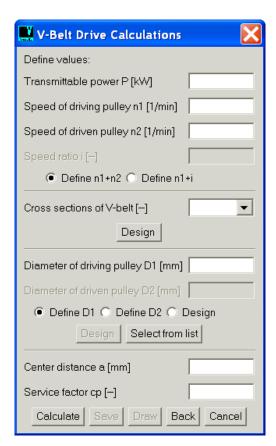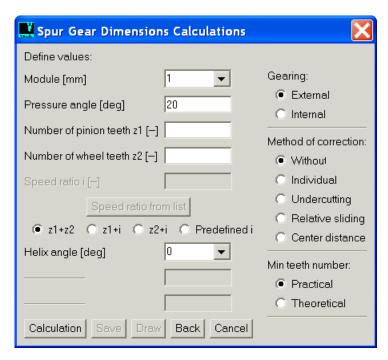

Spline Shaft Calculations window

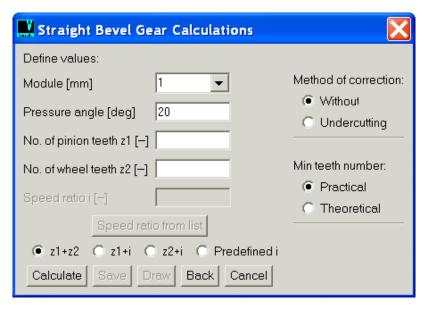Bolted Connection Calculations window

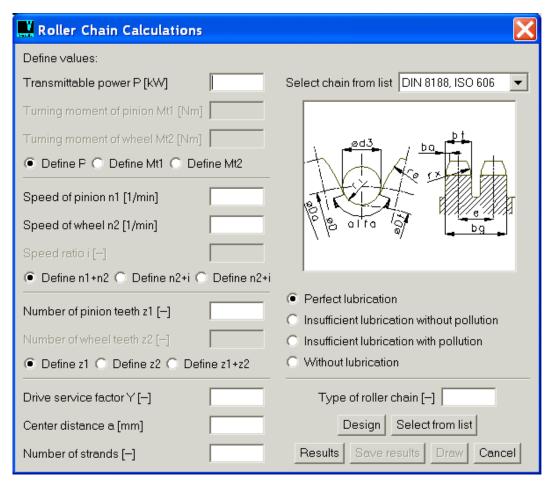Round Pin Calculations window
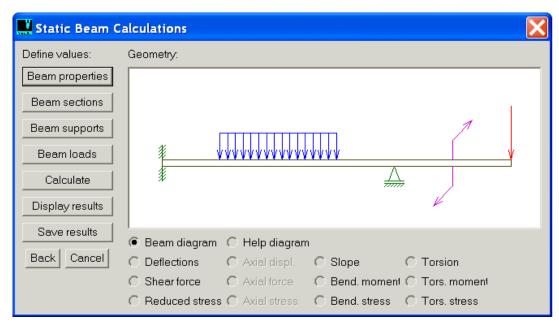
V-Belt Drive Calculations window

Spur Gear Dimension Calculations window

Straight Bevel Gear Calculations window

Roller Chain Calculations window

Beam Calculations window

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 KDE print capabilities. All users can always use VariCAD print drivers as well.

Print Settings

Select the printer first. Then you can define how and what to print. By default, the 2D drawing format is printed at a scale of 1:1.

 Print Settings - WPS

Enables you to define the printed area, scale, rotation, and line mapping.

Defining the Printed Area

There are two ways to define the printed area:

• Entire drawing - 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.

• Only displayed objects - prints only objects that are visible onscreen. You can change the zoom or the view window for the desired print area.

Changing the Print Scale

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

• 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.

Changing the Print Rotation

By default, the area is rotated so that the longer side of printed area will be parallel to the larger side of the printed sheet. There are the 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
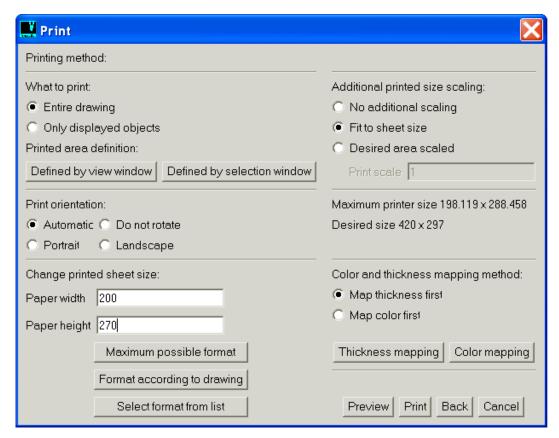
Defining the Printed Sheet Size

By default, the sheet size is the same as drawing format, or the maximum size allowed by the printer. There are the 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 formats such as A1, A2, etc. The selected size is the sheet size of printed sheet.

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 by the Line Mapping function (described below).

Print window

 Line Mapping - LWD

Defines printed line thicknesses or colors.

Selecting a Printer

 Print - WPR, Ctrl + P

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

🖨 **Print, VariCAD Drivers - PRN**

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 means sending more files to print 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 77)*.

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, Predefined - BPRP**

🖨 **Batch Print - BPRW**

🖨 **Batch Print, VariCAD Drivers - BPRV**

Exporting Images as Bitmaps

🖼 **Bitmap File from 3D - BMP**

You can create a bitmap file from 3D display. The image is saved to *.bmp format in true color mode (24 bits per pixel). Bmp file can be used for printing or inserted into documents. Before the file is saved to disc, 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 precise (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. The white background is usually the best option for printed 3D objects. The pixels number (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

If your computer is connected to the internet, you can use the following tools to easily access VariCAD web pages from within VariCAD.

Home Page - INH

Opens the VariCAD home page

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 4)*.

Feedback - INF

Opens a page your can fill out with your feedback

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 keycode that will convert your trial version to a fully functioning 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 keycode

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 methods to change your view dynamically:

- 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 zoom: you can also use the mouse wheel.
- 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.

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

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 Deg - X90

Rotate View Around X 180 Deg - X180

Rotate View Around X 270 Deg - X270

Rotates the view by the specified angle around the global X axis

Rotate View Around Y 90 Deg - Y90

Rotate View Around Y 180 Deg - Y180

Rotate View Around Y 270 Deg - Y270

Rotates the view by the specified angle around the global Y axis

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

Predefined View - PRV

Sets the view according to predefined angles.

Predefined View Settings - PVS

Predefined view is created from front view by rotation around X, Y and Z axes of display. This function defines the rotation angles.

Saving Views

You can save up to eight views that contain rotation, zoom and pan settings. The Predefined Tools tool-bar 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 87)*.
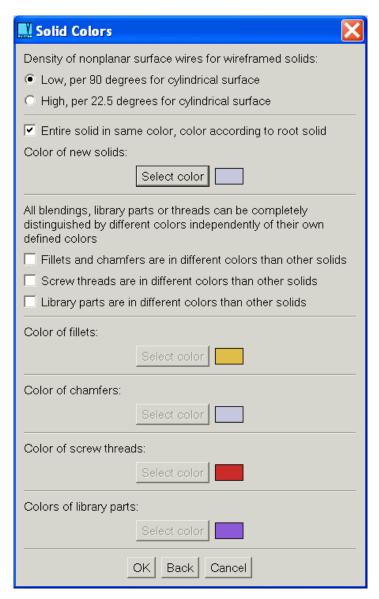
Colors and Wires of Solids - SCO

This function defines:

- Density of nonplanar surface wires. Density setting affects solids displaying, if all space is displayed as wireframed or if a particular solid is displayed as wireframed.

- 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.

- 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

Solid display settings window

 View Rotation - Arrow Keys - S3K

Sets the increments of view rotation when Shift, Ctrl, and arrow keys are used.

 Auto View Rotation Center - VCN

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

 Define View Rotation Center - VCNI

Sets the view rotation center to a specified location.

Precise Displaying

Precise Rendering - SRD

Renders 3D objects precisely. Smoothly rendered images are more realistic and can be used in product presentation materials such as brochures. For this type of rendering, it is recommended to work in true-color mode. You can use a graphics application to capture VariCAD screens. Once selected, precise rendering persists until any 3D edit function is used. It means that you can use all functions working with display, like standard views, view rotation etc. and precise rendering is still present. Some settings are similar to standard display settings, however set values can be different for each display mode.

For precise rendering, you can set the following attributes:
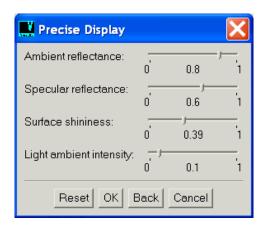
• Surface reflectance. Enables to set light attributes and defines a method of surface shading. See *Surface Shading (page 87)*.

• 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 nonplanar 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. Optionally, you may select a schematic display of threads. Threaded surface is distinguished by a helix curve exactly corresponding to the thread pitch. We recommend turning off the different color of threads if you want to display helixes at threaded surfaces.

Surface Shading

For surface shading, you can set the following attributes:

• Ambient reflectance. Defines darkness of the surface lighted under a great angle (diverted surface). The angle of lighting is angle between ray of light and surface's normal.

• 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 precise displaying with selection of the proper color palette. All values can be easily changed, effect is seen immediately and values can be reset whenever.

Shading surface settings window

Setting 3D Display Performance

OpenGl Settings - OGL

You can tune up performance of 3D objects displaying. This function allows you to select between two different approaches to OpenGl display-lists:

- Safe Mode. This mode is generally better, requires less memory, file open functions are faster and activating any file from the open files is significantly faster.

- Alternative Mode. This mode can be faster in 3D objects redrawing, especially when you rotate, rescale or pan the view dynamically. Alternative mode is not recommended for ATI-based graphic adapters.
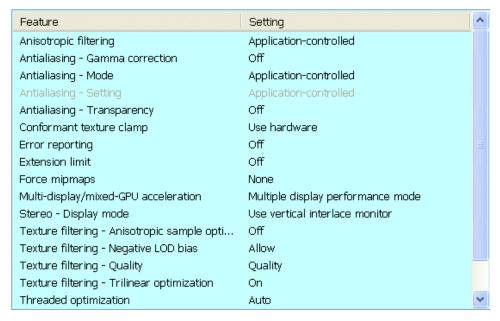
Other settings are related to the method of objects displaying. They can be commonly set to two levels:

- Maximum Performance Settings. This option is default and gives you more comfortable display of 3D objects, especially highlighting detected objects. If the graphic adapter is set to antialiased display, this option is necessary. Otherwise, the objects may not be detected properly (especially the edges alone), or the objects are not displayed correctly, when they are highlighted under the cursor.

- Basic Settings. This option is better for slow computers without an accelerated graphic adapter.
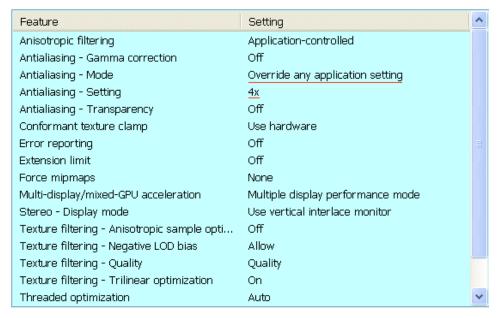
Settings related to the method of displaying can be performed individually, according to the current system behavior. For most of the hardware, the default settings are the best option.

Setting Graphic Adapter

We recommend that you set the graphic adapter (graphic card driver) to obtain the best display of 3D objects. The setting must be done from your operating system and depends on the currently used driver. Usually, the accelerated adapters allow you a lot of settings. The following images show you settings of a NVIDIA adapter. The other types (like ATI) allow you similar settings.
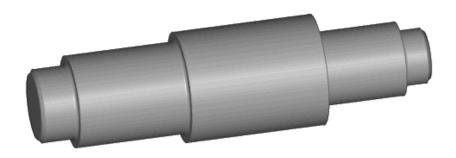
Feature	Setting
Anisotropic filtering	Application-controlled
Antialiasing - Gamma correction	Off
Antialiasing - Mode	Application-controlled
Antialiasing - Setting	Application-controlled
Antialiasing - Transparency	Off
Conformant texture clamp	Use hardware
Error reporting	Off
Extension limit	Off
Force mipmaps	None
Multi-display/mixed-GPU acceleration	Multiple display performance mode
Stereo - Display mode	Use vertical interlace monitor
Texture filtering - Anisotropic sample opti...	Off
Texture filtering - Negative LOD bias	Allow
Texture filtering - Quality	Quality
Texture filtering - Trilinear optimization	On
Threaded optimization	Auto

Typical default settings

Feature	Setting
Anisotropic filtering	Application-controlled
Antialiasing - Gamma correction	Off
Antialiasing - Mode	Override any application setting
Antialiasing - Setting	4x
Antialiasing - Transparency	Off
Conformant texture clamp	Use hardware
Error reporting	Off
Extension limit	Off
Force mipmaps	None
Multi-display/mixed-GPU acceleration	Multiple display performance mode
Stereo - Display mode	Use vertical interlace monitor
Texture filtering - Anisotropic sample opti...	Off
Texture filtering - Negative LOD bias	Allow
Texture filtering - Quality	Quality
Texture filtering - Trilinear optimization	On
Threaded optimization	Auto

Recommended settings

Do not set Antialiasing-setting value to more than 8. Otherwise, detection of 3D edges may not work properly. Without antialiasing, the edges and boundaries of shaded objects are jagged, if displayed under an angle close to 0 or 90 degrees.

Example of the aliased display method

Example of the antialiased display method

Test of Hardware - HWTEST

This function is useful if you want to compare performance of different hardware, like graphic adapters, main boards, processors etc. You can also use this function for comparison of the safe and fast modes set in "OpenGl Settings". Basically, hardware test prepares solids for 3D display and displays them. Time of both tasks is measured.

Results are only informative and can vary with different VariCAD versions. To obtain correct values, use rather large assemblies or more complex objects. For comparison of results, use always the same unchanged file, 3D view, graphic resolution and settings.

2D Drawing in 3D

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

You can select the drawing plane, if you create a new profile (see *Creating 3D Solids from 2D Profiles (page 92)*)

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 wireframed

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

The functions are limited to drawing or editing lines, arcs, curves or points. 2D drawing in 3D does not support any functions 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.

Usage of construction lines is the same as in 2D. Auxiliary grid, however, can be switched between sparse or dense pitch and switched off. The pitch is initially defined automatically according to dimensions of edited profile. You can change the pitch in a command:

Parameters of Aux. Grid - AXGP

Working with 3D

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. Press the following icon in the 2D location toolbar to allow this:

3D location projected into the drawing 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 Drawing Plane - SXDP

2D Object as Patch X Drawing 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 are comprised of 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.

Creating 3D Solids from 2D Profiles

Using 2D profiles to create solids enables you to model a wide range of objects. The solid shape can be edited by modifying the original 2D profile.

Defining a 2D Profile

When using a 3D function that requires a 2D profile as input, you will be optionally switched to the 2D drawing area. You can stay in 3D and create the profile using 2D drawing in 3D. In this case, you can define a drawing plane:

• As an existing plane at a solid

• Plane created by selected axes at a selected solid

• Plane defined by 3 points

• Plane created by selected axes of 3D space

If you create a new profile used for extrusion, rotation or other similar method of solid creation, the created solid is pre-inserted according to the profile's location in 3D space.

If you edit an existing profile of a solid, you will always stay in 3D and the profile will be edited with 2D drawing methods in 3D.

Profiles are comprised of 2D line, 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
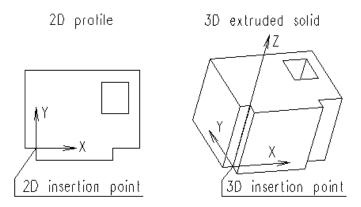
Pressing Enter or right-clicking finishes the profile definition. When the solid is created, you return to 3D space in which you can define the object location. See *Transforming and Copying Solids (page 110)*.

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 with the revolving axis.

Lines, circles or circular arcs can be selected for all type of solid creations. 2D splines (curves) can be selected only for full or partial rotation or for extrusion.

Solid Insertion Point

Prior to selecting a 2D profile, most solid functions require you to enter the solid height or revolving 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.

Example of Insertion Points

Solid Insertion Point Setting - IPST
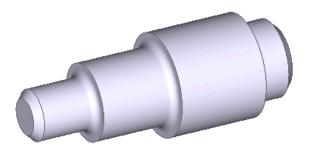
This function is available under System Settings. Although this setting can be done within the profile definition, not all functions offer this. Insertion point setting during profile definition is possible only when the solid height or rotation angle is defined. You can use this function at any time, and it will set the insertion point for subsequent solids. You can also choose whether to define the insertion point and X axis direction automatically.

Revolving, Extruding, and Lofting Profiles

Full Revolve - RSO

Revolves one open or closed profile 360 degrees about a revolving axis. For an open profile, the axis is defined by the profile endpoints. For a closed profile, you must define the axis. If the insertion point is defined automatically, 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.
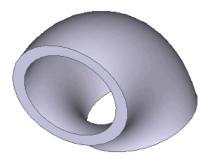
Example of Full Revolve using an open profile

![icon] **Partial Revolve - RSOP**

Similar to Full Revolve, except that you can enter a revolving angle less than 360 degrees.

Example of Partial Revolve using closed profiles

![icon] **Extrude - ESO**

Extrudes, or sweeps, a profile to a 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.
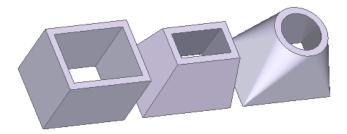
![icon] **Prismatic Loft - PYR**

Creates a solid by lofting one closed profile, or one closed profile with one inner profile, by a specified height toward another, similar profile or profiles. Lofted profiles must be similar - two circles, rectangle of the same orientation, polygons of the same orientation, etc.

![icon] **Loft Rectangle to Circle - PRC**

Similar to Prismatic Loft, this function lofts a rectangle toward a circle or vice-versa. The rectangle can have rounded corners.

Example of Extrude, Prismatic Loft and Loft Rectangle to Circle

Helical Surfaces and Coils

 Coil (Rotated Profile) - PRR

Similar to Prismatic Loft, Coil creates solids by lofting one profile to another while rotating as well. The 2D profile can be open, in which case only one profile is used and the revolving axis is defined by the profile endpoints. With closed profiles, multiple profiles are allowed inside the outer profiles - this will create holes in the solid, and the revolving axis must be defined. The base and top profiles must be similar to one another, as in *Prismatic Loft (page 93)*.
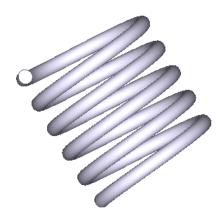
Example of solid coil, created by lofting and rotating profiles

 Helix - HLX

Creates a solid by extruding and rotating a 2D closed profile. Multiple profiles are allowed inside the outer profiles - this will create holes in the solid. 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 will result in a helix with left ascent.

Example of spring helix defined by a normal section

Basic Solids

Basic solids are easy to create; you only need to specify 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.

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
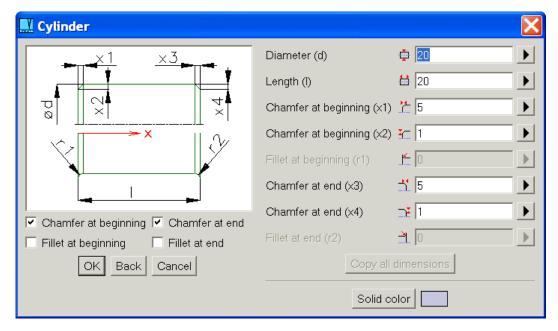
Cylinder window

Editing Solids

Solids can be edited by:

- Changing the location of their components (see *Inserting and Transforming Solids (page 110)*)
- Deleting their components (see *Deleting Solids (page 104)*)
- Adding or removing volume (see *Boolean Operations (page 100)*)
- Changing basic solid shapes (see *Editing Basic Solids (page 105)*)
- 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 wireframed displaying is used or if a particular solid is displayed as wireframed, 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 85)*. 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.
	N/A	Selects an entire constrained group.
Ab	E	Names or Attributes - select objects according to their names or attributes
	N/A	Undo Selection – cancels last selection step
	X	Select/Deselect - switches between adding and deleting objects from the selected set.

3D Selection Settings - SLS

For 3D objects selection, you can modify:

• To highlight wires of nonplanar 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, shading can be time-consuming, and you do not always need to have all items visible. Blanking objects can improve system speed. When loading files, you have the option to be warned when objects are blanked. Blanked objects, however, are not processed in any operations.

Blank - BL3, Ctrl + B

Makes selected objects invisible.

Unblank - UB3, Ctrl + U

Unblanks objects that were blanked.

3D Groups Management – 3GR, Ctrl + F1

Among other possibilities, this function also changes visibility of 3D groups. See *Groups of Solids (page 118)* 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 display in wireframe. 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.
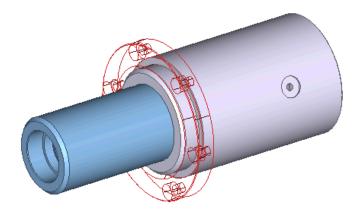
Shade/Wireframe Selected Solids - SHC

First select shaded or wireframe then select the solids to display.

3D Groups Management – 3GR, Ctrl + F1

Among other possibilities, this function also changes shading of solid groups. See *Groups of Solids (page 118)* for details on solid groups.

Example of selective shading

⚏ Change Color - CS3

Changes colors 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 85)*.

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
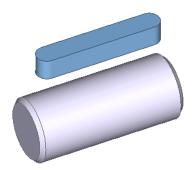
⚏ Add Solid – ADD, Ctrl + A

Combines two solids into one object. First select the solid to be added, 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.
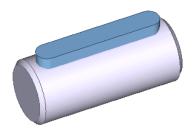
⚏ Cut, Delete Cutting Solid – CUT, Ctrl + W

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.
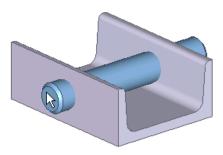
Cut, Keep Cutting Solid - CUTS

Similar to Cut, Delete Cutting Solid, except that the cutting solid is not deleted.
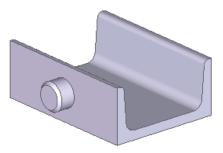
Selective Add - ADDPC
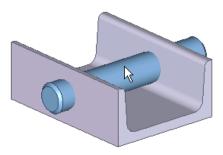
Similar to Add, used to add solids that overlap or extend past the root solid. Select the added solid at the section you want to remain; other sections will be trimmed.
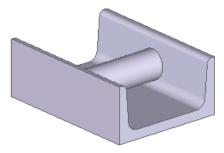
Example of Selective Add. The cylindrical pin is to be added to the U-iron. The pin is selected where indicated.

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.

Common Boolean Operations
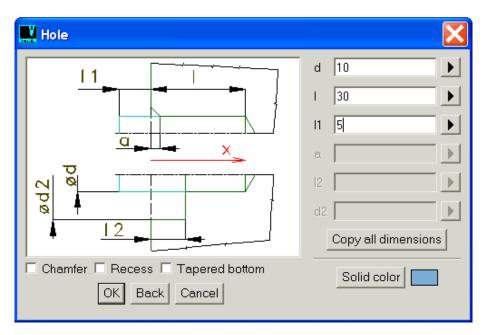
Hole, Mill, and Groove all use a cutting tool to remove volume. The dimensions of the cuttings tools are specified as for basic 3D elements; you can also copy dimensions from another object. For example, in the Hole function, you can copy the dimensions of an existing hole.

Holes, Grooves, Cutting by Planes

Hole - HOL

Remove cylinders (holes) from objects. You can optionally create recesses, or use a chamfered or tapered bottom.

Hole creation window

Mill - MILL

Removes volume by using a box as a cutting tool. If the box dimensions are large enough, you can, in effect, cut by plane.

Groove - GRV

Removes volume by using a spline tool.

Resolving 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.

3D Filleting and Chamfering

When using these blending functions, first select the edges to be filleted (rounded) or chamfered. Selected edges must be continuous. Then specify the fillet radius or chamfer distance. Rounded or chamfered edges are kept if the solid changes during editing. The Fillet function resolves fillet overhangs and undercuts in most cases.

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. Edges rounded previously to version 2007 do not create rounded corners. To change this, edit their properties.

• 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 110)* 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

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 more 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 96)*.

- 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:
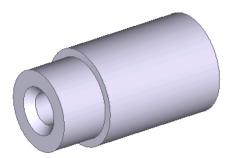
Cursor	Use
	Object cannot be selected – object belongs to different solid than first selected object or object is different than first solid
	Spatial dimensions will be used for object's shape modification
	Object is fillet
	Object is chamfer
	2D creation profile will be edited
	Object is shell, thickness 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
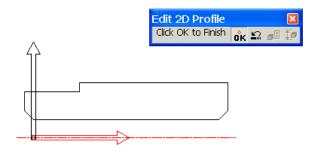
	Pipe segment editing, shape of straight segment or elbow will be changed.

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 92)*. 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.
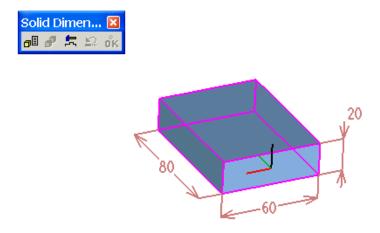
 Rotated object created from a profile
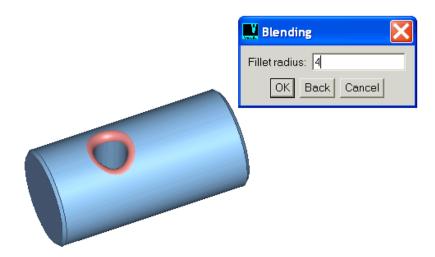
 Editing the object's profile

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 dimensions definition in the table (window). Such an option allows you to change also basic parameters, like add a recess to a hole.

Example of the solid edited using spatial dimensions

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 drilled into a 20 x 20 x 20 cube. If the diameter of the hole changes 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 110)* 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 110)* instead.

Editing of Shells
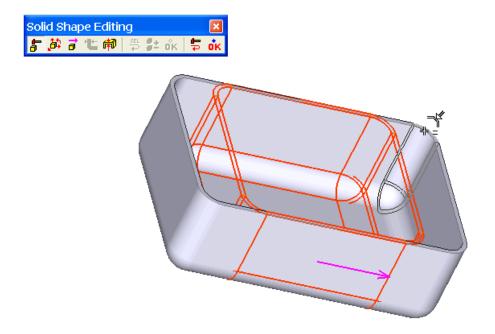
Shell is created as offset patches connected to selected patches at a given thickness. You can change:

- 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 105)*.

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 X, Y, and Z axes of 3D space 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:
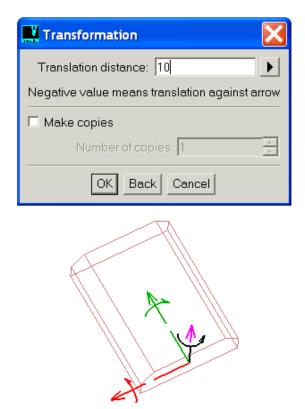
• Two points

- 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 transforming or inserting any objects, 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 in 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 more 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. Pop-up menu appears and you can select dragging, axis direction along or against the defined vector or additional rotation around the corresponding axis. Two more options are also available for each object – inserting into the circle's center or halfway between two points.

Example of solid's transformation axes.

The color of the axes could be different for color palettes. 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 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 right-click to finish translation. 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 117)*.

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 reference point. The reference point is rotated to a plane, which is defined by a rotation axis and a defined point. You can also use the cursor to dynamically rotate around the axis. Press Enter or right-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 117)*.

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

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 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 117)*.

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

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 121)*. 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.

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 110)*.

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 (page 116)*.

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.

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 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 wireframed object. In such cases, approach the plane boundary from inside the plane, proceeding toward the boundary.

3D Locations Settings

3D Location Settings - 3DL

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 112)* or *Dynamic Rotation (page 113)*.

- 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 are 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.
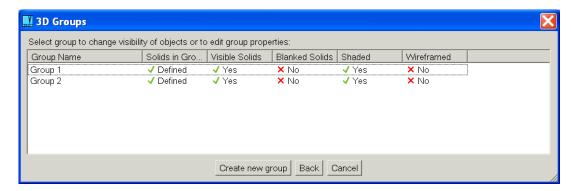
Groups of Solids

Solid groups are identified by a unique name. Each solid can belong to any defined group. When selecting solids, you can also select objects from or in a group. Groups are also useful for changing visibility or shading, and they can be used to define subassemblies.

3D Groups Management - 3GR, Ctrl + F1

Manages groups of solids. You can define new groups, 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

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 35).*

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
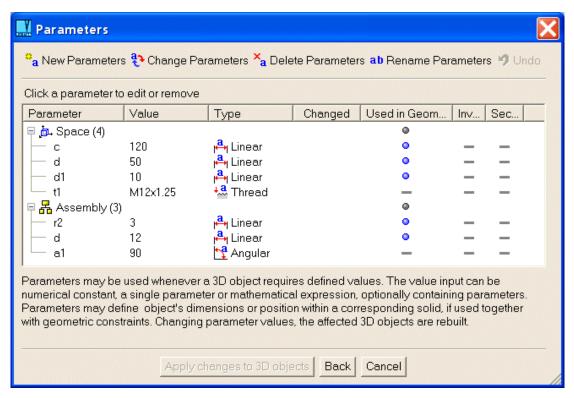
This function allows you to work with all parameters used in the current 3D space and assembly groups. You can:

• 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

Geometric Constraints

Geometric constraints allow you to fix a position of an element inside the Boolean tree. Whenever a shape of the solid is changed, the position of constrained elements is redefined according to a new shape.

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 add 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)
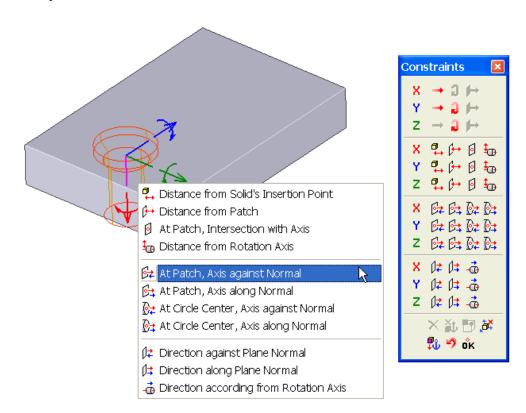
You may work with constraints within a function transforming solids or their elements, see *Additional Boolean Operation, Constraints Definition (page 114)*. The constraints can be created or edited after the location is defined. Another option is to skip a location definition:

Geometric Constraints, CST

This function works with constraints without necessity to define a location.

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.

Constraint definition, Constraint panel

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.

If you select elements of a solid 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

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.

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.

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 both 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 usefull 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
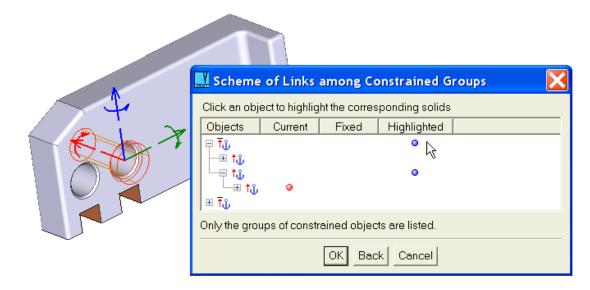
Constraints within a solid cannot be defined without a limitation. Basically, if an object A is constrained to an object B, then the object B cannot be constrained to the object A. All objects can be constrained to an object which is not constrained further.

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 123)*

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.
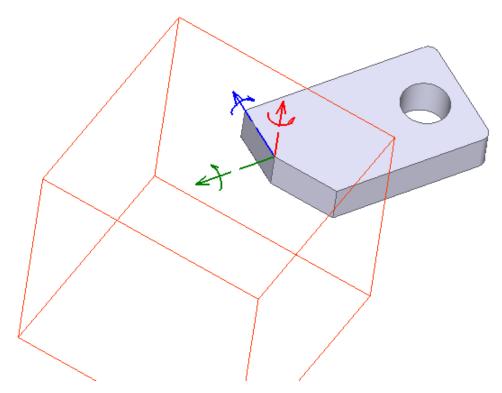
Constraining Angles

To constrain an angle, perform an additional rotation around an axis first.

See *Additional rotation around an axis (page 113)*. As angular value, enter or define a new parameter. Then constrain selected objects, if they are not already constrained.

If you change the corresponding angular parameter, the angle is recalculated and the constrained objects are rotated according to the new angle. The necessary condition for fixing angles by parameters is that the objects must be constrained at least once – no matter which constraining method is used. Otherwise, after 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 an axis
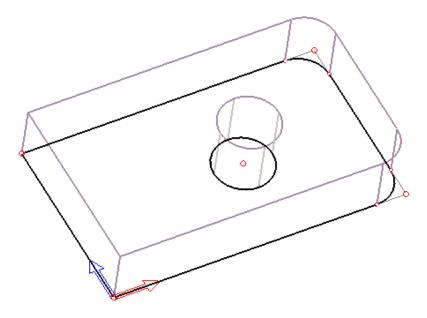
Constraints in Solid Creation 2D Profiles

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
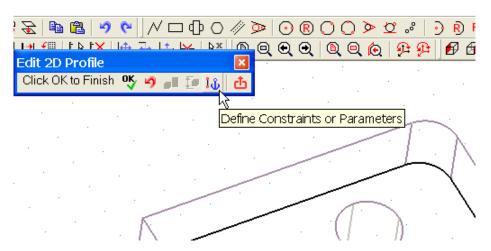
You can define constraints for a group of vertexes. Vertexes are:

• 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, 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.

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 wireframed

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
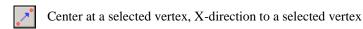
To edit a constraint, select a vertex first. If the vertex is a member of two constrained groups (for instance, a group constrained in x-direction and a group constrained in y-direction), select which group is edited. Add or delete vertexes to or from the selected group. Finally, confirm or edit the constrained distance. It is not possible to change the reference or anchoring point, if the edited constraint is constrained to another vertex.
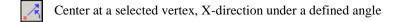
Deleting Constraints

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.

Creating 2D from 3D

2D View from 3D - 32E, Ctrl + X
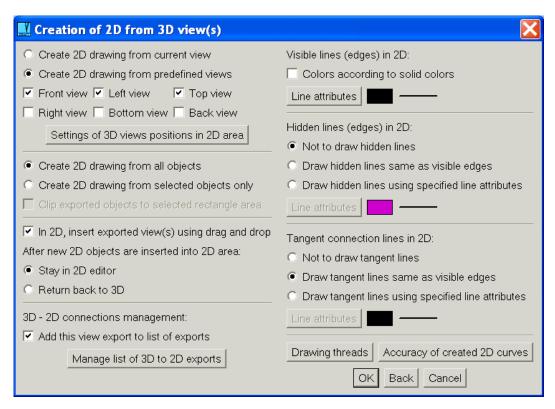
The following options are available for exporting views:

- 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 option, the 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.

- 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 3D Sections (page 64) (page 18)*.

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 window

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 29) (page 18)* - selecting view. If only some 2D objects are transformed, this change is not stored in the list of transformations.

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 64)*.

Update 2D after 3D Changes - 32EN

Creates new 3D-to-2D view exports.
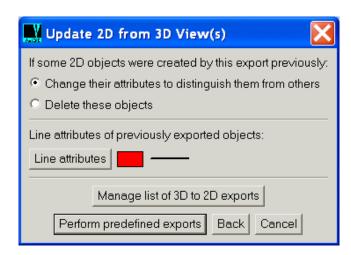
Old/New View Export, Updated 2D - SON

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.

Updating 2D after changes in 3D
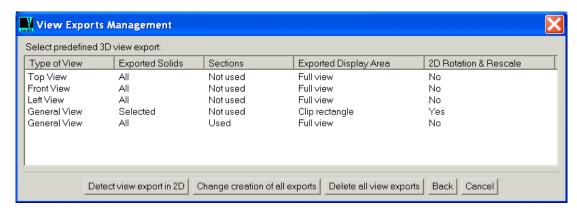
Update 2D after 3D Changes Setting – 32SET

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

Management of defined view exports

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 99)*. 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 6) (page 5)*.
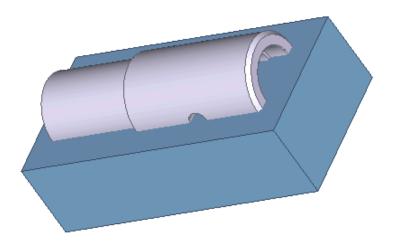
Section Planes, Sectioning Tool

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.
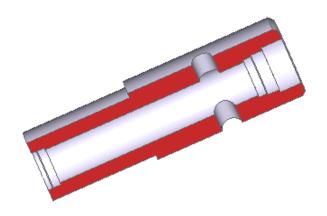
3D Section Management - SEM
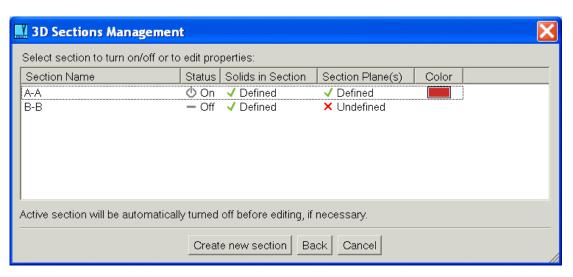
Turns on or turns off selected section. Allows you to define a new section, define the sectioned solids, section planes or cancel definition of 3D section.

3D section - the box is the sectioning tool

Sectioning results (rotated view)

3D Sections window

3D Comprehensive Shapes

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 110)* for basic location modes.
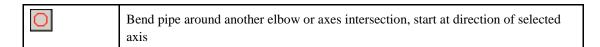
You can define straight segments, while each connection is automatically rounded by an elbow. You can insert an elbow directly at the finished straight segment and define its rotation. You can also bend a pipe around a corner in space as well as around or according to other pipe's elbow. You can finish 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 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, Enter 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, Enter 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
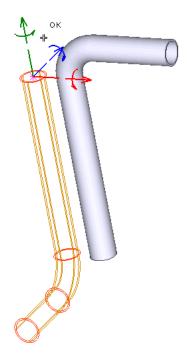
🔧 Pipes - PIPES

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

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:
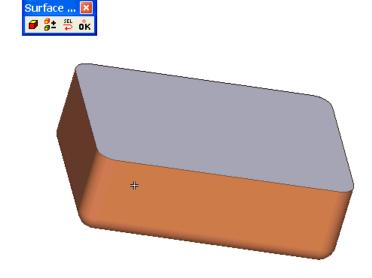
All solid's patches are selected. Then you can some of them deselect.

Shell objects can be used as sheet metal parts.
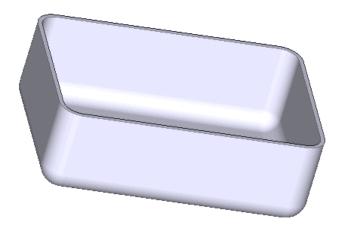
Offset Patches (Shells) - OFP

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 containing threads is to be rescaled, scaling values are limited to available standard thread diameters.
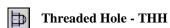
Parts inserted from mechanical parts libraries are correctly fitted with threads, too. The threads are present at screws or nuts inserted from the libraries. Screws and nuts created in versions prior to VariCAD 2008 are not automatically changed to objects fitted with threads.

If you create a threaded hole or outer threaded cylinder (a screw), you can select a thread from a list of threads. There are two types of threads available:

• Metric Threads, for work with ISO units (millimeters)

• Unified Screw Threads, for work with imperial units (inches)

Type of an existing thread can be changed only after a change of the units in the entire document (millimeters to inches or vice versa).
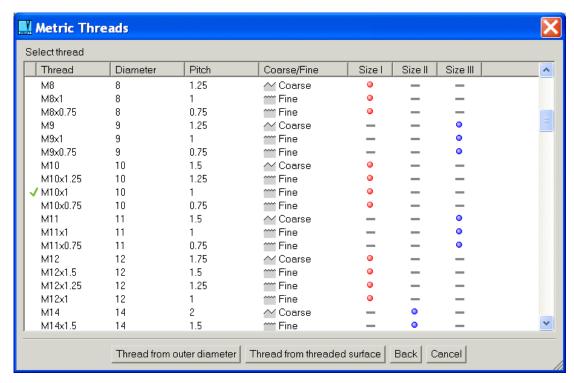
Functions available for thread creation:

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

3D Results Units Settings – RUS

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.

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 116)*.

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.

Interference between 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 wireframed 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 wireframed 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 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 the current file contains any parts inserted from another files, the file becomes as 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.

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.

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.

Simultaneously Open Assembly and Part Files

VariCAD allows you to work with multiple open files. While working in the assembly file, you can open part files from the assembly. Transferring changes between parts is managed by the following function:

Assembly Changes Transfer Settings - DSO

This function enables you to define links between open files - from part to assembly, assembly to part, or both. When changes are transferred, undo-redo history is lost when the file becomes active (windows are switched).

Open Part File from Assembly - EDE

Select the part, and its file opens. If the part in an assembly is changed, changes are transferred into a newly open file automatically only if the transfer is allowed (see above).

Managing the Assembly Connections

Create Link from Part- DIA

Defines an object as a part and exports it to its own file. The current file becomes the assembly, and the part-assembly link is established.

Break Link from Part - CSI

Breaks the assembly link for one group defined by the selected solid.

Break All Links from Parts - CAI

Break all part-assembly links in the current assembly. The current document loses the assembly status.

Create Link to Assembly (Assemblies) - DEE

Defines the selected object as a part. The current document becomes a part file.

Break Link to Assembly (Assemblies) - CDE

Break the part definition. The file is no longer a part file.

Change Link to Assembly (Assemblies) - 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.

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.
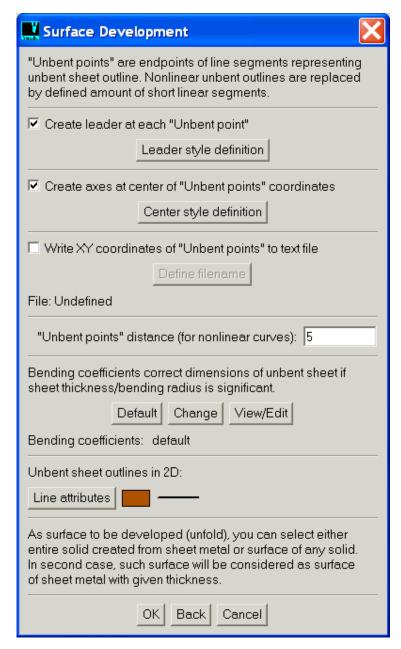
If the sheet is thin enough, you can ignore its thickness. Otherwise, the thickness must be defined and its value is incorporated while calculating the surface development.

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:
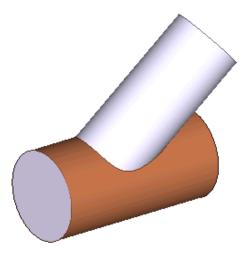
Select the entire solid created from a sheet metal. This option is available only if no other patches are selected. If the entire solid is selected, you can deselect some of its patches, if necessary.
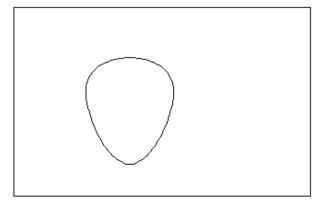
Other available options allow you to switch between select and deselect mode, undo previous selection or finish selection and perform unbending.

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.
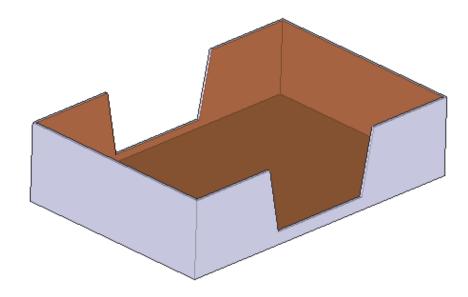
The final step is to drag to insert the unbent surface in the 2D drawing. If necessary, define the surface origin and leader positions.
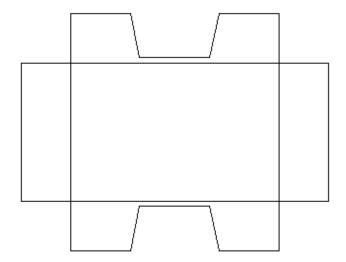
Selection of surface to be developed (unbent)

2D display of developed surface

Selection of more 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 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.

BOM, Attributes and Title Blocks Settings

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 154)*

Attribute Type and Attribute Value

The type of attribute determines the method of attribute definition or other attribute 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:

Ab   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.

   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 149)*. 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:
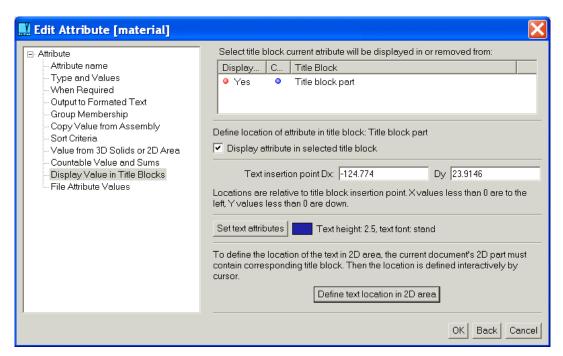
• 2D file with objects representing the title block. Prepare the file first. You can select a corresponding file from the list of files. In this case, the title block must be created with its insertion point at x=zero/y=zero coordinates. You can also select objects of the title block from the current 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

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 take it rather as an example. First time used, the settings are initialized. 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:

Create BOM at Basic Level – BOM, Ctrl + E

Each BOM object is a part belonging to the current assembly (to the current file).

Create BOM Containing Assembly – DSS3

The first object in BOM is the current assembly; next objects are the parts belonging to this assembly.

Create BOM from 3D Assembly Group – BOMG
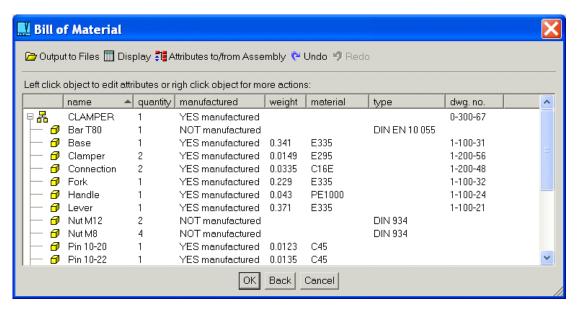
Each BOM object is a part from the current assembly (from the current file), same as the BOM creation at basic level. Only parts belonging to the selected solids group are inserted into BOM.

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 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.

Bill of Material

Output to Files ▥ Display ⬍⬍ Attributes to/from Assembly ↺ Undo ↻ Redo

Left click object to edit attributes or righ click object for more actions:

	name	quantity	manufactured	weight	material	type	dwg. no.
	CLAMPER	1	YES manufactured				0-300-67
	Bar T80	1	NOT manufactured			DIN EN 10 055	
	Base	1	YES manufactured	0.341	E335		1-100-31
	Clamper	2	YES manufactured	0.0149	E295		1-200-56
	Connection	2	YES manufactured	0.0335	C16E		1-200-48
	Fork	1	YES manufactured	0.229	E335		1-100-32
	Handle	1	YES manufactured	0.043	PE1000		1-100-24
	Lever	1	YES manufactured	0.371	E335		1-100-21
	Nut M12	2	NOT manufactured			DIN 934	
	Nut M8	4	NOT manufactured			DIN 934	
	Pin 10-20	1	YES manufactured	0.0123	C45		
	Pin 10-22	1	YES manufactured	0.0135	C45		

OK   Back   Cancel

Bill of materials

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 152)*.

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 158)* 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 154)*

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.

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.

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 149)*. 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 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 149)*. 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.

Assembly Attributes, Title Block Filling

Assembly/Part Attributes, Fill Title Blocks - AAT

Define new attributes, change existing attributes or delete attributes of the current file – either assembly or part file. Attributes are defined according to the *BOM, Attributes and Title Blocks Settings (page 149)*, 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.

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 149)*. 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 the middle and then left mouse buttons simultaneously.

- To zoom, press Shift and the left mouse button while moving the cursor. You can also press the right and then middle mouse buttons simultaneously.

- To zoom, you can also use the mouse wheel.

- 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 the right and then left mouse buttons simultaneously. For large zoom, press Shift, Ctrl and then right mouse button simultaneously. If the left mouse button is used, the speed of view rotation is dependent on dimensions of all visible objects. If the right mouse button is used, the speed of view rotation is dependent on dimensions of a displayed area.

These view shortcuts are always available.

More tips

- To go one step back in a function, press the middle mouse button or Ctrl + Backspace.

- You can open a file by pressing Tab, if you are not currently in a function and "Ready" is displayed in the Status Bar.

- If the current function does not use a window, right-clicking is the same as pressing Enter.

- If the current function uses a window, right-clicking while the cursor is inside the window is the same as clicking OK.

- You can invoke last command (function) by right-clicking the drawing area, if you are not currently in a function and "Ready" is displayed in the Status Bar.

- If you need to enter any value, like distance or angle, you can enter a mathematical expression instead of a single value.

Chapter 16. List of All VariCAD Functions

Drawing Lines and Curves

Icon	Command	Hotkey	Description
	LIN	Ctrl + L	Line
	ARR	N/A	Arrow
	GRA	N/A	Graph
	AXI	N/A	Axes
	RECT	N/A	Rectangle
	POL	N/A	Polygon
	TAN	N/A	Tangent Line
	ELL	N/A	Ellipse
	BOR	N/A	Sheet Border
	MLL	N/A	Multi Line
	SHA	N/A	Shaft
	SPL	N/A	Spline
	LAX	N/A	Axes
	CAX	N/A	Circle Axis

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

Icon			
	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
	PFF	N/A	Points from File

Hatching

Icon	Command	Hotkey	Description
	HAT	N/A	Simple Hatch
	CHP	N/A	Create Pattern
	AHB	N/A	Simple Hatch, Find Boundary
	PHA	N/A	Pattern Hatch

Icon	Command	Hotkey	Description
	APH	N/A	Pattern Hatch, Find Boundary
	HSE	N/A	Simple Hatch, 3D Section
	HPSE	N/A	Pattern Hatch, 3D Section

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
	FSY	N/A	Finish Symbols
	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
	POS	N/A	Leader
	WSY	N/A	Welding Symbols
	TSY	N/A	Tolerance Symbols

Editing 2D Objects

Icon	Command	Hotkey	Description
	DOB	Ctrl + D	Delete 2D Objects
	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	Text Attributes
	EDI	N/A	Edit Dimension Text
	MDT	N/A	Move Dimension Text
	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

Icon	Command	Hotkey	Description
	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
	EDM	N/A	Edit Dimension
	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
	STRVECT	N/A	Stretch by Vector
	CTA	N/A	Array Copy
	DST	N/A	Stretch to Direction
	SOB	N/A	Stretch

Grid, Construction Lines

Icon	Command	Hotkey	Description
	GRI	Ctrl + G	Grid
	HCL	N/A	Horizontal C.L.
	HCI	N/A	Horizontal C.L. Offset
	HCF	N/A	Horizontal C.L. from Point
	HCT	N/A	Horizontal C.L. Tangent
	VCL	N/A	Vertical C.L.
	VCI	N/A	Vertical C.L. Offset
	VCF	N/A	Vertical C.L. from Point
	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
	D1H	N/A	Delete Horizontal Construction Line
	D1V	N/A	Delete Vertical Construction Line
	D11	N/A	Delete Angle 1 Construction Line
	D12	N/A	Delete Angle 2 Construction Line
	DAH	N/A	Delete All Horizontal Construction Lines

Icon	Command	Hotkey	Description
	DAV	N/A	Delete All Vertical Construction Lines
	DA1	N/A	Delete All Angle 1 Construction Lines
	DA2	N/A	Delete All Angle 2 Construction Lines
	DACL	N/A	Delete All Construction Lines

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	2D Area
	2DA	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 Attributes
	ARA	N/A	Arrow Attributes
	DMA	N/A	Dimension Attributes
	FMT	N/A	Drawing Format

Icon	Command	Hotkey	Description
	DCC	N/A	Displayed Cursor Coordinates
	ORTH	N/A	Ortho, Next Horizontal
	ORTV	N/A	Ortho, Next Vertical
	LAY	N/A	Layers
	STP	F9	Drawing in Increment Mode
	ORT	F11	Drawing in Ortho Mode
	UCO	N/A	User Origin
	OMO	Shift + F11	Turn off Ortho Mode
	STO	N/A	Increment Mode Off
	SWS	N/A	Weld Symbol Settings
	SCH	N/A	Change Drawing Scale

System Settings

Icon	Command	Hotkey	Description
	IGSO	N/A	IGES Export Settings
	IGSI	N/A	IGES 2D Import Settings
	STPO	N/A	STEP Export Settings
	STLO	N/A	STL Export Settings
	DWGI	N/A	DWG/DXF Import Settings
	OFIS	N/A	Old 3D Files Import Settings
	WMF	N/A	Working with Multiple Files Settings
	PTH	N/A	Directories
	SOU	N/A	Sound
	BAK	N/A	Backup
	SCR	N/A	Circle Drawing
	APE	F7	Set 2D Cursor
	COL	N/A	Colors

Icon	Command	Hotkey	Description
	SBD	N/A	Sheet Borders Definition
	CHU	N/A	Change Units
	WPST	N/A	Windows and Mouse
	2RG	N/A	Selection Windows
	ALS	N/A	Automatic Layer Changing
	PAL	N/A	Palettes
	OST	N/A	Miscellaneous Settings
	TLBS	N/A	Toolbar Settings

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
	SLA	N/A	Open Symbol Library
	SLC	N/A	Create Library Symbol
	SLI	N/A	Insert Symbol
	SYC	N/A	Edit Symbol Names

Files and Windows

Icon	Command	Hotkey	Description
	DOP	Ctrl + N	New

Icon	Command	Hotkey	Description
	DAD	Ctrl + O	Open
	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
	TXV	N/A	List Text Files
	DRV	N/A	List 2D/3D Files
	RCFA	N/A	Open Recent Files
	EXT	N/A	Exit
	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

Printing

Icon	Command	Hotkey	Description
	WPR	Ctrl + P	Print
	WPS	N/A	Print Settings
	PRN	N/A	Print, VariCAD Drivers
	BPRP	N/A	Batch Print, Predefined
	BPRW	N/A	Batch Print
	BPRV	N/A	Batch Print, VariCAD Drivers
	LWD	N/A	Printed Lines Mapping
	BMP	N/A	Bitmap File from 3D

Other Functions

Icon	Command	Hotkey	Description
	INFO	Shift + F3	Information
	RED	Ctrl + Y	Redo
	UND	Ctrl + Z	Undo
	CAL	Shift + F9	Calculator

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
	REG	N/A	Regenerate
	ZOOM	F5	Zoom
	ZWI	N/A	Zoom Window
	ZCS	N/A	Dynamic Zoom
	PAN	N/A	Pan
	ZALL	N/A	Zoom All
	EAV	N/A	Move View Window
	ZFO	N/A	Zoom Drawing Format
	ZMM	N/A	Zoom Window Coordinates

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

Icon	Command	Hotkey	Description
DISP 2←	RS2	N/A	Restore View 2
DISP 3←	RS3	N/A	Restore View 3
DISP 4←	RS4	N/A	Restore View 4
DISP 5←	RS5	N/A	Restore View 5
DISP 6←	RS6	N/A	Restore View 6
DISP 7←	RS7	N/A	Restore View 7
DISP 8←	RS8	N/A	Restore View 8

3D Views

Icon	Command	Hotkey	Description
	ZPR	N/A	Undo View
	ZRD	N/A	Redo View
2D	2D	Alt + 2	Switch to 2D
3D	3D	Alt + 3	Switch to 3D
	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
	PRV	N/A	Predefined View
	VCN	N/A	Auto View Rotation Center
	VCNI	N/A	Define View Rotation Center
90 X	X90	N/A	Rotate View Around X 90 Deg
180 X	X180	N/A	Rotate View Around X 180 Deg
270 X	X270	N/A	Rotate View Around X 270 Deg
90 Y	Y90	N/A	Rotate View Around Y 90 Deg

Icon		Command	Hotkey	Description
		Y180	N/A	Rotate View Around Y 180 Deg
		Y270	N/A	Rotate View Around Y 270 Deg
		ZRD	N/A	Redo View
		SHW	N/A	Shade/Wireframe Entire Display
		RNP	N/A	View to Plane
		PVS	N/A	Predefined View Settings
		SRD	N/A	Precise Rendering
		OGL	N/A	OpenGl Settings
		HWTEST	N/A	Hardware Test

2D Drawing in 3D

Icon	Command	Hotkey	Description
	SXDP	N/A	2D Object as Solid X Drawing Plane
	PXDP	N/A	2D Object as Patch X Drawing Plane
	AXG	N/A	Aux. Grid On/Off
	AXGP	N/A	Parameters of Aux. Grid
	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 Drawing Plane
	VCNI2	N/A	Define View Rotation Center in Drawing Plane

Creating Solids from 2D Profiles

Icon	Command	Hotkey	Description
	RSO	N/A	Full Revolve
	ESO	N/A	Extrude
	RSOP	N/A	Partial Revolve

Icon	Command	Hotkey	Description
	PYR	N/A	Prismatic Loft
	PRC	N/A	Loft Rectangle to Circle
	HLX	N/A	Helix
	PRR	N/A	Coil

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

Editing Solids

Icon	Command	Hotkey	Description
	STC	N/A	Translate, Rotate, Copy Solids
	PAR	N/A	Parameters
	CST	N/A	Geometric Constraints
	RMS	Ctrl + D	Delete Solids or Blendings
	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 Selected Solids
	MIRR3	N/A	Mirror Solid
	RSSO	N/A	Scale Solid

3D Comprehensive Shapes

Icon	Command	Hotkey	Description
	OFP	N/A	Create Offset Patches (Shells)
	PIPES	N/A	Create Pipes in Space
	WIRES	N/A	Create Wires in Space
	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

Common Boolean Operations, Blending

Icon	Command	Hotkey	Description
	HOL	N/A	Hole
	MIL	N/A	Mill

	GRV	N/A	Groove
	RN3	Ctrl + F	3D Fillet
	CH3	Ctrl + R	3D Chamfer

Interference between 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
	DIA	N/A	Create Link from Part
	CSI	N/A	Break Link from Part
	CAI	N/A	Break All Links from Parts
	DEE	N/A	Create Link to Assembly
	ROI	N/A	Change Link to Assembly
	CDE	N/A	Break Link to Assembly
	EDE	N/A	Open Part File from Assembly
	DSO	N/A	Assembly Changes Transfer Settings
	RSI	N/A	Break Identical Copy Link
	RIC	N/A	Break Identical Copy Group

Creating 2D Views from 3D

Icon	Command	Hotkey	Description
	32E	Ctrl + X	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

3D Groups

Icon	Command	Hotkey	Description
	3GR	Ctrl + F1	3D Groups Management

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
	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

3D Calculations and Check Functions

Icon	Command	Hotkey	Description
	RUS	N/A	3D Results Units Settings

	3DCO	N/A	3D Coordinates
	3DD	N/A	3D Distance
	VOL	N/A	Volume, Mass, Center of Gravity
	SAR	N/A	Surface Area
	MIN	N/A	Moment of Inertia
	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
	SDE	N/A	Surface Development
	APL	N/A	Angle between Planes
	CPP	N/A	Check Pipes
	PATCHI	N/A	3D Patch Information
	CURVI	N/A	3D Curve Information

3D Settings

Icon	Command	Hotkey	Description
	3DL	N/A	3D Location Settings
	3DS	N/A	Shading and Edges Settings
	SCO	N/A	Colors and Wires of Solids
	SLS	N/A	3D Selection Settings
	S3K	N/A	View Rotation - Arrow Keys
	IPST	N/A	Solid Insertion Point 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
	ELCD	N/A	License Code
	TREG	N/A	Registration
	PCHS	N/A	Online Purchase
	INST	N/A	Web Browser Settings
	INH	N/A	Home Page
	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		32E	2D View from 3D
Ctrl + Y		RED	Redo
Ctrl + Z		UND	Undo
F1		HFU	Context-Sensitive Help
F2		DSV	Save
F5		ZOOM	Zoom
F6		RDR	Redraw
F7		APE	Set 2D Cursor
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

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 + 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
	X90	Rotate View X 90 Deg
	X180	Rotate View X 180 Deg
	X270	Rotate View X 270 Deg

	Y90	Rotate View Y 90 Deg
	Y180	Rotate View Y 180 Deg
	Y270	Rotate View Y 270 Deg
	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

Chapter 19. B. Tutorial

Using Tutorial

In this section of manual there are examples of 2D drawing and 3D modeling. Each example contains an animated flash, set of related images and description. Some of the flash frames or separate images contain labels displaying which key or mouse button will be pressed in the next step. Clicking only the VariCAD objects is described; obvious behavior like a left click icons or buttons is omitted. Description of zoom, pan or 3D view rotation is omitted, too. The printed version of the manual contains only static images and description.

All images were scanned in 800x600 resolution. Generally, we don't recommend working in such display resolution. It is the lowest possible for VariCAD. This resolution was selected because all tutorial images can be displayed completely in any higher resolution.

Location of toolbars can be different from your settings – partially because in 800x600 resolution the amount of toolbars is limited and partially because you may change the position or configuration of toolbars.
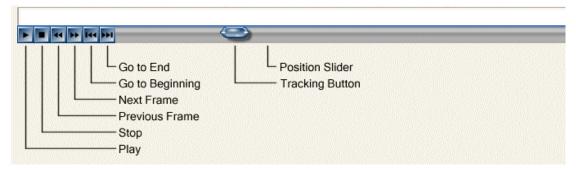
If the system requires user input like a button click or key press, corresponding text is displayed in the drawing area. Omitted are descriptions of obvious inputs like clicking the icons or panel buttons. In some cases, you can click an icon, press Enter or right click elsewhere (usually if you have to finish selection, etc). Not all possibilities may be displayed in such situations.

Working with Flash Examples

To use Flash examples, you need the proper plug-in in your web browser. For the help system under Windows, the proper plug-in must be installed in Microsoft Internet Explorer - regardless of your default browser. Windows HTML help system works with Explorer core.

If you have not installed any plug-in yet, and if your computer is connected to the Internet, it should be installed automatically. If automatic installation fails or if your browser is not able to run flash examples properly, visit the www.macromedia.com and download the flash plug-in.

Controlling Flash Examples
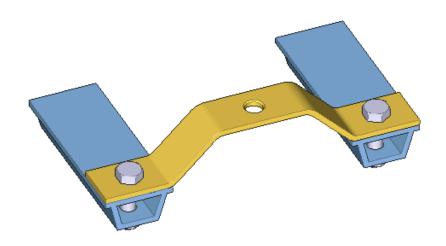
Buttons controlling flash behavior
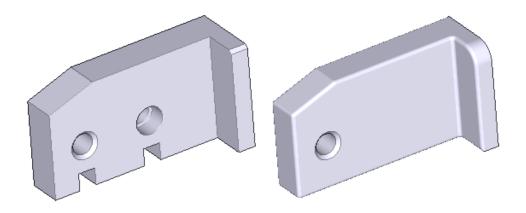
The Flash examples used in the tutorial can be controlled with the following elements:

- Tracking Button. The best method to explore each Flash example is to drag the tracking button. Left click the tracking button, hold the mouse button down and move the mouse. Dragging right displays the next frames (images), dragging left displays the previous frames. Thus you can easily change frames as fast as you need and easily return back, if necessary.

- Position Slider. Click to the left side of the tracking button to display the previous frame. Click to the right side of the tracking button to display the next frame.

- Other buttons. You can also use "play", "stop", "next", "previous", "to beginning" or "to end" buttons. If you play a sequence continuously (click "play" button), each frame is displayed for 1.5 second and then changed.

Objects Created in Tutorial, Corresponding Files

Example 1

Example 2
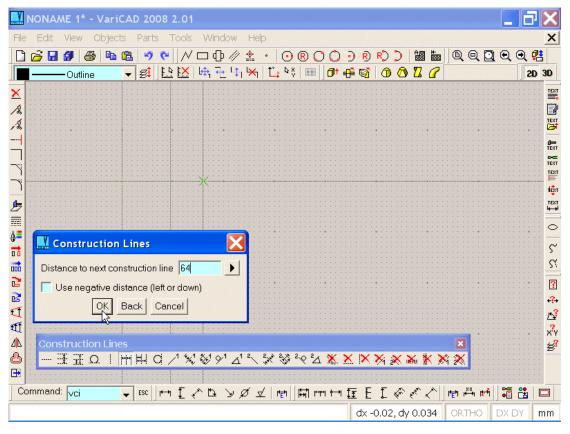
The examples displayed above are built from scratch. The tutorial shows each step of the creation. 2D drawing is demonstrated when creating 2D profiles, which are used for 3D extrusion later. After finishing the creation of the 2D profiles, 3D editing is demonstrated.

Other examples, like creation of 2D from 3D, sheet metal unbending, assemblies or shells use the existing files.

2D drawings and 3D objects created in the particular steps are available in the directory samples¥tutorial in VariCAD installation. File names of the corresponding files appear in the VariCAD window upper frame. The paths of files can be different, however.

Examples of 2D Drawing
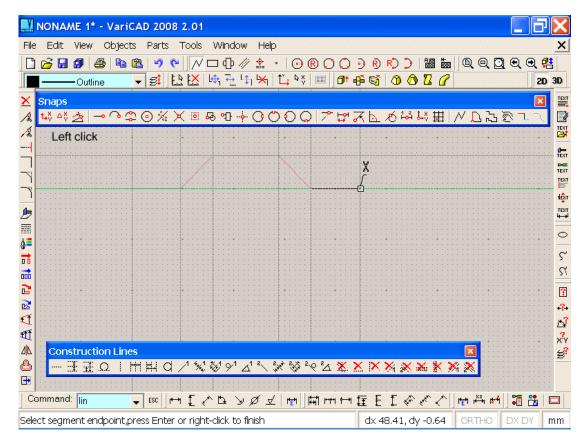
Using Construction Lines

Creating grid of construction lines

When creating any 2D drawing, you can use a grid of auxiliary construction lines – see *Construction Lines (page 27)*. Here, each new vertical construction line is defined by a distance from the previously created construction line. Horizontal construction lines are defined by a distance from the fixed point in the following step. Location of the origin (first point or fixed point) is defined at the cursor nearest grid point in both cases.
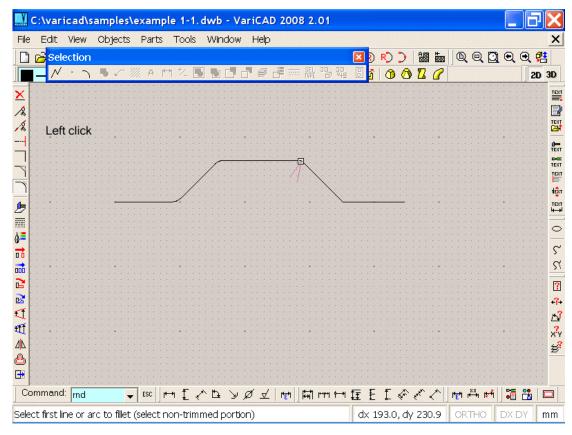
To define a location at the grid point nearest to the cursor, simply press the spacebar. The auxiliary mesh of points is useful for initial location of objects. Coordinates of such a location are rounded according to the grid settings. See *Grid (page 27)*.

Drawing lines using a grid of the construction lines
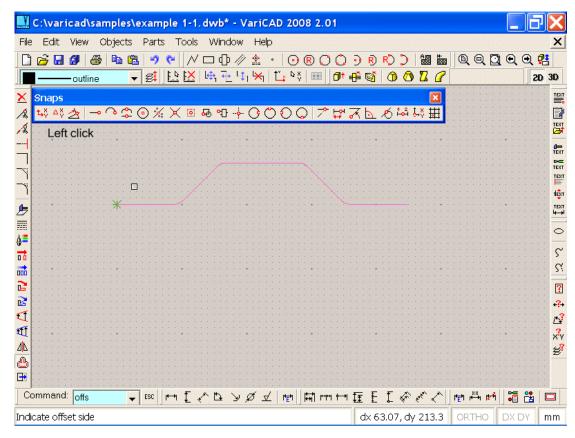
To define a location at the intersection of construction lines nearest to the cursor, you can press the key "c". In this example, a line is created and each segment's endpoint is located at the intersection of the construction lines. Generally, you can select a method of 2D location from "Snaps" toolbar. Automatic detection is available in many cases.

2D Automatic Detection, Drawing Lines, Filleting, Offsets
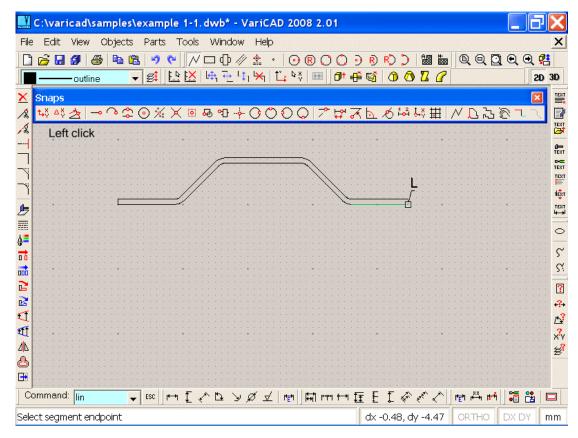
2D filleting

To fillet corners, you can detect them automatically or select both line or arc segments individually. The automatic detection is used in this example. Similar methods are available also for chamfering and creation of corners. See *Creating Corners, Chamfers and Fillets (page 46)*.

2D offset, a curve is temporarily integrated from arcs and line segments

Define an offset distance before the offset is created. In this case, offset is created to the object which is temporarily integrated from line segments and arcs. Click the first segment near its outer endpoint. Then a side of the offset is defined. The side is relative to the first selected segment. Such a segment is marked with the star at the endpoint. See also *Offsets of Objects (page 51)*.
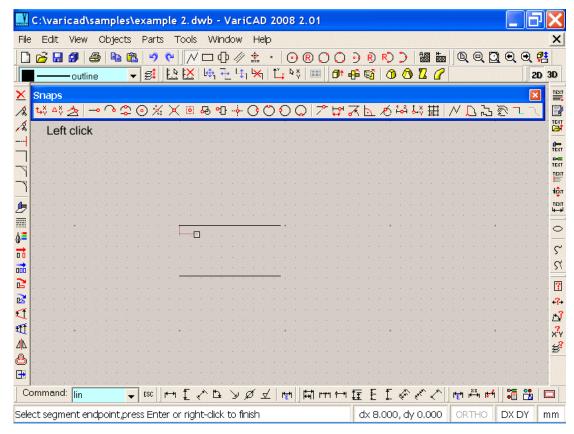
Drawing a line, automatic 2D location

Finally, both curves (original and offset) are connected at the ends. The location of the points of the connecting lines is detected automatically.
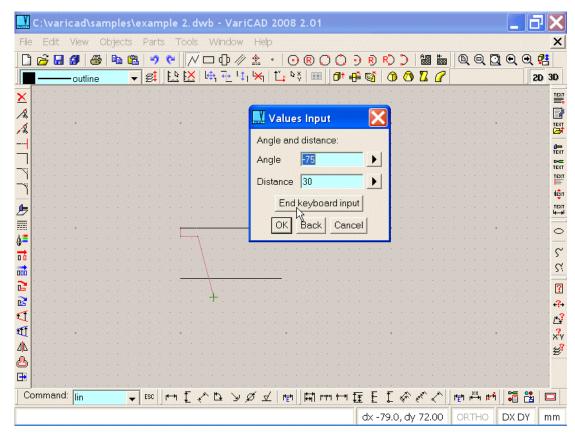
To learn more about various methods of 2D location (definition of line endpoint, arc center, arc endpoint, point on 2D NURBS curve etc), see *Selecting 2D Locations (page 32)*.

Drawing in Cursor Increment Mode and Keyboard Location Input

Drawing in the cursor increment mode
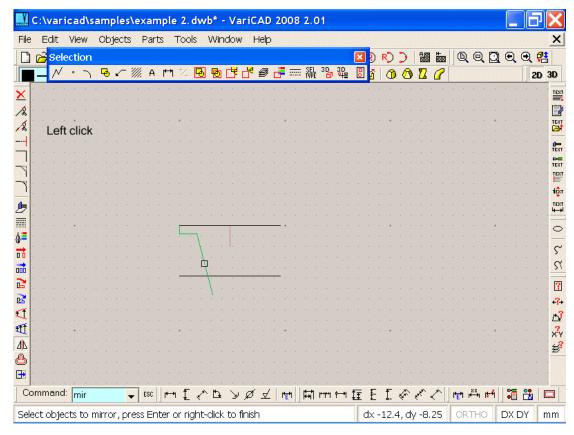
The cursor increment mode is a very convenient method of 2D drawing. Located positions are rounded to the defined values. The cursor increment mode can be set, turn on or off whenever you click the corresponding icon. If used, the increment mode is signaled in the status bar as highlighted "DX DY". You can check the exact cursor position in the status bar when drawing a line or defining a 2D location. Consider, whether you need to display the distance from the last 2D enter or relative to the user defined coordinates origin. In this particular example, the cursor increments are set to 1 mm and displayed coordinates are x and y distance from the previous enter.

Keyboard location input
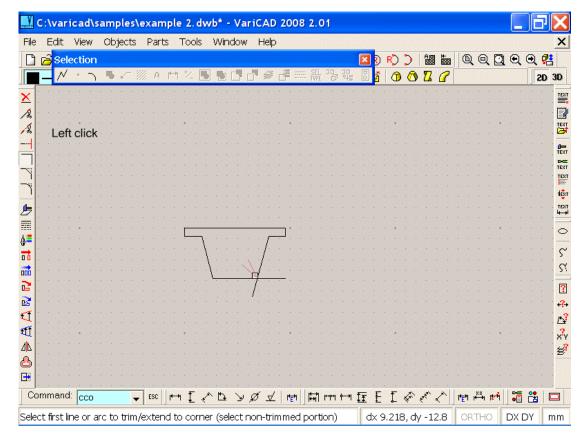
If you need to define a 2D location as x-y distance from the last enter, in the absolute coordinates, or at the distance and angle from the last enter (like in this example), select the corresponding mode from the toolbar or press an accelerating key (D, K or R) – see *Selecting 2D Locations (page 32)*. It is necessary to finish the keyboard input before line drawing itself is finished.

2D Mirroring, Creating Corners
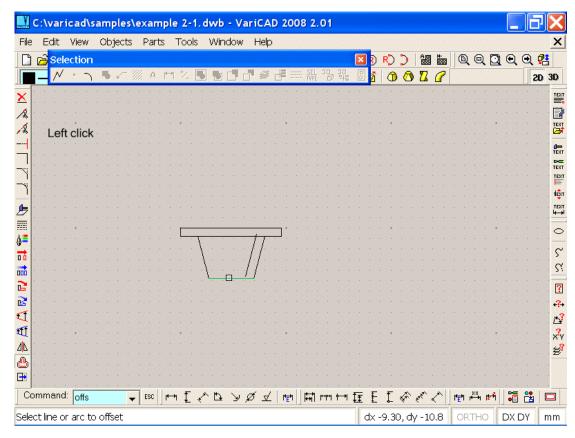
Mirroring

To create the right side of the profile, the left side is mirrored. Still in the cursor increment mode, you can easily define the vertical mirror axis.
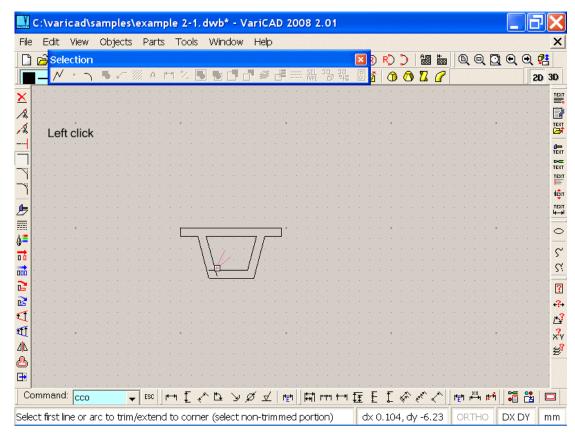
Creating a corner

Exceeding segments are cut off by creating a corner. Automatic detection of the corner is similar as in the fillet or chamfer functions (*Creating Corners, Chamfers and Fillets (page 46)*).

Offset of a single segment at the defined distance
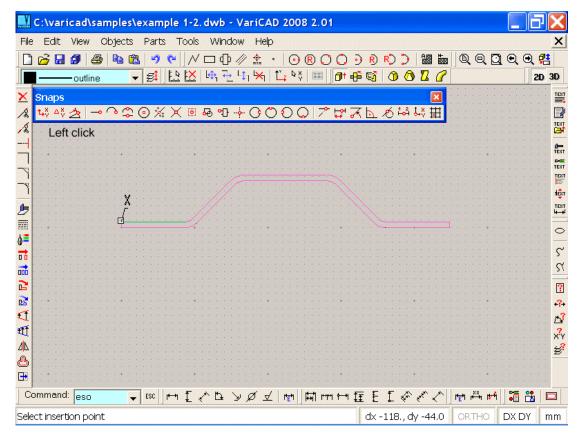
Creating a corner

The inner part of the profile is created as a line offset to the outline. Again, the exceeding segments are cut off by the Corner function. Another method how to cut off such segments is, for instance, trimming by a line or arc segment. After trimming the segments, the exceeding segments are cleared from display. If any other objects are lying under them, they are temporarily blanked. To fix this, redraw the display by pressing "F6" or, very easily, by turning the mouse wheel one notch forward and backward.

Examples of 3D Modeling
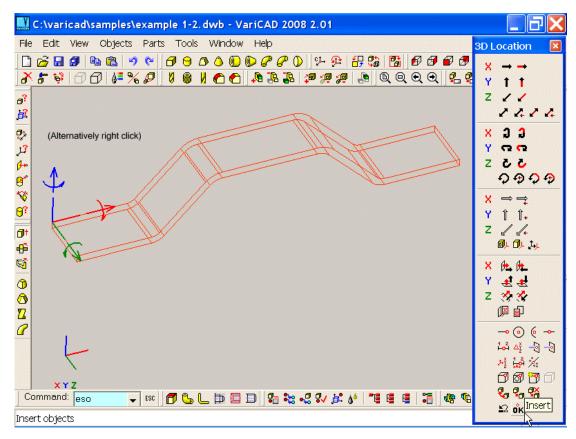
Creating Solids, Profile Extrusion

Solid's insertion point definition relative to the extruded profile

To create a solid by the profile extrusion, prepare a corresponding 2D profile first. Select 2D objects to define the extruded profile; then define the height of the extrusion and the insertion point. Automatic detection of the entire profile is used as a method of the profile selection in this example. For more information about the profile selection, see *Defining a 2D Profile (page 92)*. For more information about the solid insertion point, see *Solid Insertion Point (page 93)*.

Inserting New Solids into 3D Space
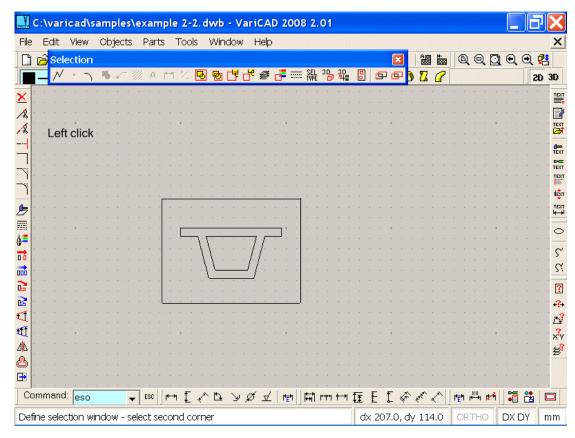
New solid insertion

When inserting objects into 3D space, you have a lot of functions allowing you to define the object position. Some of them are described in the next examples. By default, the objects are inserted into the same location the previous objects were inserted into. Then you can transform them to another desired location. Transformation of the solids is described in the reference manual at *Inserting and Transforming Solids (page 110)*.

In this case, there are no previously inserted objects. Thus the solid is inserted into the coordinates' center (0, 0, 0). In some situations, new solid can be inserted out of the viewport (displayed area). This can happen if you perform any solid insertion and then change the angle of the view, zoom or pan. For a beginner, this can be confusing. In such a situation, simply perform "zoom all", or locate a new object insertion point at any visible or desired location. Then the new objects will be transformed, and at least a part of them will be displayed in the current viewport.
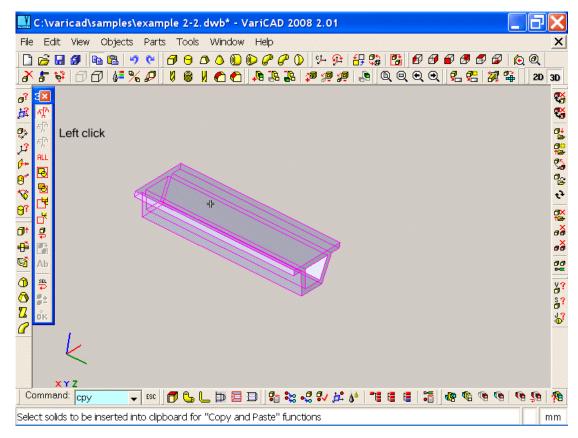
If you work in 3D, you need to change the display frequently. See *Tips and Tricks (page 160)* or *3D Display (page 82)* how to perform this.

Selection of another solid profile

The file containing 2D profile for another solid is open as a second file in this case. The solid profile is selected by a rectangular area - all objects completely inside the defined rectangular area will be extruded.
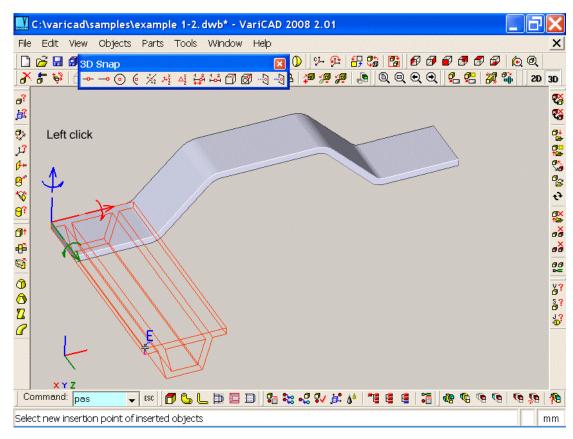
To work with multiple files (or, by another word, documents) is always convenient. The documents were switched from menu "Window" in this example. If you work only with two documents, you can switch between them pressing "Ctrl + TAB". In this case, each file contains 2D profile. Solids are extruded from both profiles.

Selection of the solid for copy and paste

Another extruded solid was created. This solid is inserted into the clipboard to be used later for the "paste" function. Then the files are switched again and this solid is inserted from the clipboard into 3D space.
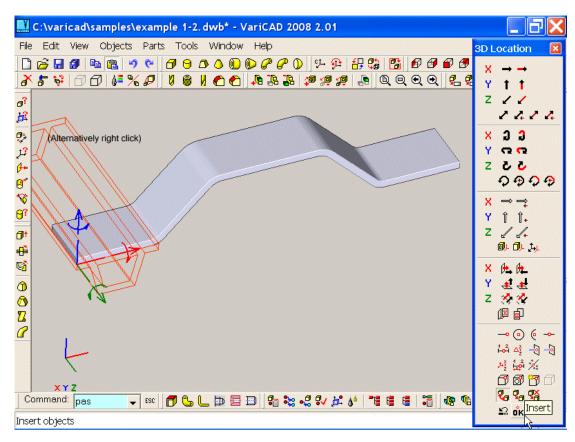
Changing Solid Insertion Point

Changing the solid insertion point

The solid is inserted at the location of the previously inserted object. In this situation, it is convenient to change the location of the insertion point of the inserted object before the inserted object is transformed. Here, the location of the new insertion point is at the automatically detected end of the highlighted edge. All next transformations will be relative to this new insertion point (until the next insertion point change).
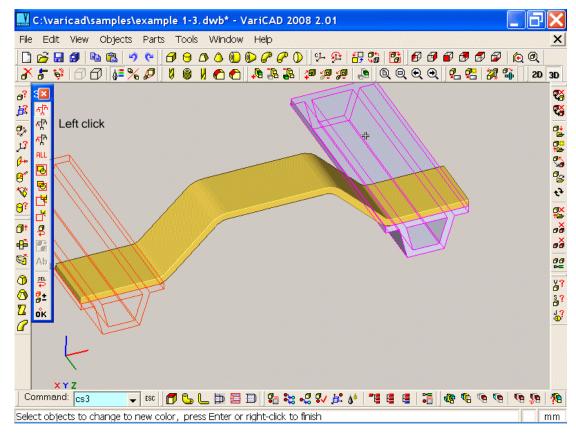
This insertion is performed within the function "paste". Insertion of a new solid or solids, insertion of objects from other file, function paste and transformation of the selected objects use similar methods.

Inserting and copying solid

The solid is inserted into a new location after the change of the insertion point. Here, the location was automatically detected at the endpoint of the edge. Because of the necessity to insert the solid again to another location, copying solids was selected – in a panel on the right side, click the copy icon before inserting objects. In this example, the identical copying was selected. All objects copied in this mode share future editing changes. The insertion is confirmed by pressing "enter" key or by the right click. The solid is inserted, a new one is created automatically at the same location and you can insert the new solid to another location. All procedures are repeated similarly for the insertion on the right side in the following steps.

Selecting Solids
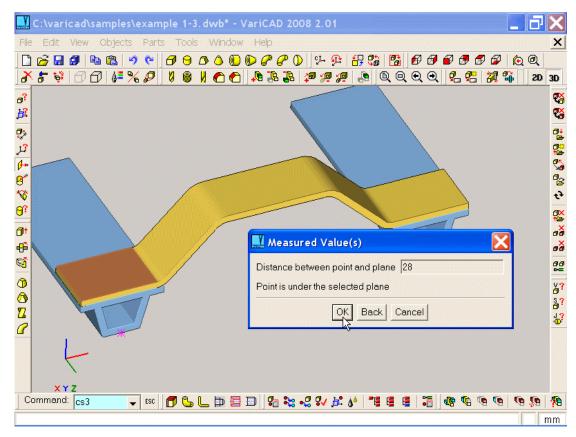
Selecting solids
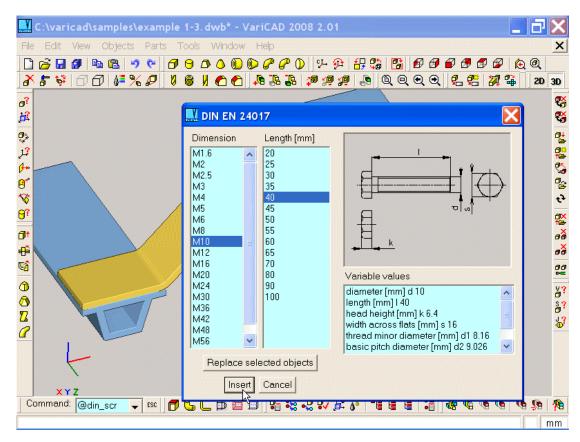
While working with multiple solids in 3D space, you may need to change colors of some objects. The entire scene is then more legible. In this example, the cross bar is changed to yellow color and the two beams to blue color. Here, the solid selection is a part of the function changing colors of the solids. The most simple selection is used – selection of the entire objects by their automatic detection. For more information about solid selection, see *Selecting Solids (page 97)*.

Working with Library Parts
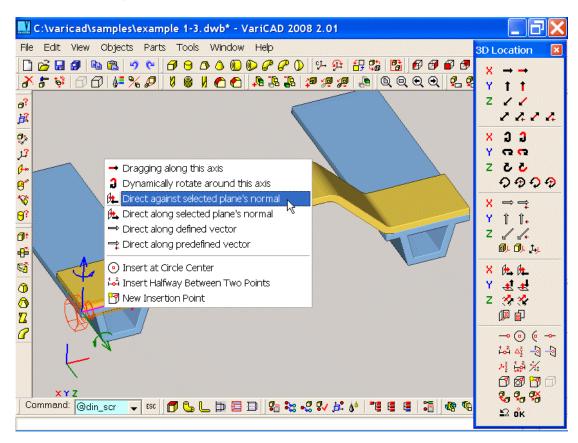
Checking the distance

The cross bar will be attached to the beam with a bolt connection. You need to know the distance between the two opposite planes before the correct screw dimensions are selected. Here, the distance between the upper and lower planes is important. The distance is measured from a point which is at the lower plane edge to the upper plane.

Selecting a screw from the list
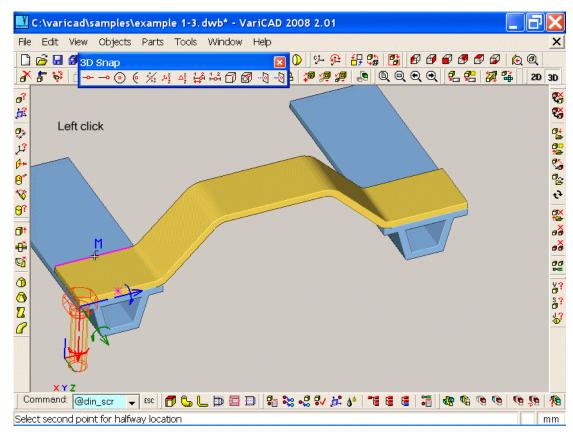
Transforming Solids

 Changing the direction of the axis

Some important methods of solid transformation are demonstrated in the example of screw insertion. For more information, see *Inserting and Transforming Solids (page 110)*.
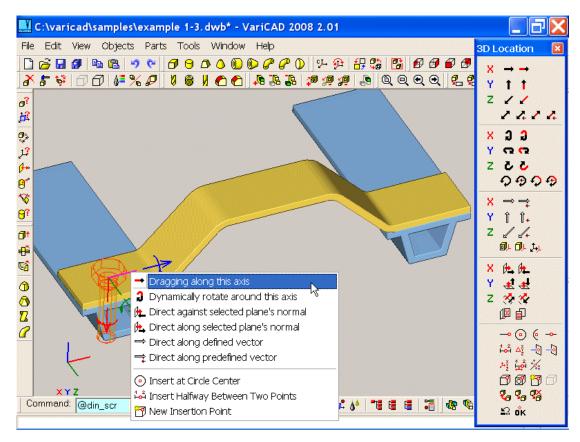
X axis of the screw (red color of axis) must be oriented against the normal of the upper plane. Left click the inner part of X axis of the inserted object, select the direction against plane's normal and then select the plane.

Inserting between two locations
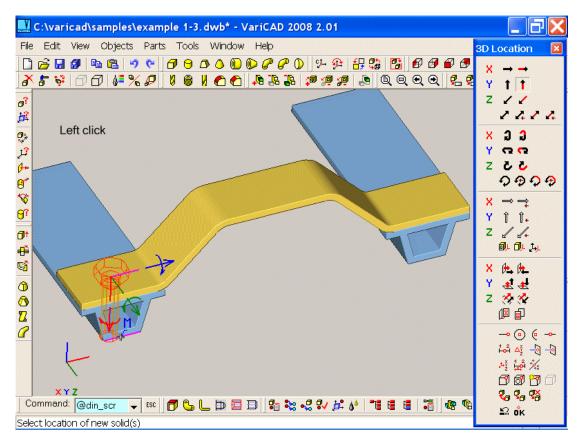
The screw is located at the half width of the yellow cross bar. Both locations on the sides of the cross bar can be detected automatically; the screw is inserted between them.

Inserting between two locations, inserting at the circle center (center of circular edge) or changing the insertion point of the inserted object are the often used functions. Because of this, you can reach them by left clicking the inner part of any axis of the inserted object. These functions are not specific for the particular axis.

Dragging along an axis
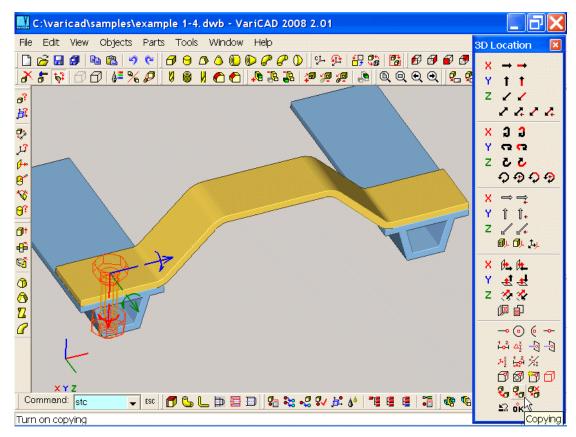
To define the location in Y axis direction (blue-color axis), you can drag the inserted screw along this axis. While dragging, all selected locations are projected perpendicularly to a point on the dragging line (axis) and the object is located to this point.

Dragging along an axis, location definition
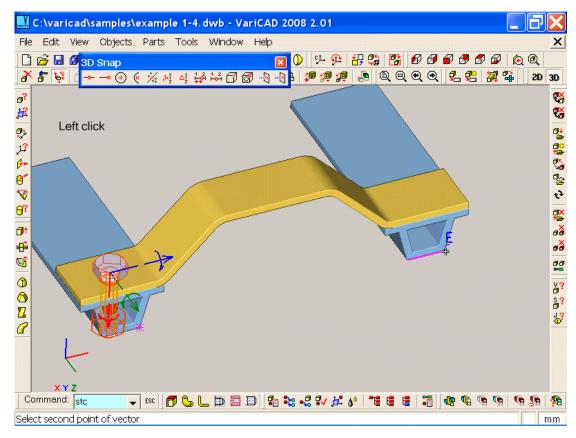
The screw will be inserted exactly at the half of the blue beam width. To achieve this, select the midpoint of the highlighted edge. The blue beam is symmetric; the midpoint of this edge is at the beam half width. Confirm the insertion of the screw. The nut is inserted similarly in the following steps.

Copying Solids

Copying solids
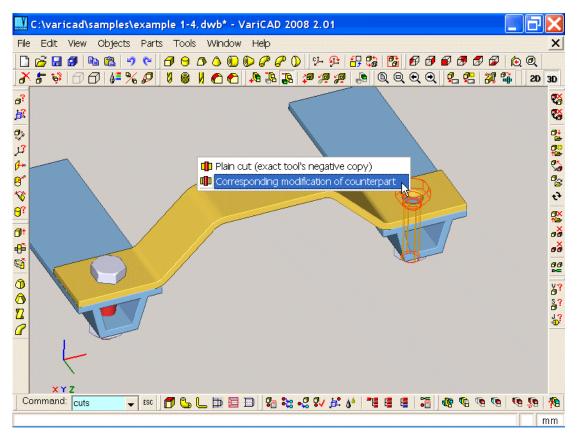
The screw and nut were selected. To allow copying, click the icon copy, then the icon OK (insert) or right click. Objects are copied and you can define their new location. Unlike in the previous example, there was used the plain copy mode. Objects do not share any future editing changes.

Translation of solids

The translation of copied solids is defined by two points in this case. In the next step, the translation distance is defined. The distance between the two points is offered by default. Click OK to accept the default value and the objects are translated. Then confirm the insertion.
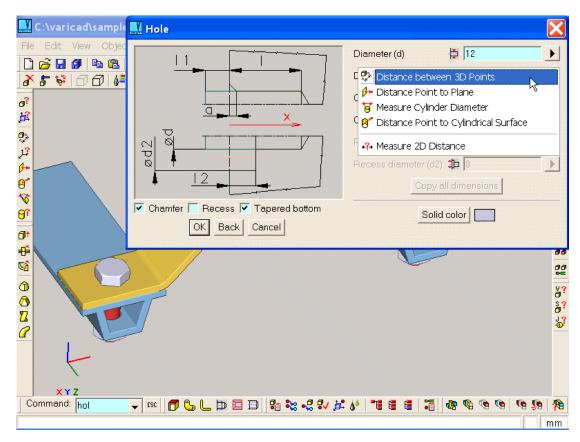
Boolean Operations - Cut

Cutting solids – creation of a hole for the screw
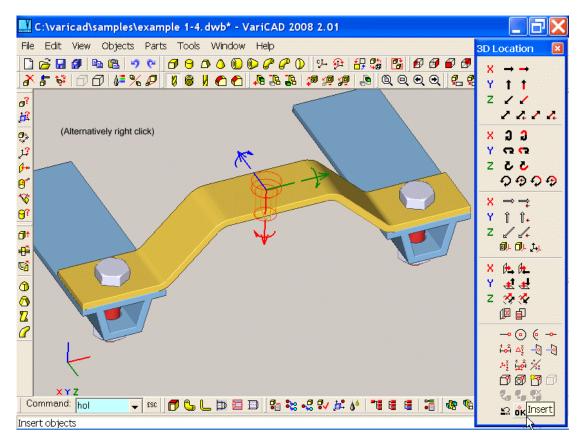
To create a hole for the screw, the screw was selected as a cutting tool. The hole was cut into the upper part (yellow cross bar). In this step, the screw is selected as a cutting tool again and the solid to be cut is the blue beam. If the cutting tool is a library part (like this screw) with the defined ability to modify the counterpart, a pop-up menu is displayed and you can select the option. In this example, the hole is created according to the respective standards – with the exactly defined diameter for the screw diameter. For Boolean cut, the function "Cut, Keep Cutting Solid" was selected. Because the two blue beams are identical copies, the hole is created into both of them at once.

VariCAD allows you to modify solids in any files regardless of how many other solids are in the particular file. There are no exactly defined assembly files or part (detail) files. One of the major advantages is the fact that you can modify any part according to other part, like in this example. On the other side, you may define assembly links to make editing changes at once for more identical objects. See *3D Assemblies (page 143)*.

Drilling hole – shape definition
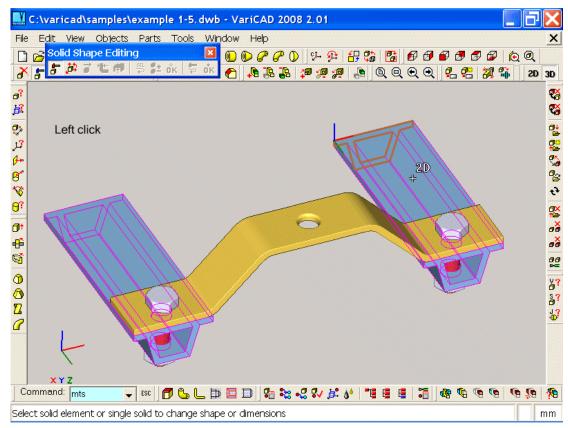
Drilling a hole, cutting material off with a box tool and creating a groove are so-called predefined Boolean operations. This example demonstrates the creation of a hole. To define depth of the hole (dimension l), measuring distance in 3D is performed. The thickness of the cross bar is measured in the following steps and then the certain excess is added to the result. The extension of the measured thickness allows drilling through the entire material.

Drilling hole – tool insertion

The tool creating a hole was transformed similarly as other inserted objects were transformed in the previous examples. The hole is created after confirmation of the tool position. For more information about Boolean operations, see *Boolean Operations (page 100).*
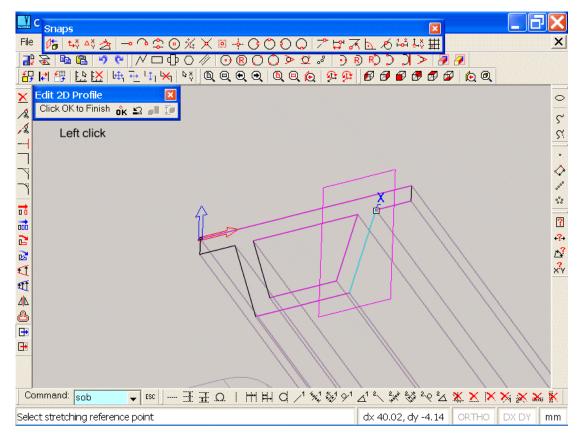
Editing Solids Created from 2D Profiles

Selecting a solid to be edited
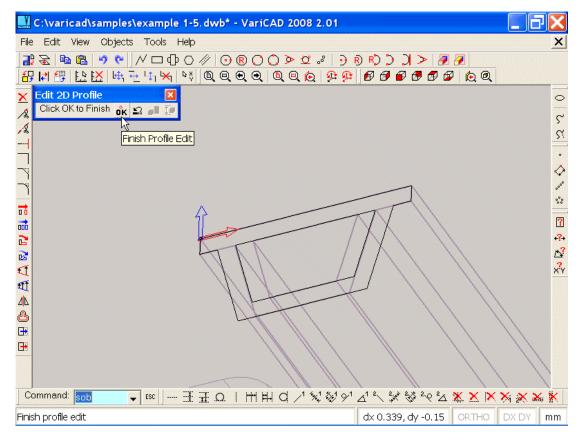
The function "Edit Solid or Blending" is selected to change the shape of the profile which creates the blue beam. If the cursor hovers above any object created by the profile extrusion, rotation or lofting, the profile is highlighted. 2D mode in 3D space is activated after the selection, and you can change the shape of the profile. See *Editing Basic Solids (page 105)*. Editing solids created as basic shapes (box, cylinder, hole etc) is demonstrated in *Editing Basic Solids (page 228)*. Here, the second beam is highlighted too, because it was created by identical copying. Editing changes will be applied to both profiles equally.

Solid's profile is always edited by 2D tools in 3D space. For more information about 2D drawing in 3D space, see *2D Drawing in 3D (page 90)* or *2D Drawing in 3D, Tutorial (page 421)*.
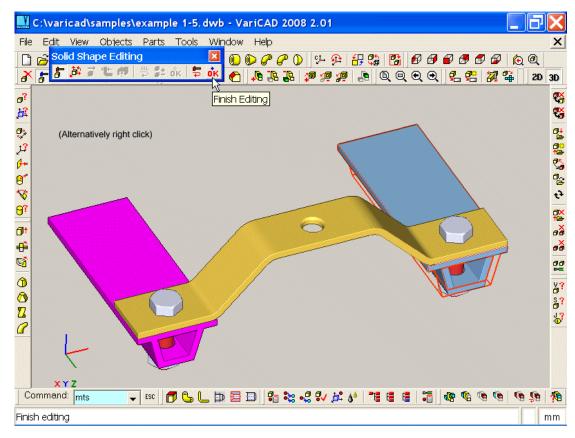
Stretching a part of the profile

To make the lower profile part wider, the function "Stretch" was selected. The parts of the objects to stretch are inside the defined rectangular area. The stretching reference point is selected. In the next steps, a new position of the reference point is defined by entering dx and dy relative to the original position. Line segment endpoints lying inside the rectangular area are moved according to the defined x and y distance. Similar steps are used for moving the left part of the profile to the left.
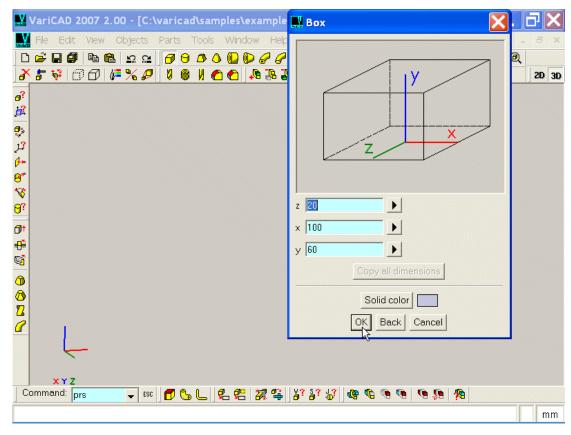
Finish profile editing

After finishing profile editing, the height of the extrusion is defined (remains the same for this example) and objects creating the profile are confirmed, added or removed.
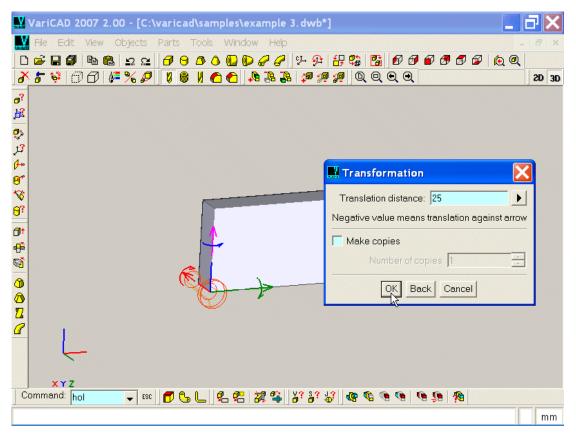
Confirm the editing change for both original and identical copies

Basic Solids
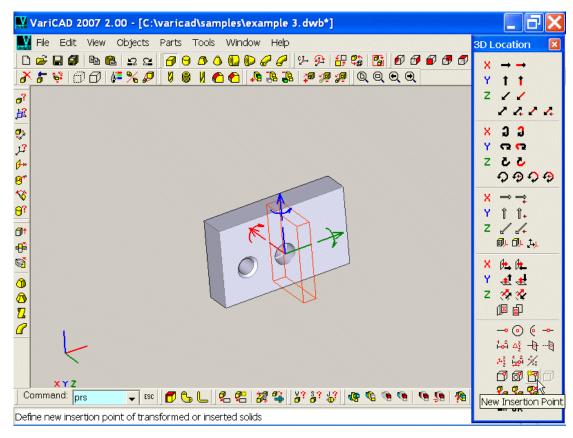
Creating a box

In this example, the box dimensions are defined, the box is inserted similarly as in the previous examples, and then the hole drilling is selected. For more information about basic solids, see *Basic Solids (page 96)*.
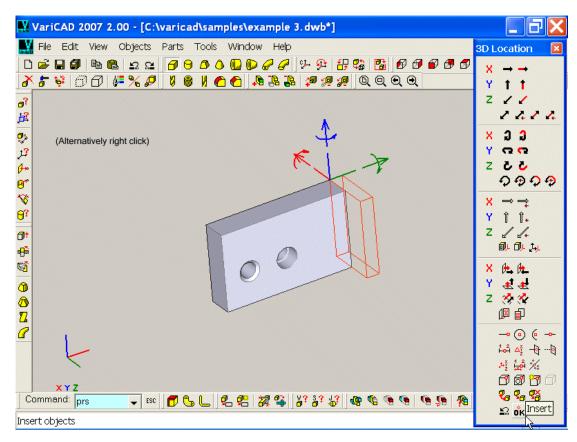
Translation of the tool drilling the hole

The tool drilling the hole was directed against the normal of the selected plane, moved to the lower left corner (automatically detected the end of the highlighted edge) and then the translation in Y direction was selected. To perform such translation, click the end arrow of the respective axis. Here, the translation distance is defined. In the next steps, translation in Z direction (green axis) is performed the same way and the position is confirmed. The second hole is created and inserted similarly. All used methods of transformation were already demonstrated in the previous examples.

Insertion of the second box
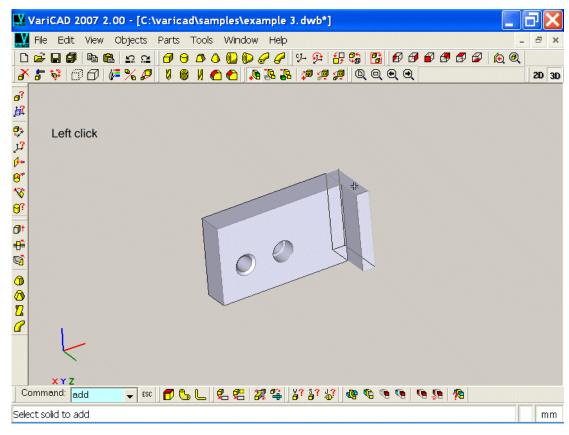
The second box was defined and initially inserted at the location of the previous object (the second hole in this example). The insertion point of the new box is redefined.

Confirmation of the final box position

The box was moved to the upper right corner, the location was automatically detected at the edge's endpoint.

Boolean Operations - Add
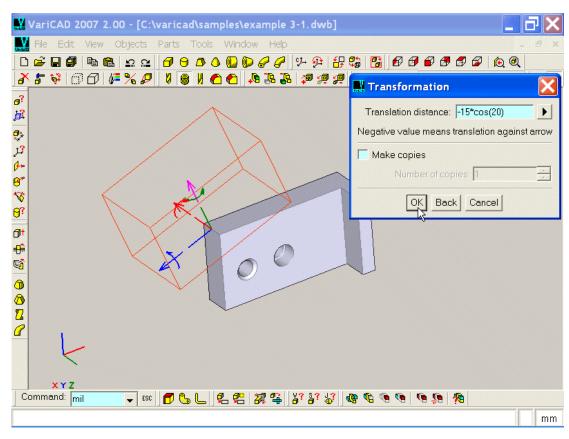
Boolean operation – add

To join the second box to the previously created solid to create one single object, the function "Add" is used. Select a solid to add and solid the previously selected solid will be added to. Both objects are joined into one new shape.
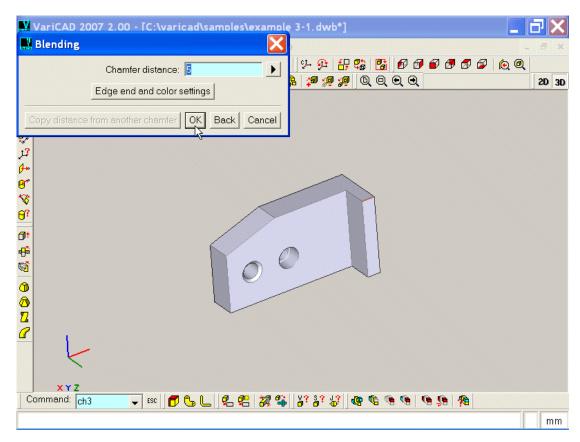
For more information about Boolean operations, see *Boolean Operations (page 100).*

Cutting Material with Box Tool
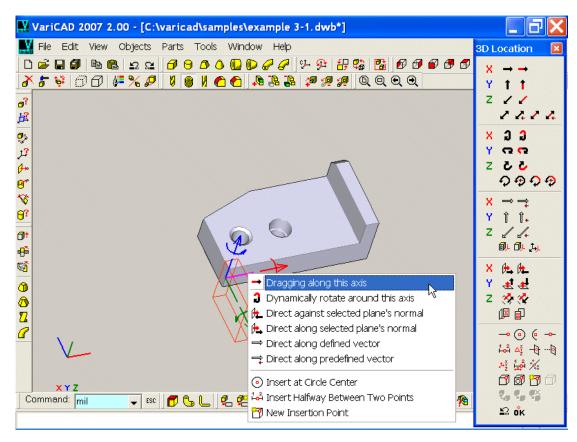
Transforming the position of the cutting tool

Function "Mill" allows you to cut material with a box-shaped tool. In this example, the tool's dimensions were defined and the tool was moved to the edge to be cut. Then the tool was rotated around X axis. The angle between the tool Y axis (blue) and the upper plane of the solid is 20 degrees. In this example, the translation against Z axis arrow will create the cut at the distance of 15 mm from the original edge, measured on the left side of solid. As you can see, mathematic expressions can be entered instead of single values. For more information about entering mathematic expressions, see *Mathematic Expressions (page 35)*
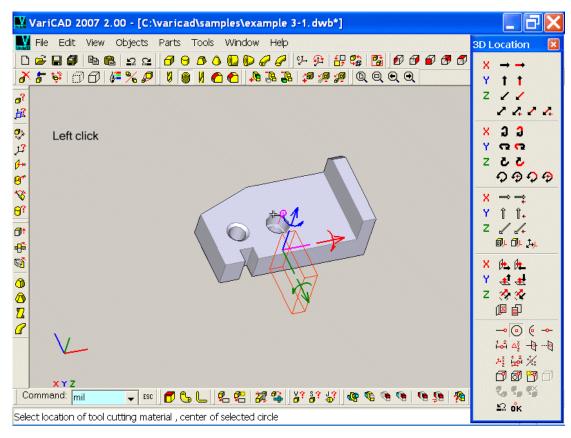
Chamfering 3D edge

Chamfering edges allows you to cut an edge under 45 degrees. Selecting the edges is similar to selecting edges for 3D filleting.

Transforming the position of the cutting tool
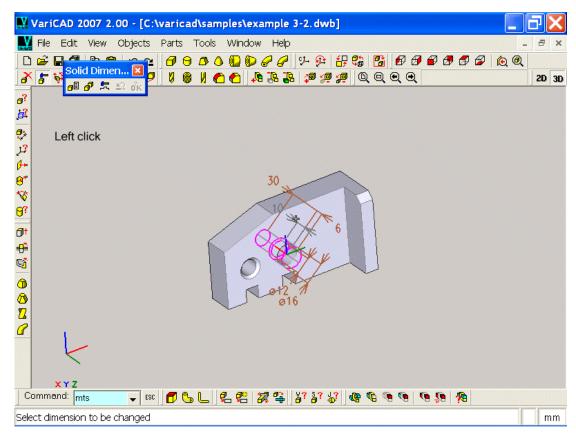
Another box-shaped cutting tool was inserted, directed according to planes and moved to the midpoint of the edge. Drag along X axis to move the tool exactly under the axis of the hole. Dragging was described in the previous example *Dragging along axis (page 206)*.

Dragging along the axis to a position
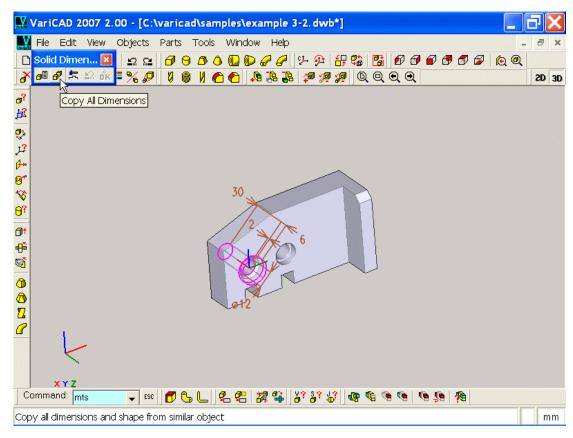
For exact definition of the tool position under the axis of the hole, the center of the circular edge of the hole was selected as the location to drag to. Then the position was confirmed, material was cut off and the second position was selected similarly. After selection and confirmation, material will be cut off at the second location, too.

Editing Shape of Basic Solids

Editing a hole
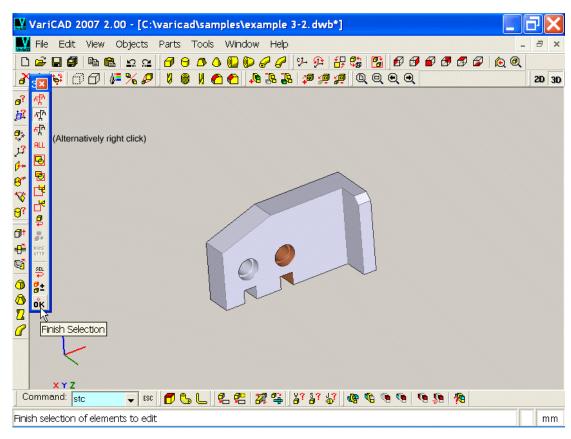
If the solids are created as basic shapes (box, cylinder, hole etc), you can edit them very easily with spatial dimensions, as shown in this example. Editing shapes created from the profile is demonstrated in *Editing Solids Created from Profile (page 215)*

 Copying the shape
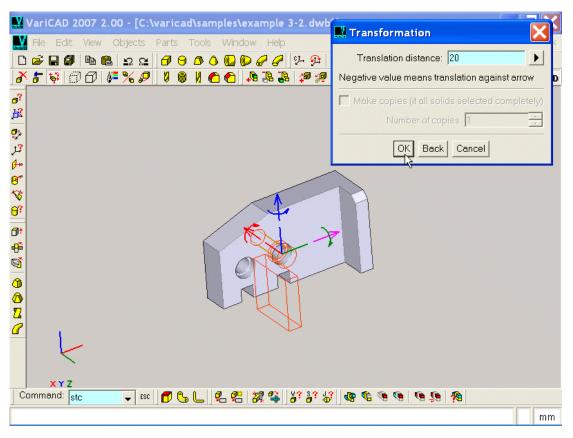
Select copying the shape to copy the exact shape of the right hole to the left hole.

Transforming Parts of Solids

Selecting a hole and groove for transformation

If you need to change the position of any basic parts creating 3D object, switch the object selection to select the single elements or to select the branches of Boolean tree. See *Selecting Solids (page 97)*. In this example, the hole and groove are selected as single objects.
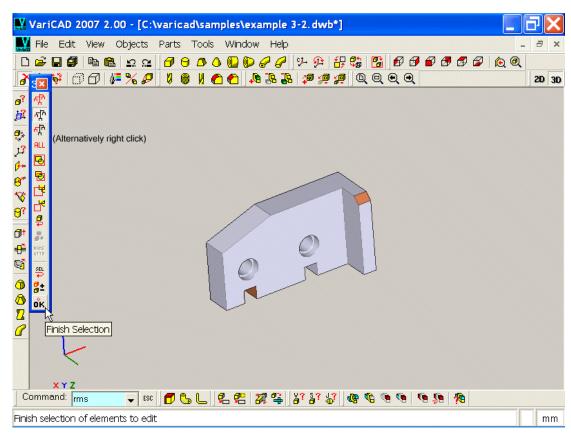
Transformation of the objects

After selection of a solid's part for transformation, the original shapes (in this example the tool creating the hole and the tool creating the groove) are displayed and you can define a new location. After confirmation, the objects are moved to the new location.

For transforming parts of solid, you can use also a respective option of the function "Edit solids". The edit function allows you to transform only sub objects from one solid at once and allows you to change shapes, too. The function "Solid transformation" used here allows you to transform entire parts as well as their sub parts at once, but does not allow to change shapes.
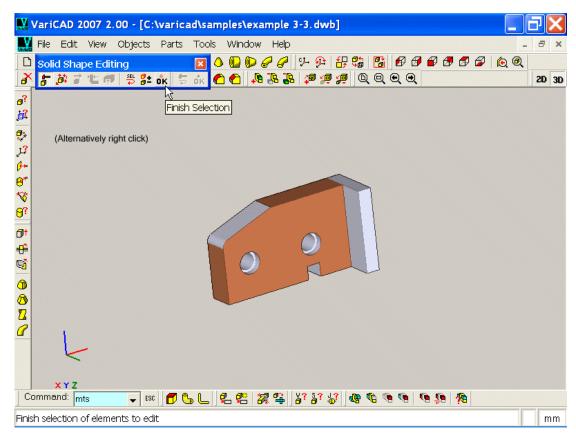
Removing Parts of Solids
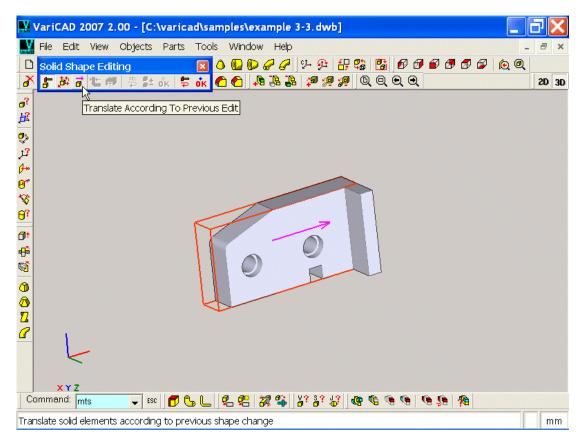
Selecting parts of the solid to be deleted

If you need to delete any part or parts of the solid, switch the object selection to select single elements or to select branches of Boolean tree. See *Selecting Solids (page 97)* - selection is the same as in the previous example demonstrating transformation of the parts of the solid. The groove and the chamfer of the edge are selected to be deleted in this example. The chamfered edge will be rounded in the next steps. Selecting edges for filleting and fillet radius definition is similar to the methods of chamfering, demonstrated previously in *Chamfering (page 224)*

Transforming parts of solids after changes of their dimensions
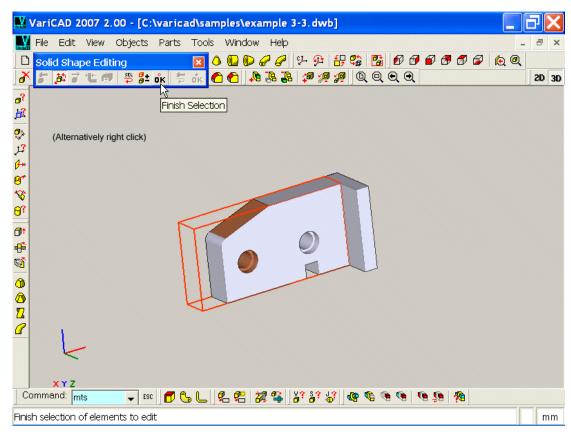
Selecting part of the solid to be edited
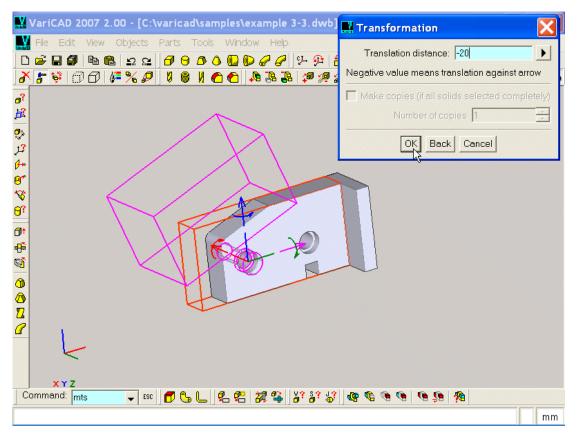
Shape after the dimension change

You can conveniently translate any parts of solids together with a change of the shapes. In this example, the box, which creates base, is wider after the editing. A new shape spreads to both sides in X-direction. Select the translation in the predefined direction and the box translates to the left – the same position of edges at right side is preserved. For more information about editing solids, see *Editing Solids (page 105)*
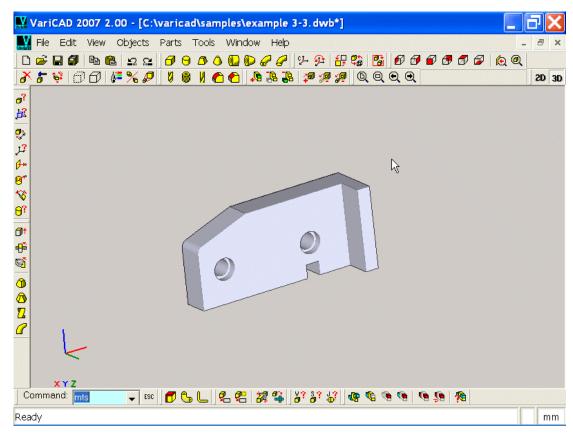
Selecting parts of the solid to be translated

Defining translation
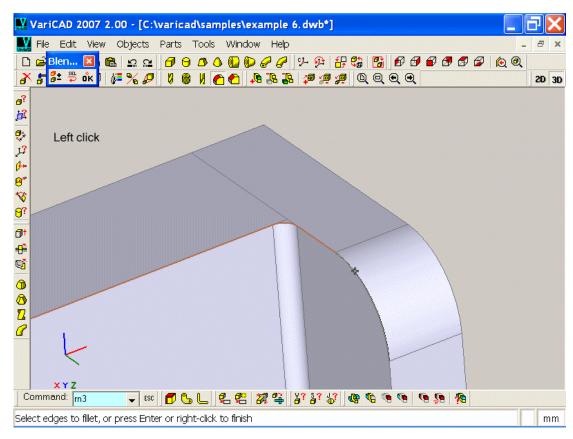
The box is edited and translated. Then, the left hole and cutting tool creating the chamfer are selected. In the next step, the selected objects are translated along X-axis to the left at a distance of 20 mm (box is 20 mm wider). The left hole and groove are again at the same distance from the left side after the editing change.

You cannot select filleting at the left side for translation. All blendings are automatically rebuilt after shape or position changes.
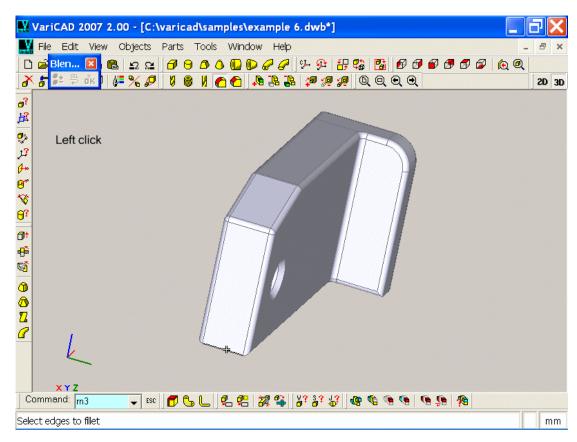
Solid after editing
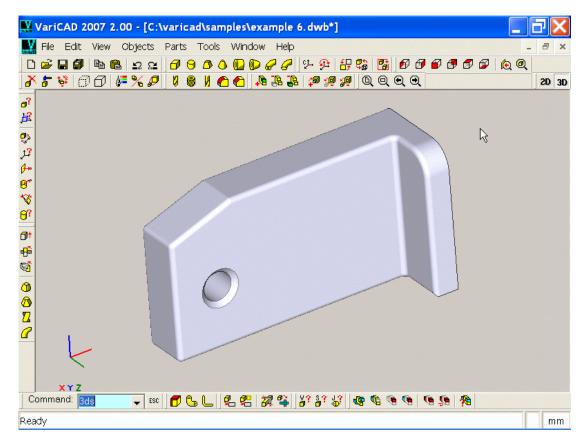
Filleting Edges

Filleting, selecting edges if two edges in a vertex are sharp and one is obtuse

Filleting, selecting edges to create a rounded corner

Filleting, final state

If you need to blend an edge created from more segments continuously as one object, select all these edges together. Especially filleting will be done more efficiently. Moreover, in some cases the edges selected separately cannot be rounded. On the other side, you cannot select three edges in one vertex together.

If you need to create a rounded corner, select two edges in this corner (preferably at once), perform filleting and then select the remaining edge and perform filleting on it. If you need the edges rounded with different radii, then select and round each edge separately.
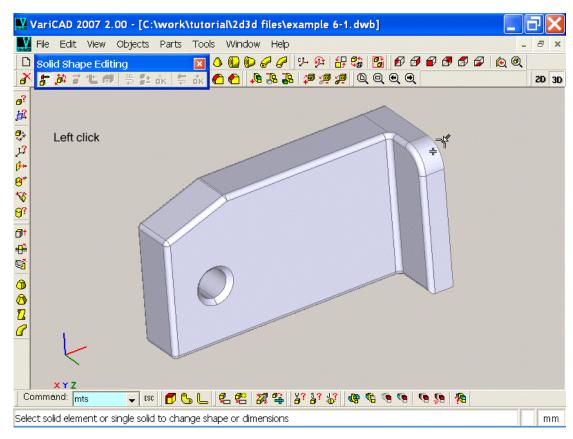
If one edge in a corner has a radius significantly greater then other two edges, you must round this edge first and then select and round remaining two edges together with a new edge at the end of the previously created fillet (see example in the image above).

Similarly, if you need to round edges in the vertex containing two sharp and one obtuse edge, round the obtuse edge first and then select and round remaining edges with the edge at the end of the previously created fillet (see again the example in the image above).
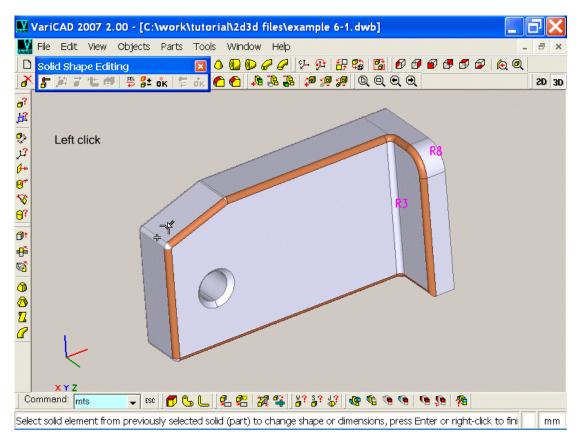
It is better to have displaying of all edges (including the tangent ones) turned on. First impression could not be the best, but you can always use different settings for precise display. In the work mode and especially while filleting edges, it is always clear where the edges have their ends and if some edge is created from more segments.

For more information about blending, see *Blending (page 104)*

Editing Fillets
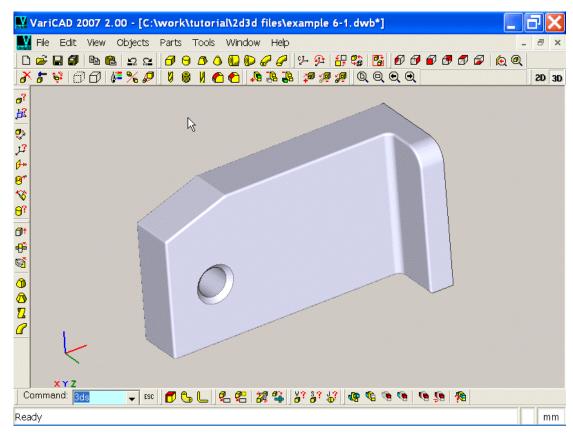
Selecting fillet to be edited

 Selecting another fillets within one function

You can select and define change for more separate fillets at once. After all changes are defined, the solid is rebuilt.
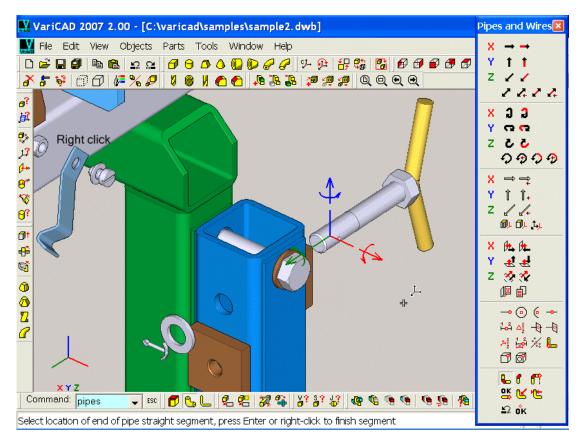
For more information about editing solids, see *Editing Solids (page 105)*

Solid after editing
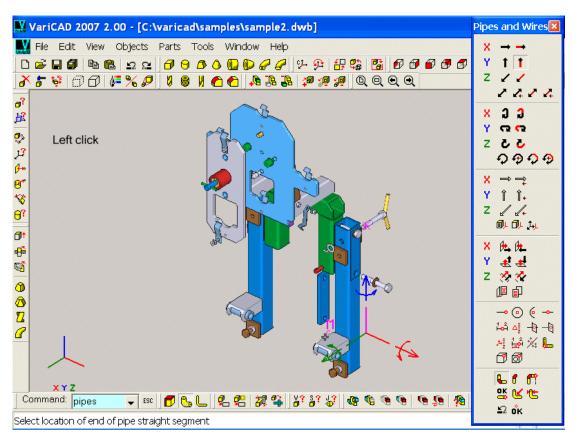
Creating Pipes

Pipe according to Other Solids in Assembly
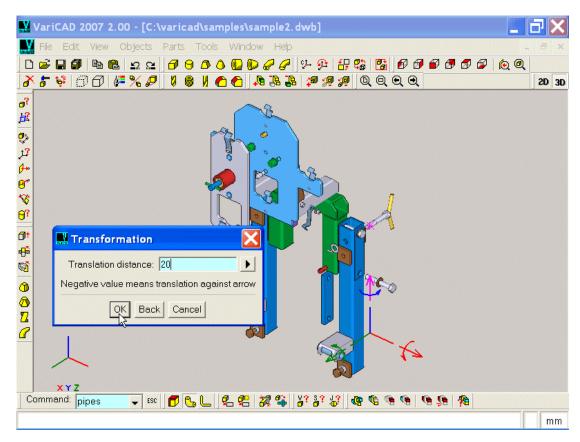
 Definition of a pipe start point

Define each location of pipe segment's endpoints similarly as in the function *3D Transformation (page 110)*. You have more methods available for definition of a pipe path, see *Pipes (page 136)*. In this example, the pipe start point is adjusted according to the edge of the blue beam, than the point is dragged along Y (blue) axis down to the selected location and finally lifted up to the desired distance from the foot. Here the end point of the straight segment is defined.
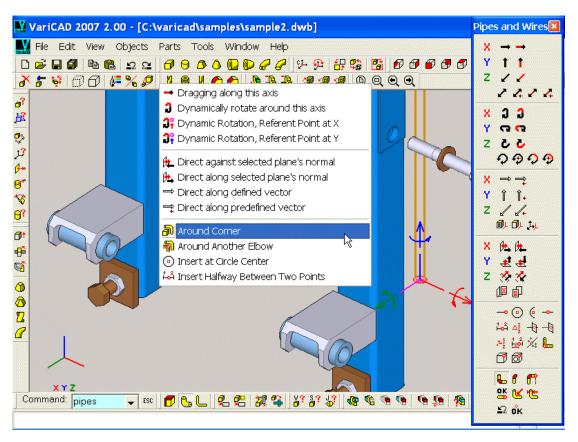
Dragging a pipe segment end point
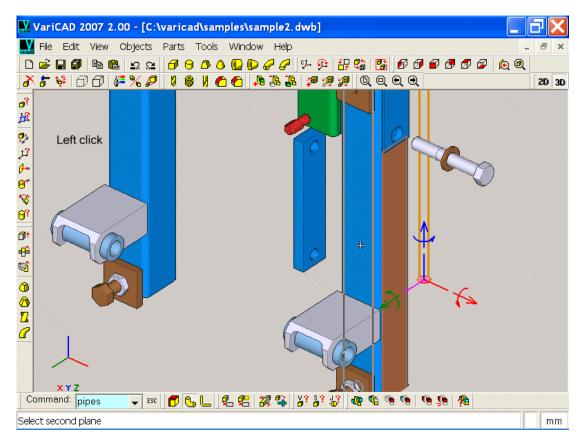
Lifting a pipe segment end point to a desired distance
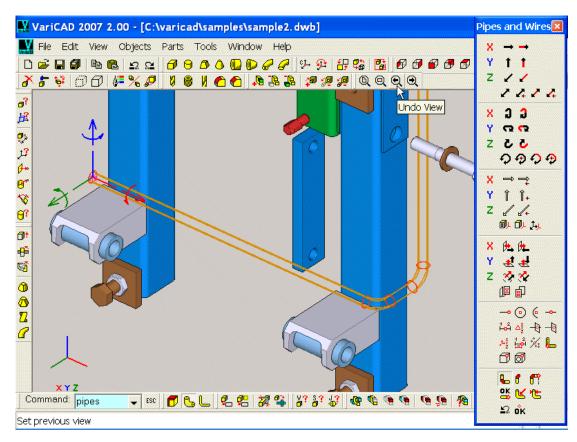
You can easily bend pipes around corners, as shown below.

Defining pipe direction, elbow will be created around corner

Selecting planes of the corner the elbow is created around
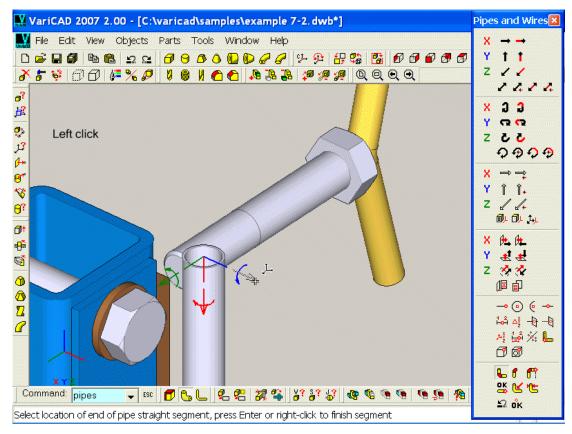
Pipe bent around a corner
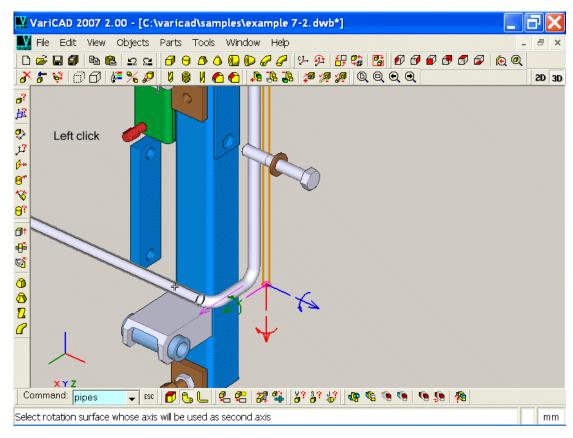
If you create straight segments, an elbow with a defined radius rounds each vertex of the pipe path. Press Enter, click the corresponding icon or right-click to complete the pipe creation.

Pipe according to Other Existing Pipe
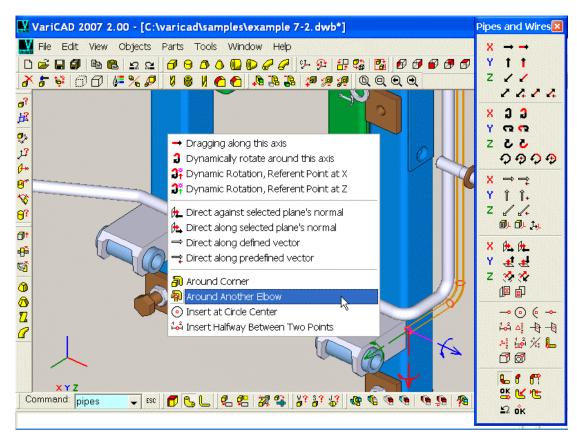
In this example, the pipe start point is adjusted according to other pipe endpoint. Then the locations of endpoints of segments are defined according to the existing pipe. Select the direction pointing to the next location. If necessary, rotate the axes properly first. Select the first and second pipe segment connected by the elbow. The new elbow is created around the existing elbow. Thus, you can create a new pipe parallel to the existing one.
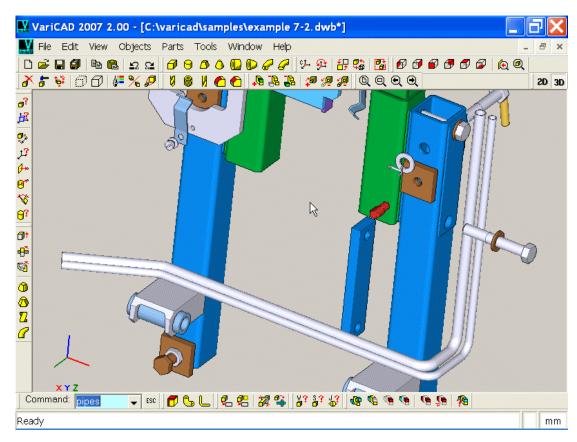
Definition of a pipe start point

Selecting a location at other pipe to create an elbow
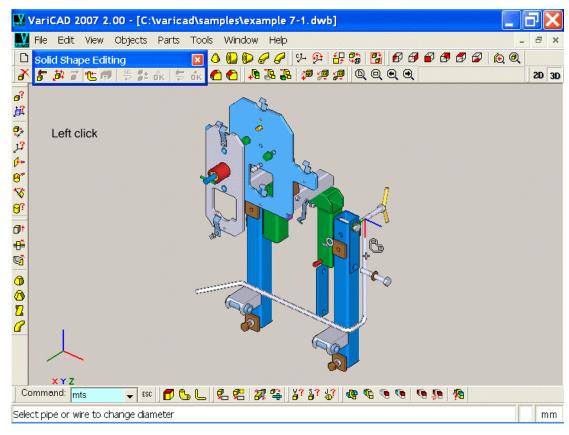
Defining pipe direction, elbow will be created around other elbow
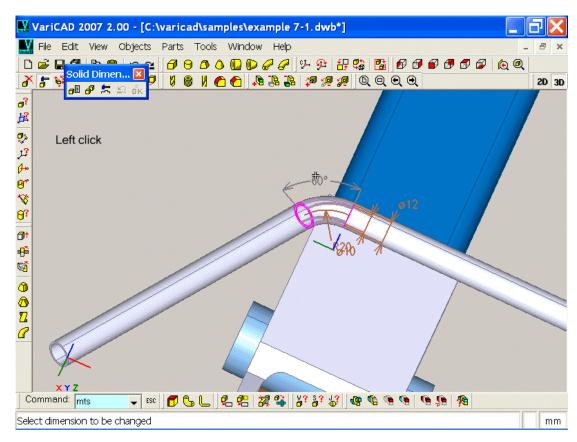
A pipe created parallel to the existing one

Editing a Pipe
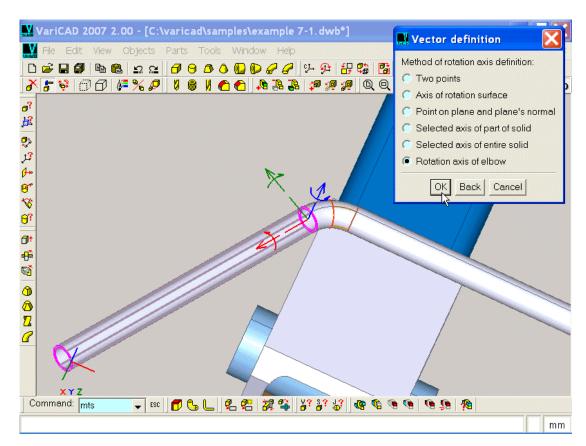
You can edit the shape of the pipe similarly as shapes of other solids – see *Editing Solids (page 105)*. First, the diameter of the entire pipe is changed. Next, the angle of the elbow is changed and the adjacent pipe segment is rotated to a new correct position. A point defining rotation is at the center of segment's end circular edge. The new location of this point is at the center of the end circular edge of the changed elbow.
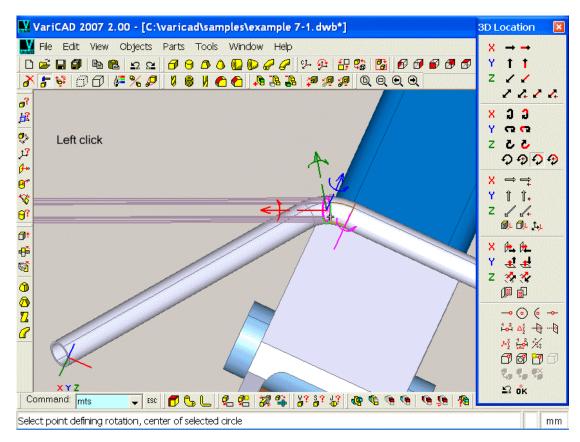
Selecting a pipe to change its diameters

Changing an angle of the elbow

Rotation of an adjacent segment to the correct position
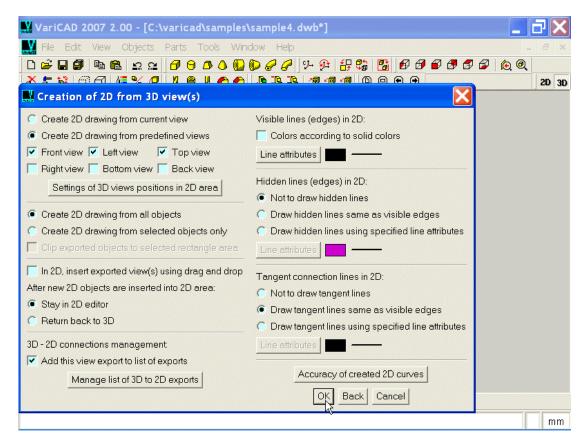
Sticking the point, which defines rotation (center of rotated segment) to new location
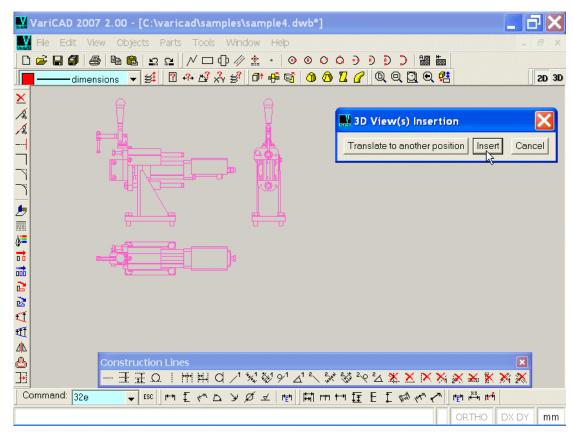
Creating 2D from 3D

Creating 2D, First 2D Definition
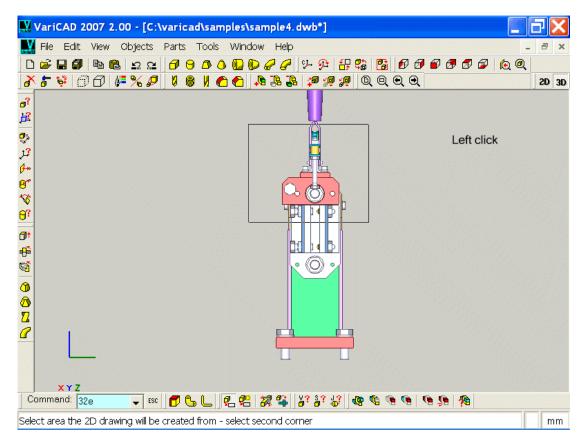
You can export 3D views to the 2D drawing area. In this example, the basic predefined views are exported (left view, front view, top view). Then the area clipped by a rectangle is exported. This export is enlarged as a detailed view in the 2D area. Finally, the general view is created. While exporting, the option "Add this view export to list of exports" is active. This enables you to change 2D drawing after changes in 3D. For more information, see *Exporting Views and Sections from 3D to 2D (page 131).*
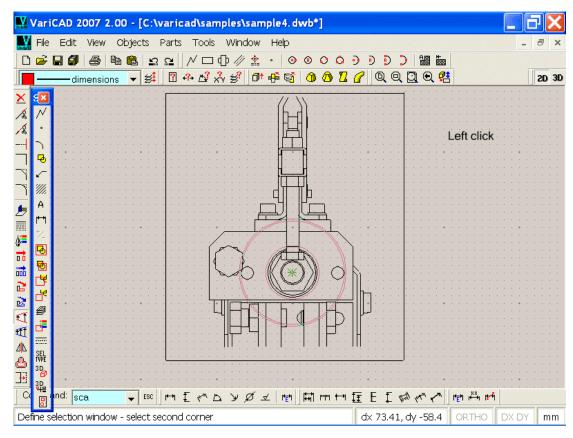
Selecting method of 2D creation

Inserting 3D views into 2D
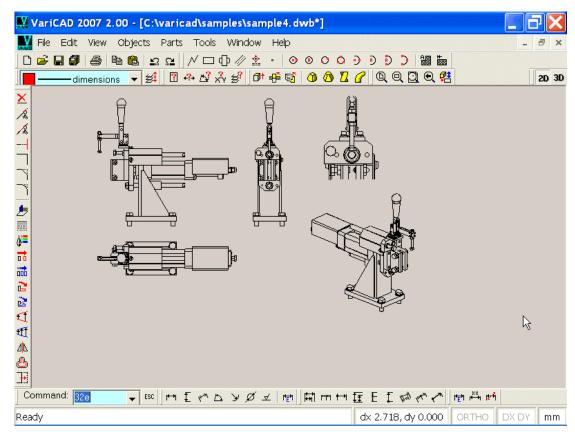
Selecting area to create detailed view in 2D
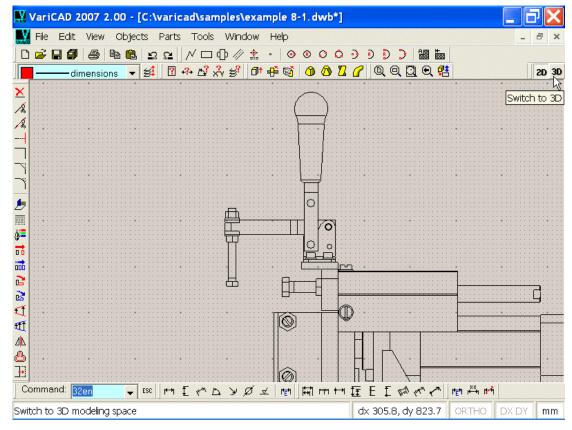
Scaling objects exported from selected area

After the 2D objects from the corresponding 3D view are created, you can translate, rotate or rescale them. Select all objects from the particular export. Thus the connections between 3D and 2D remain and 2D is properly updated after changes in 3D. Transformed 2D objects are updated at their new locations. As shown here, area clipped by rectangle is scaled by factor 2. All 2D objects are selected by 2D rectangular area before scaling.
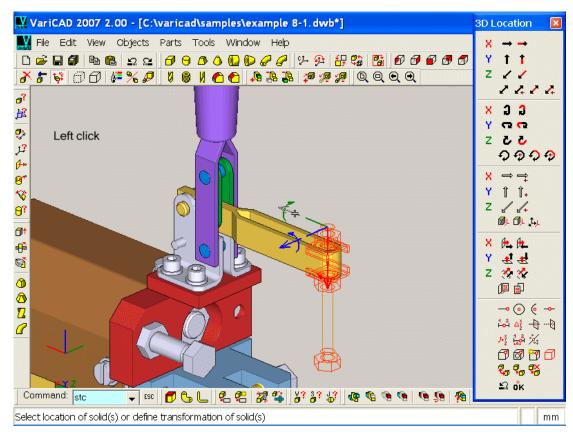
Completed 2D drawing
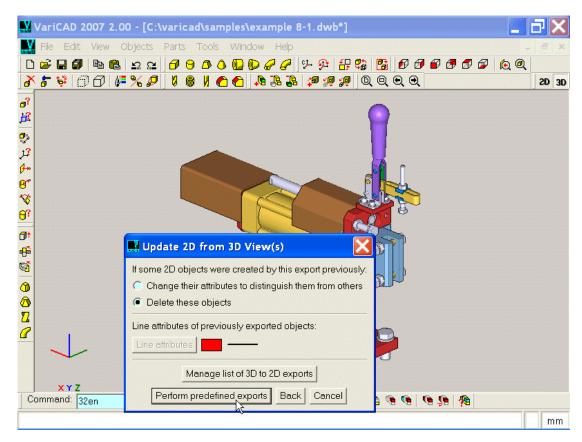
Updating 2D after 3D Changes

2D drawing before 3D change
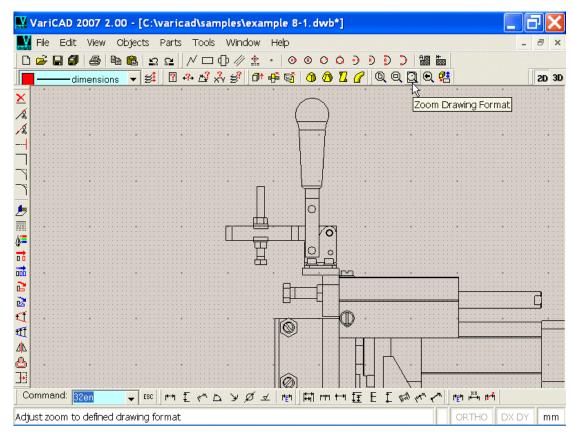
Performing one of editing steps in 3D

 Updating 2D according to predefined exports
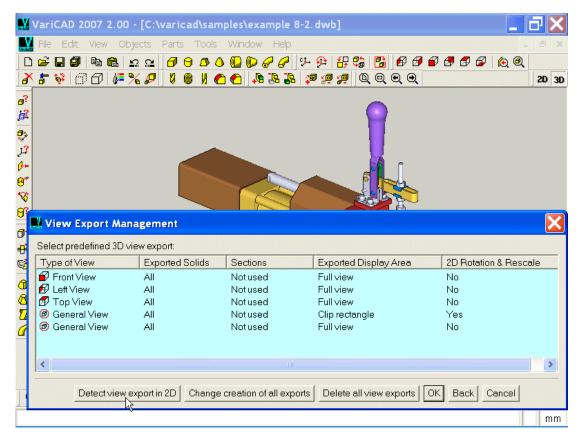
Performing 2D update after 3D changes, you can select whether the old 2D objects are deleted (as used here), or if they remain and their 2D line attributes are changed – this allows you to easily distinguish changes in 2D drawing.
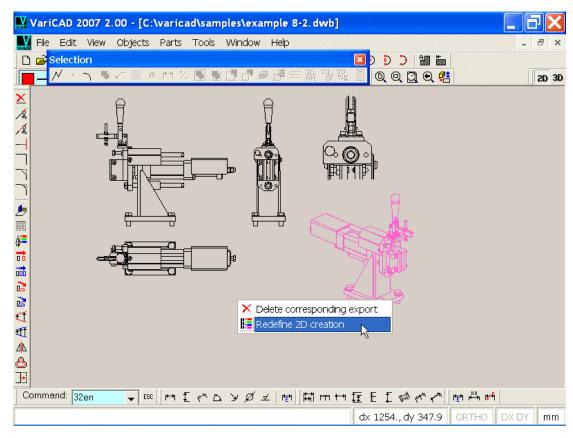
Detail of 2D updated after 3D changes

Changing Predefined Method of 2D Creation
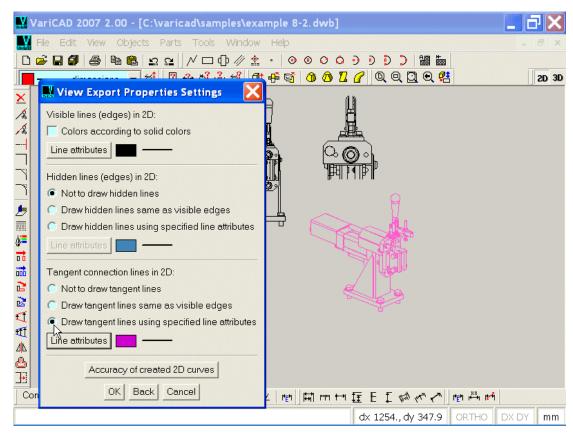
You can change or delete particular predefined exports of 3D views. This can be especially useful if you need to change the 2D attributes of created views (color or line type), or if you need to stop the creation of 3D view you do not need any longer. In this example, the change of the color of tangent connections in the selected general view is performed. Generally, the colors of the lines in 2D corresponds with line thickness for printing.
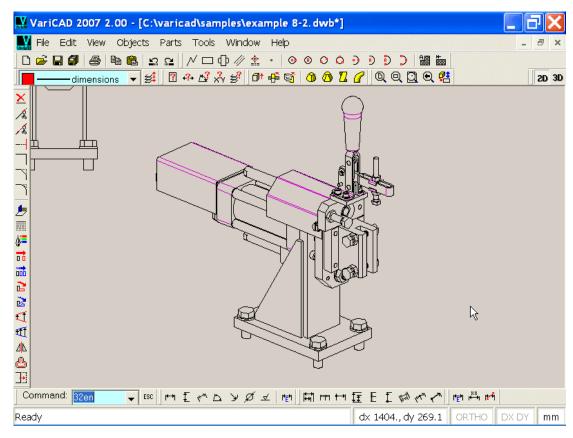
List of predefined view exports

Selecting a predefined export for change

Changing predefined export attributes
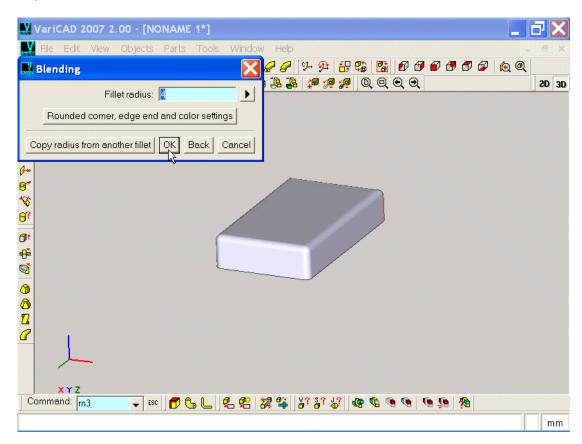
2D after changed attributes of export
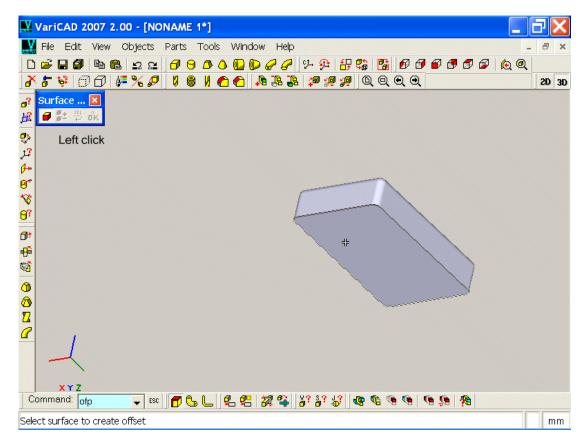
3D Shells

Creating 3D Shells
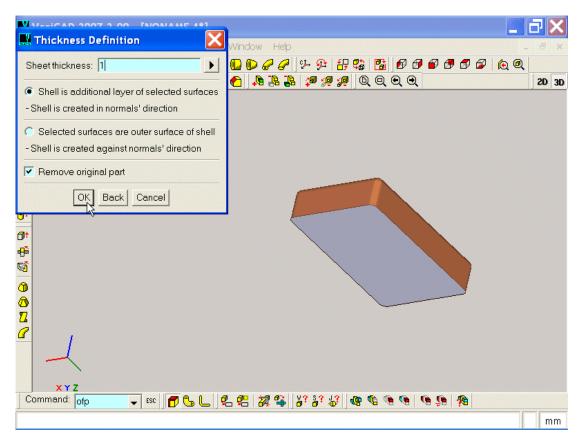
You can create a shell as connected offset patches and selected original patches. Sheet metal parts can be created initially as shells. See *Offset Patches – Thick Shells (page 138)*. In this example, the patches for offset are selected by the option "All except detected". Otherwise, you have to click each patch separately.
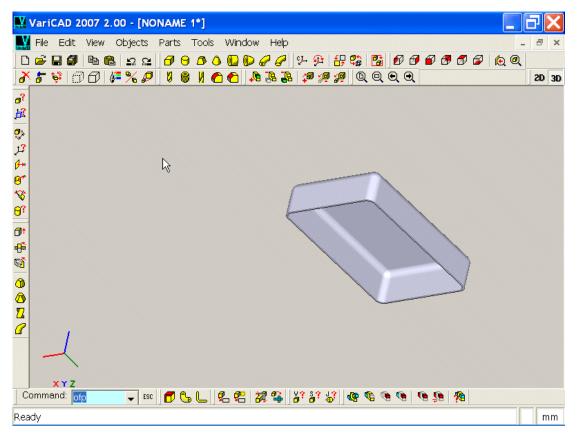
Creating a solid as a shell pattern

Selecting patches

Thickness Definition

Sheet thickness: 1

- ● Shell is additional layer of selected surfaces
- Shell is created in normals' direction

- ○ Selected surfaces are outer surface of shell
- Shell is created against normals' direction

- ☑ Remove original part

OK   Back   Cancel

Command: ofp
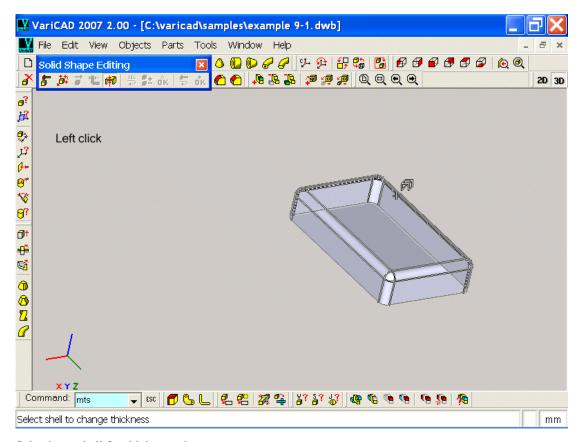
Shell thickness definition
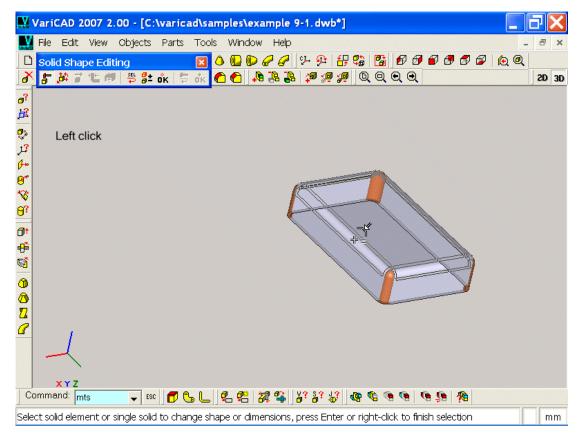
Result

Editing 3D Shells
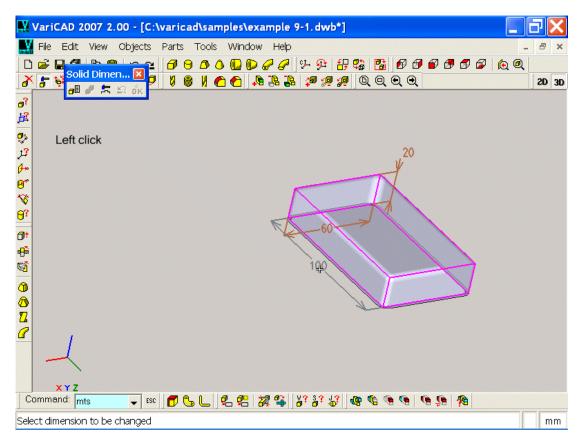
You can edit either the shell thickness or the shape of the shell pattern solid. In this example, thickness is edited separately.
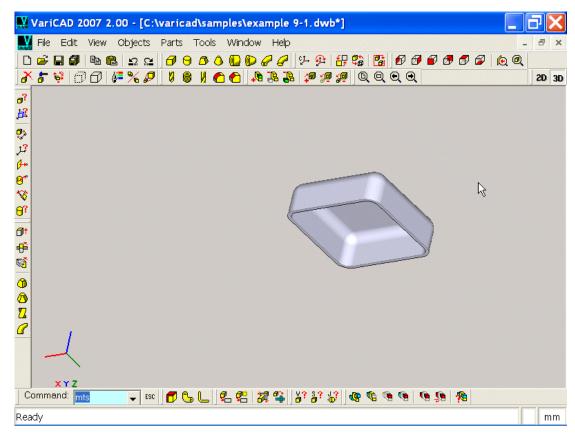
Selecting a shell for thickness change

Selecting fillets of the shell to be changed
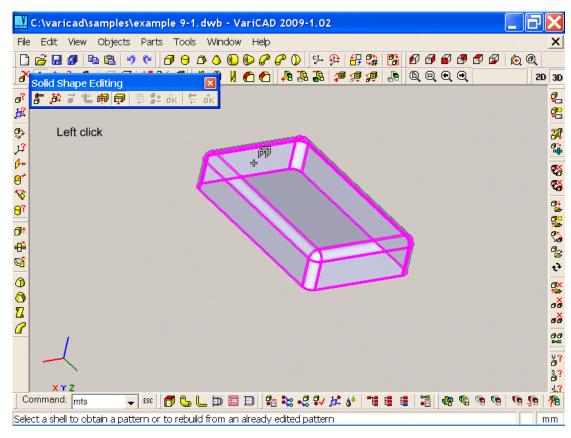
Changing dimensions of the box creating base of the shell pattern
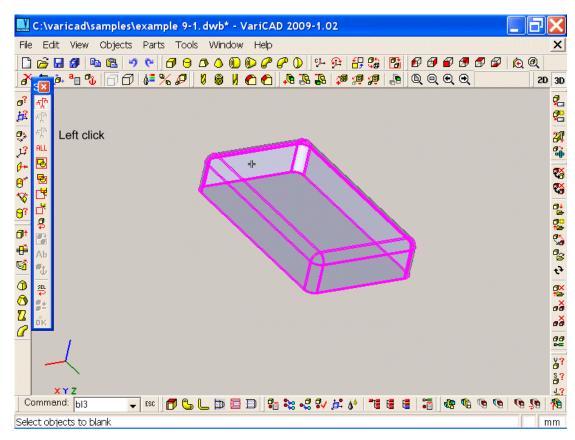
Changed shell
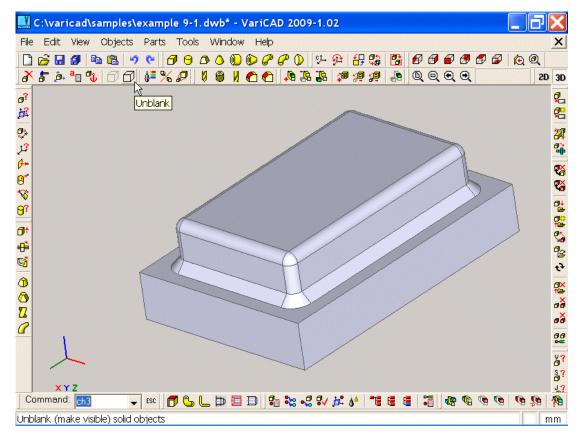
This example demonstrates how to change a shape of a pattern solid the shell was created from. If you need to change the pattern with Boolean operations, work with constraints or create the shell from different patches, you need to extract the pattern solid into 3D space. After all changes, select patches again and the shell is rebuilt.
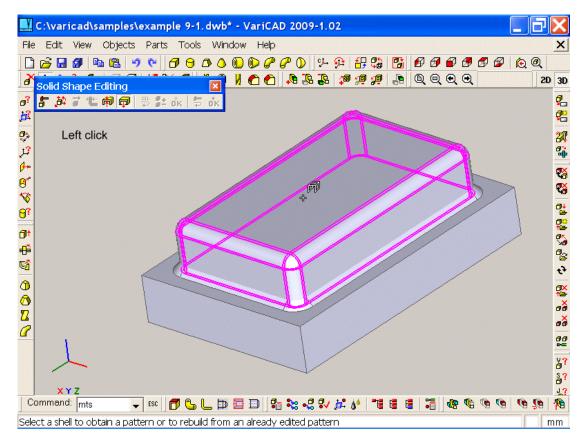
Selecting a shell
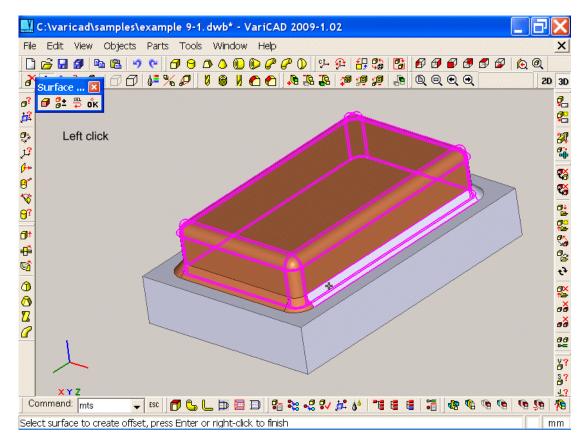
After extracting the pattern into 3D, blank the shell or move the pattern away. Otherwise the editing may be difficult – the shell would interfere if you select objects.
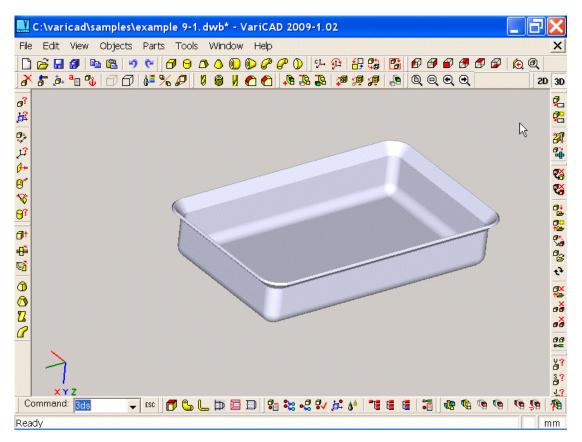
The pattern solid was edited. A box was added and connection between the original solid and the new box was chamfered. The shell will be unblanked.

The shell is selected again. If the corresponding pattern was extracted into 3D space previously, you have to reselect patches for the changed shell.

Selection of additional patches

Changed shell

Sheet Metal Unbending (Surface Development)

Detecting Sheet Metal Automatically

For unbending, you can detect sheet metal automatically. This works if detected objects have the thickness significantly smaller than the other dimensions. Surface development (sheet metal unbending) uses the bending coefficients. These coefficients correct the unbent dimensions according to the sheet thickness. For more information about surface development, see *Surface Development (page 145)*.

In this example, the entire solid is detected at once. Then confirm the thickness and locate the unbent shape in 2D area.
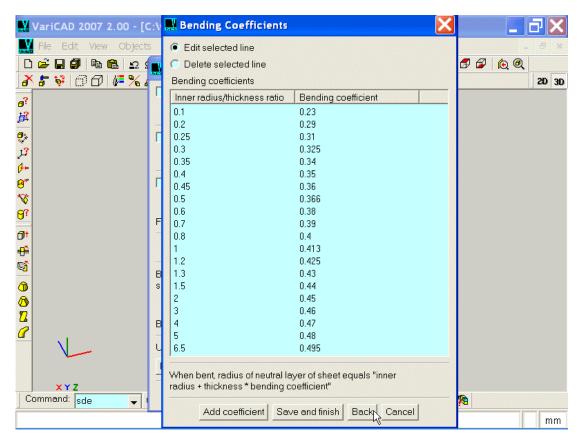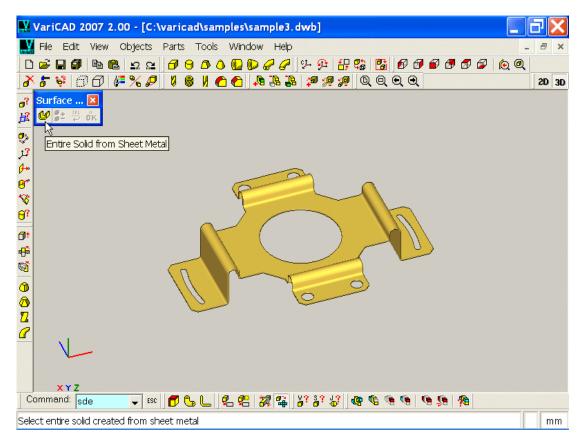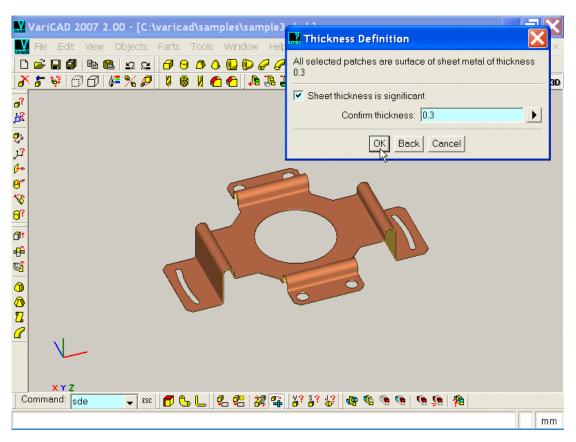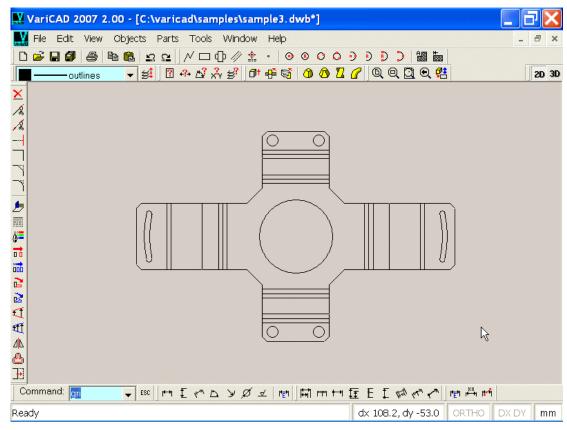
Table of bending coefficients

Selection of the option to detect the entire solid at once

Thickness confirmation
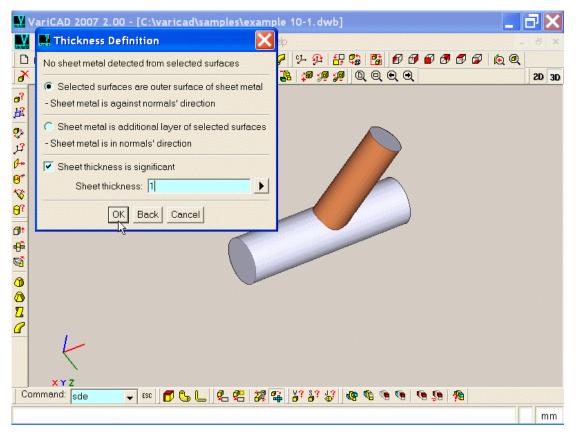
Outlines of the unbent solid in 2D area
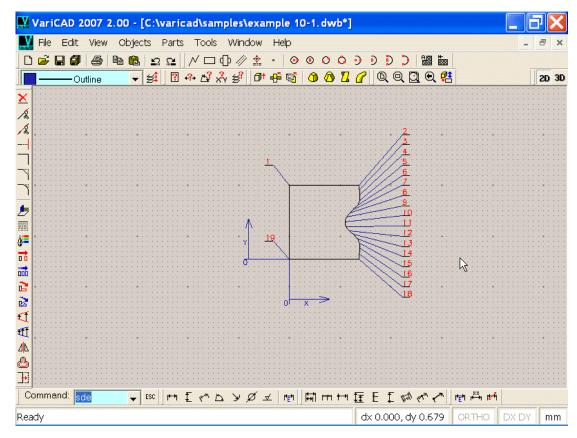
Selecting Surface for Unbending

You can select each patch of the solid for unbending. After the selection, define the sheet thickness. In the following example, the file containing XY coordinates of unbent surface is created. Coordinates are listed for outline points in 2D, where leaders mark their locations.
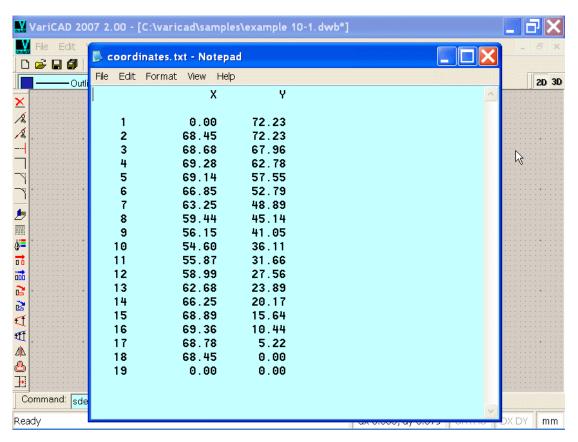
Here, the selected surface is the outer surface of the full cylinder. The result is the same as for unbending pipe.

Selecting a surface for unbending

Outline of the unbent surface in 2D
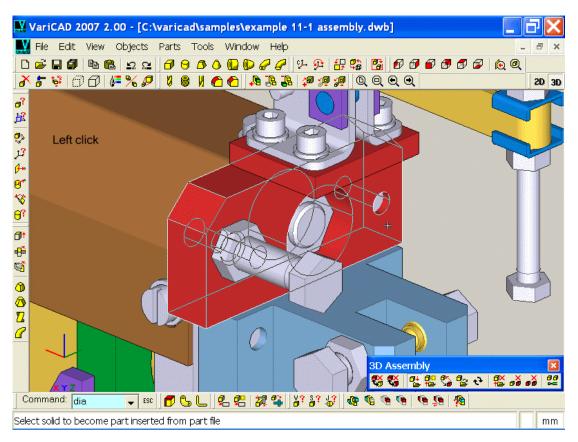
File containing XY coordinates

3D Assembly
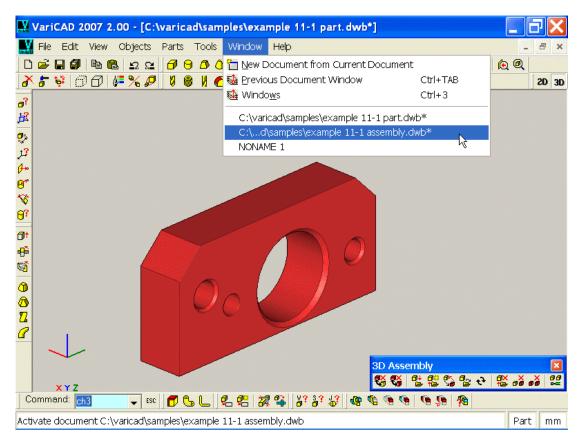
Creating Assembly Connections

You can create assembly links between assembly files and files containing parts. If an object connected with an assembly link is changed, the change is copied into all files where the object is present. On the other side, it is not necessary to define assembly links whenever you work with more solids in one space (in one file).
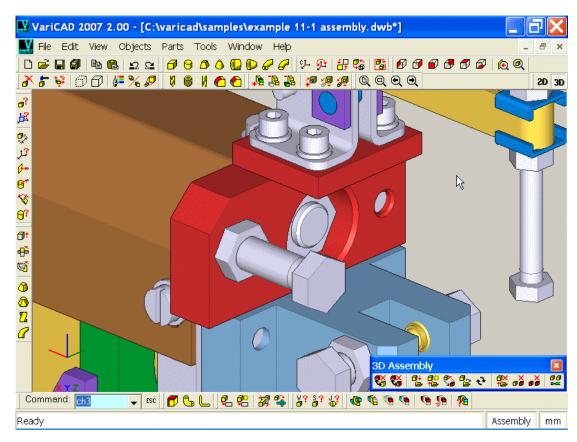
In this example, the assembly link is created, the part is open simultaneously and then edited. After switching back to the assembly, the changes are automatically created for the corresponding solid. For more information about assemblies, see *3D Assemblies (page 143)*.

Selecting a solid to become a part linked to assemblies
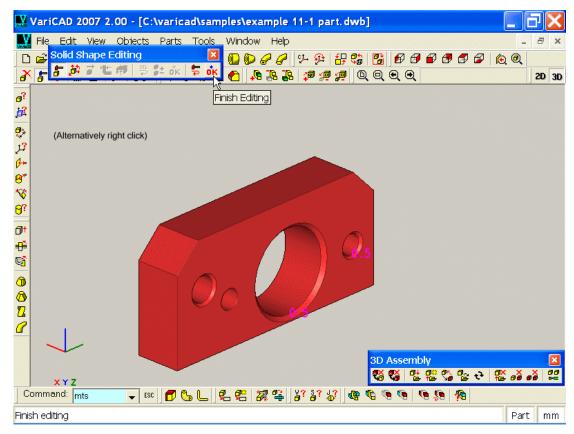
Part edited, switching back to the assembly
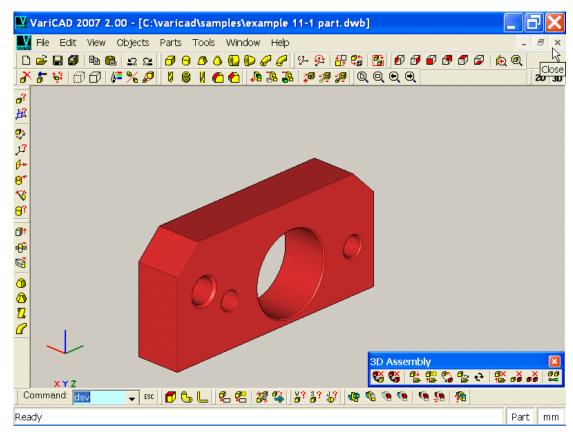
Changes in the assembly after the part was edited

Changing Part
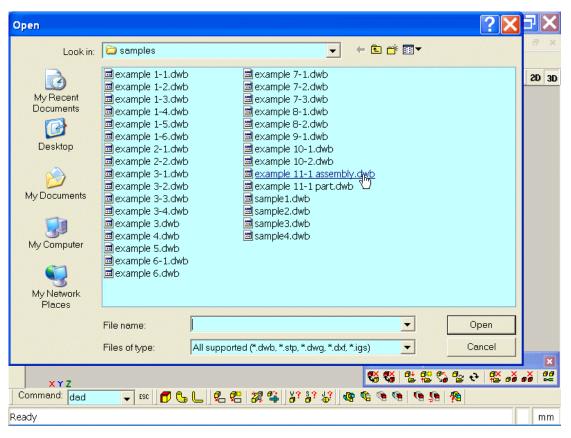
Part is edited, then saved and file is closed. Assembly file is open and the corresponding part is changed automatically.
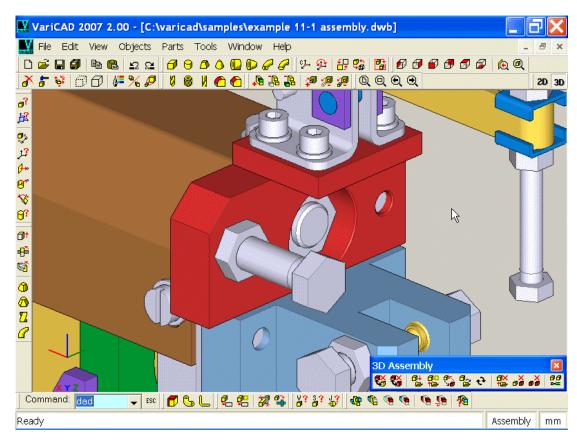
Editing a part

Closing a changed part file

Open the assembly file
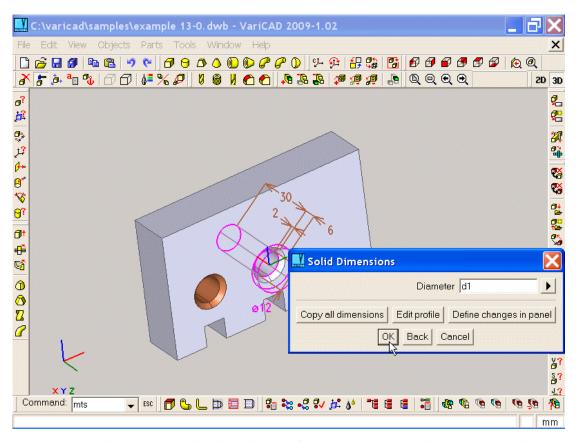
Assembly file containing changed part
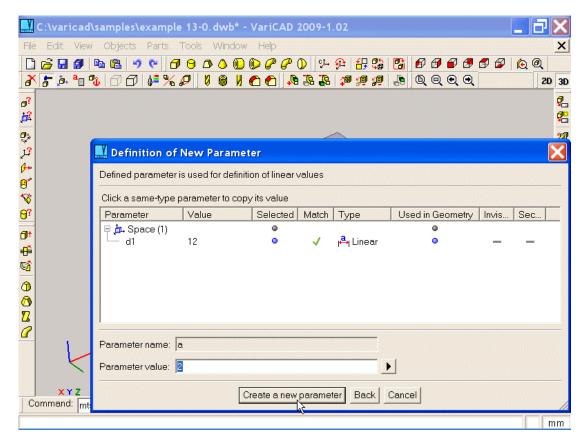
Parameters and Geometric Constraints

Definition of Parameters
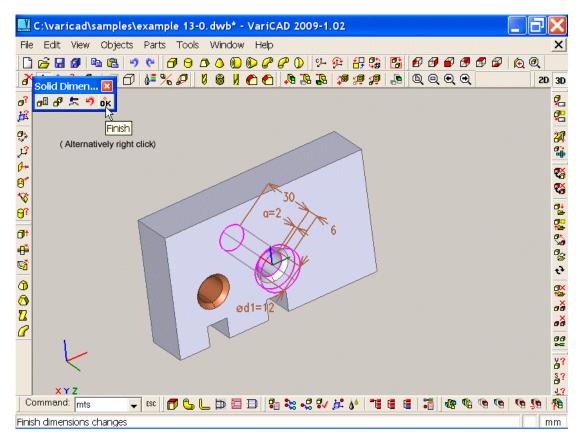
For more information about parameters, see *Parameters (page 119).*

During solid editing, a parameter is defined instead of a constant value. The diameter value 12 is offered as a default value of a new parameter.
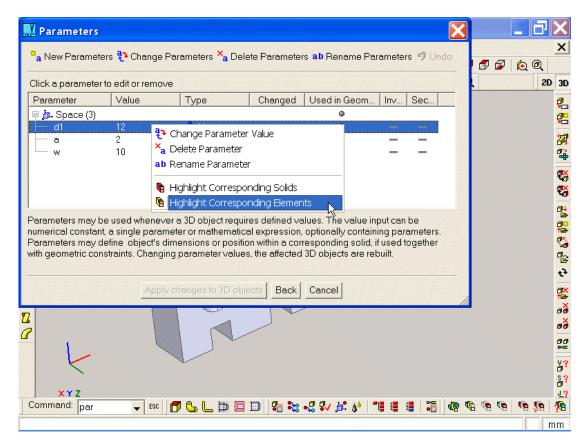
Another new parameter is defined.
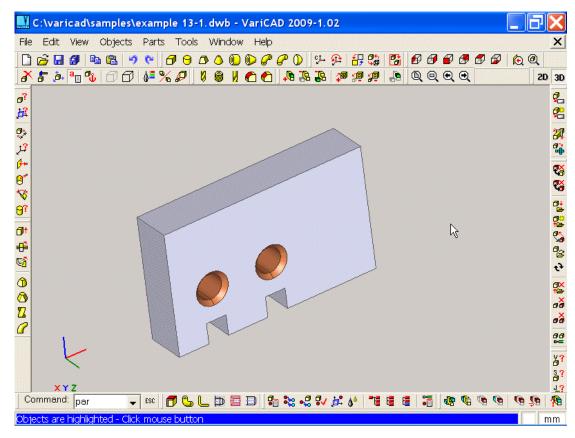
Both holes will contain the changes. Diameter and chamfer values are changed from constant numeric values to parametric ones. In the next step of this example, the width of grooves is also changed to parametric value.
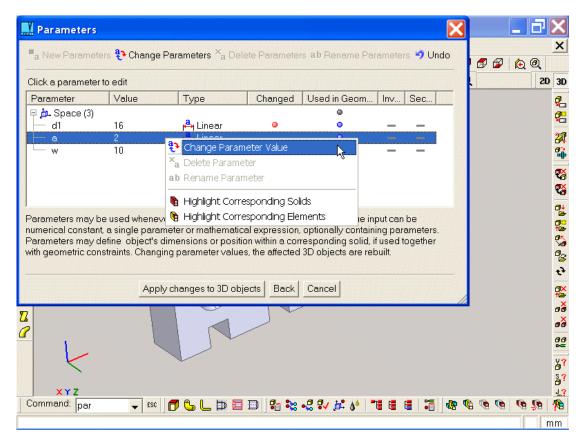
Modification of Parameters
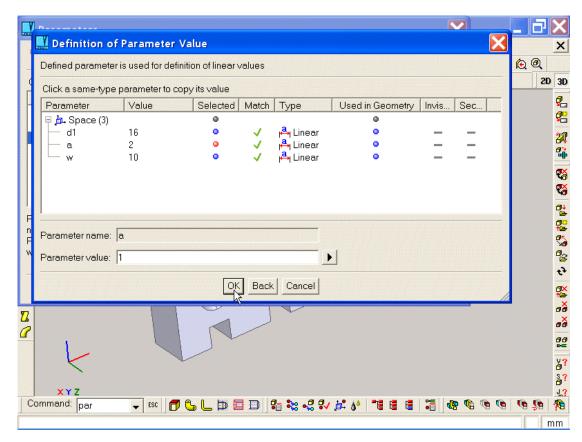
Checking solids containing the selected parameter
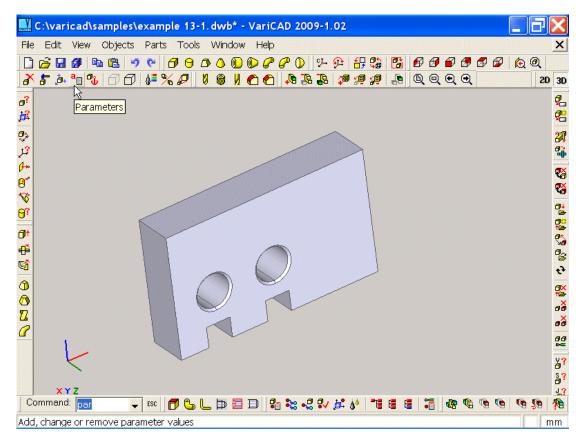
The solids are highlighted

Change of a parameter value is selected repeatedly.
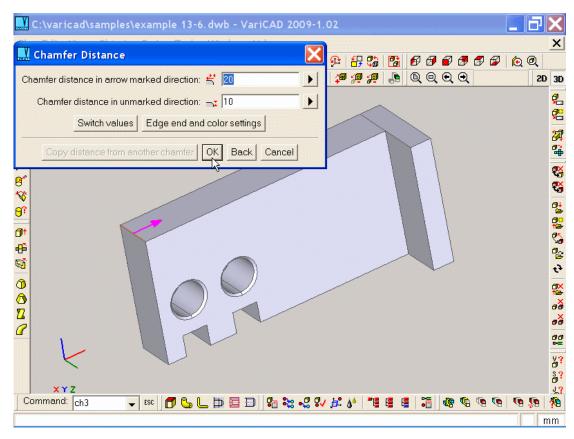
Definition of a new value
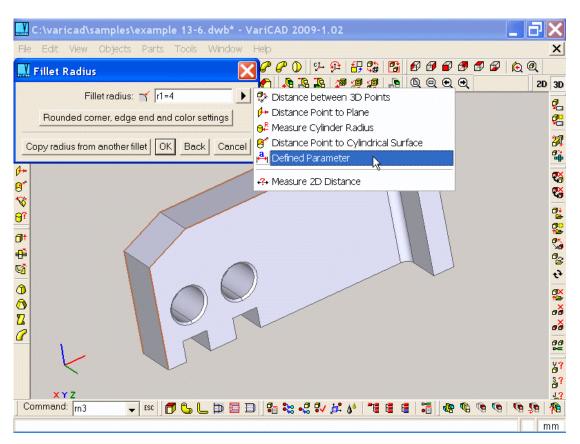
Solid after changes of parameters – diameter of holes, chamfer of holes and width of grooves are changed.
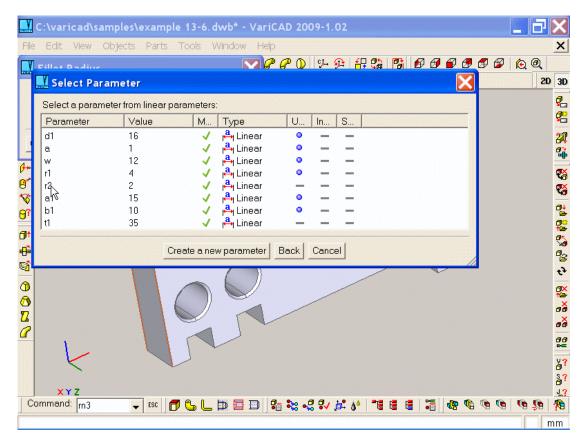
Example of Parametric Filleting and Chamfering
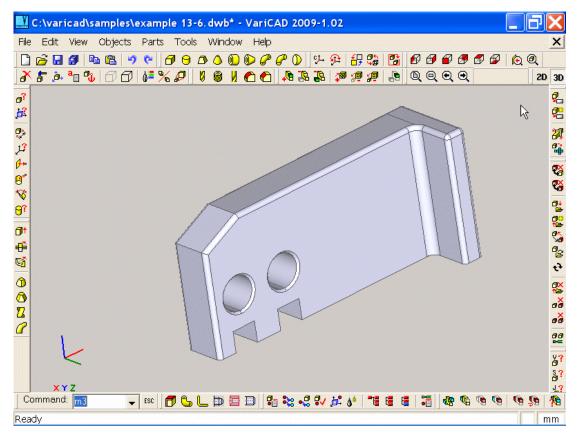
Chamfering the selected edge – the chamfer value is different for each side of the selected edge.
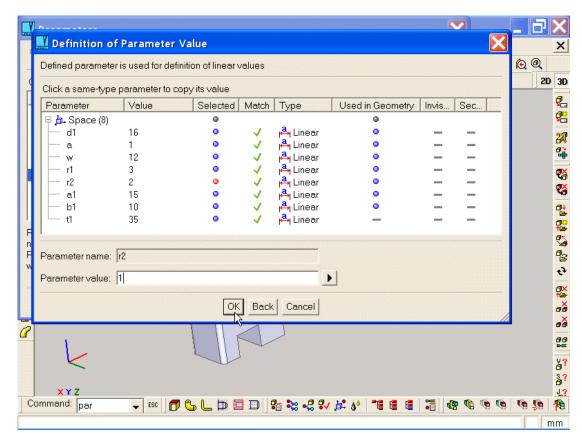
Defining the fillet radius. Parameter r1 was selected for previous filleting. The new value will be selected from a list of defined parameters.
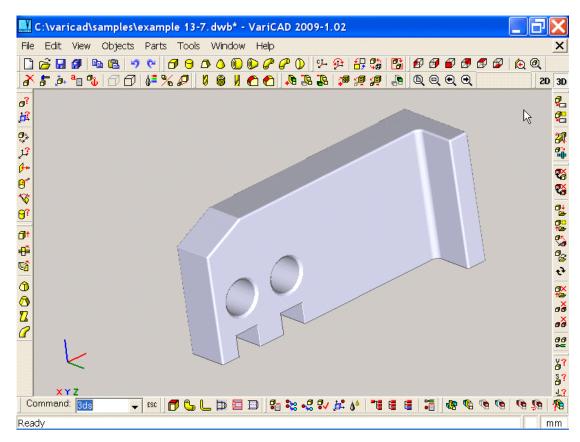
Selecting a new parameter from the list

Blending of edges is finished
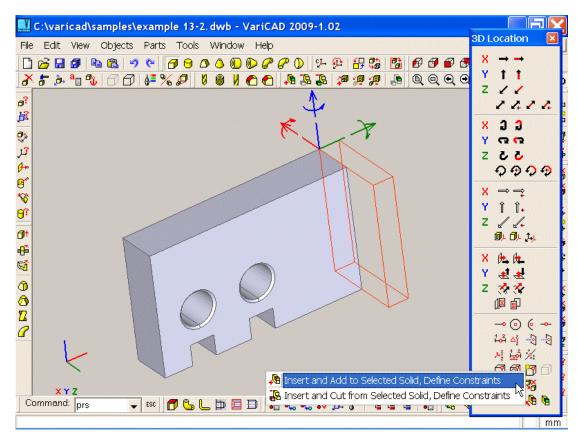
Changing the selected parameter
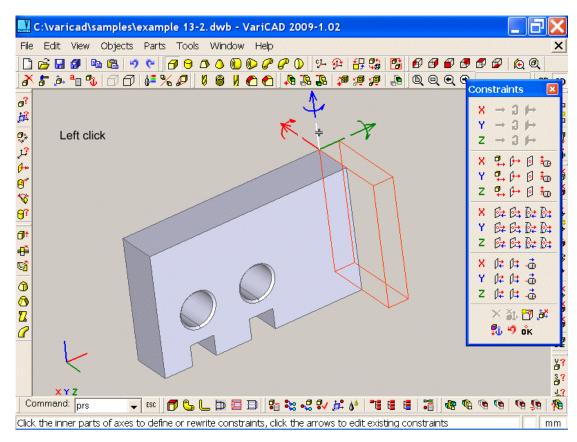
Blending after parameter changes

Defining Geometric Constraints for New Objects
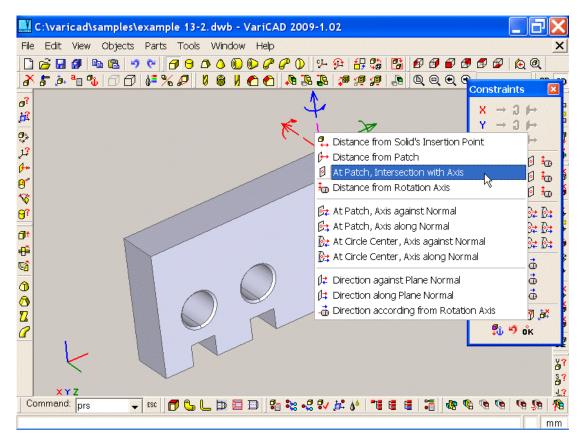
For more information about constraints, see *Geometric Constraints (page 121)*.
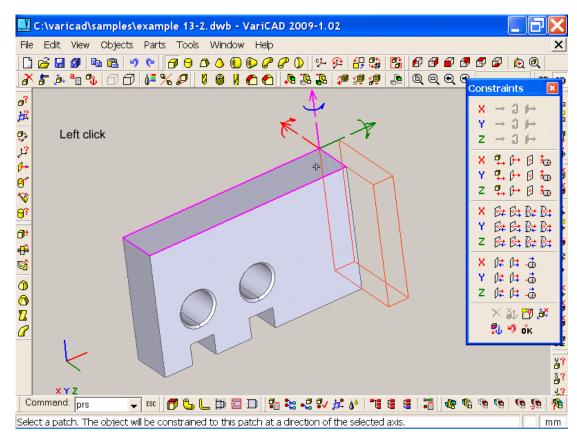
The box was selected and transformed into the current location. Instead of the definitive insertion, a geometric constraint definition was selected. The box will be added to a selected root solid. Geometric constraints can be defined only within a Boolean tree. Constrained location is the location of element's insertion point.
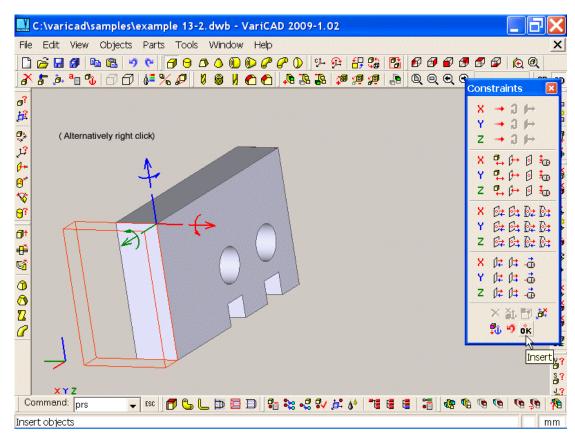
Selecting an axis the constraint is defined for

Selecting a type of constraint
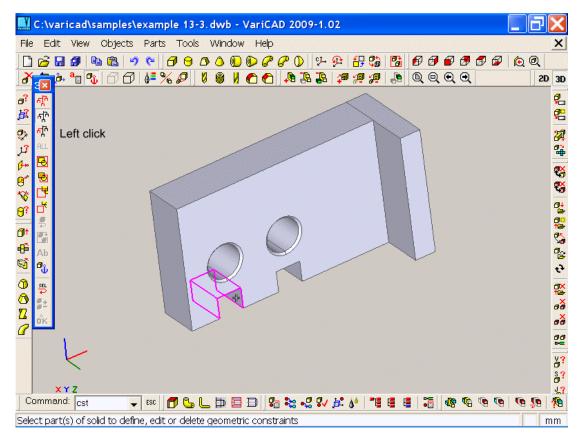
Selecting a patch - this type of constraint fixes the movement of objects along the corresponding axis. The object is fixed at the intersection of the axis and the selected patch. The rest of axes are constrained similarly in the next step of this example.
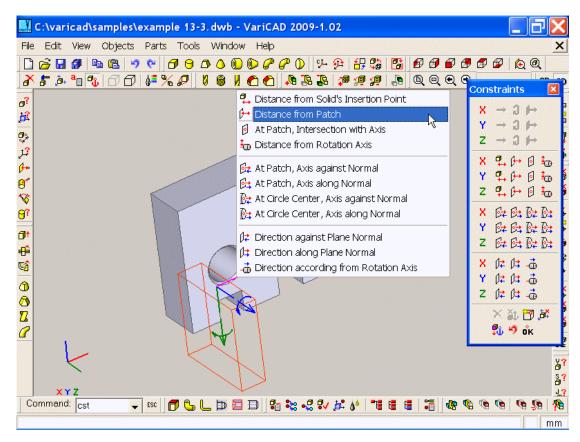
All defined constraints are confirmed. In the upper part of the Constraint panel there are icons displaying constrained movements relative to axes X, Y and Z.
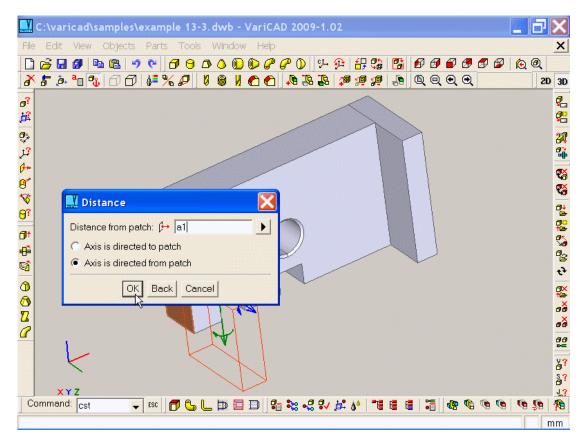
Defining Geometric Constraints for Existing Objects
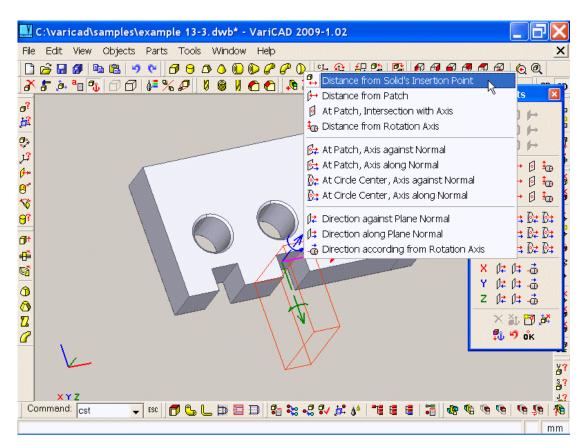
Selecting an existing object the constraints will be defined for
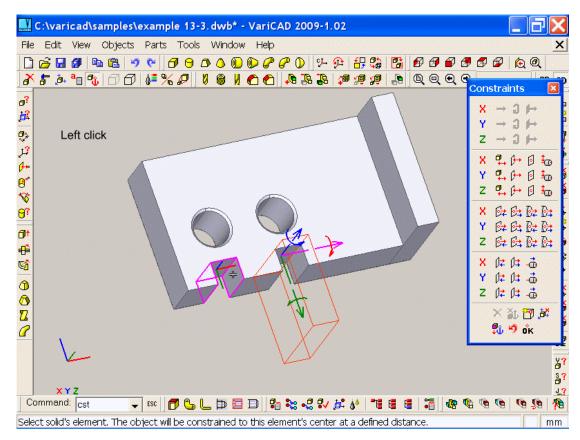
Selecting a constraint, in this case the object will be constrained at a distance from the selected patch.
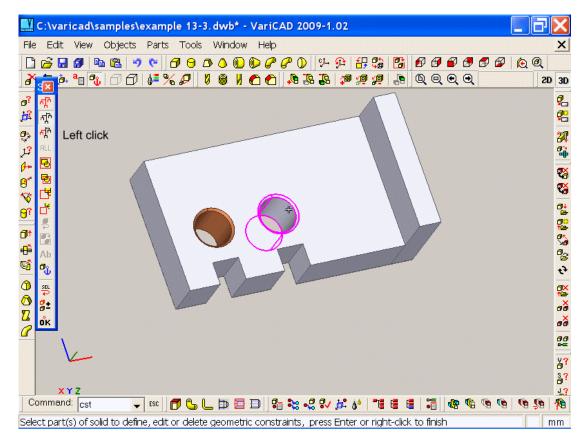
Defining a distance from the patch - a parametric value is selected instead of a constant.
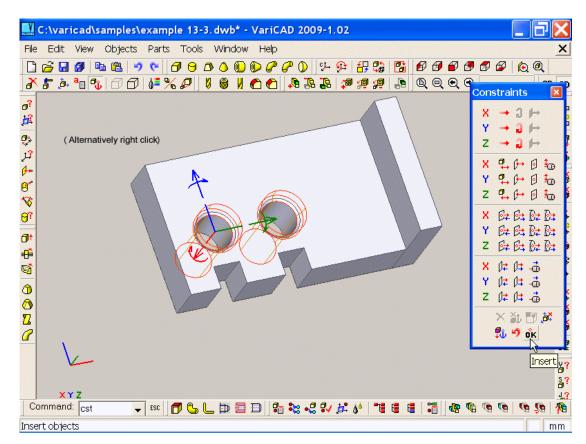
Selecting a constraint for another object, the selected method will constrain object at a distance from a solid.

Selecting a solid - the previously selected object will be fixed at a defined distance. The defined distance is a distance between the two grooves.
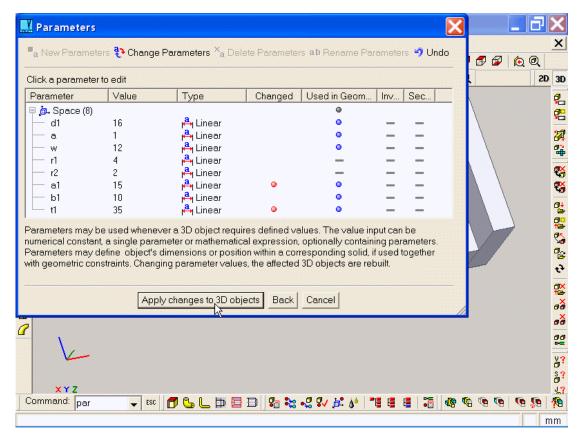
Selecting other solids for constraint definitions. Constraints may be defined for multiple objects. Any changes of position within the Boolean tree are performed for the entire constrained group at a time.
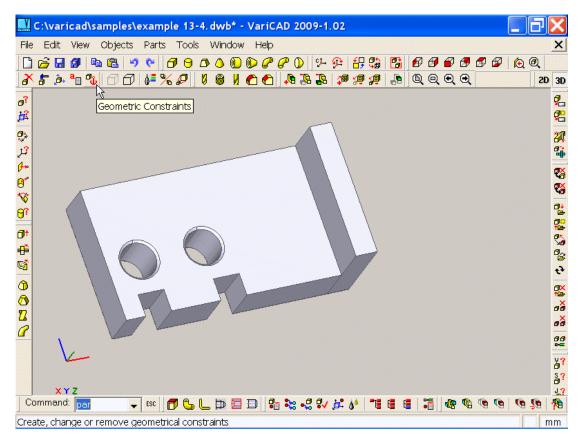
Finishing constraint definitions for a pair of holes. The holes are constrained with two types of constraints. First, they are fixed at a distance from selected patches. Second, the axis of the hole is directed against the normal of the selected patch and the insertion point of the hole is fixed on the surface of this patch.

Changing and Checking Geometric Constraints
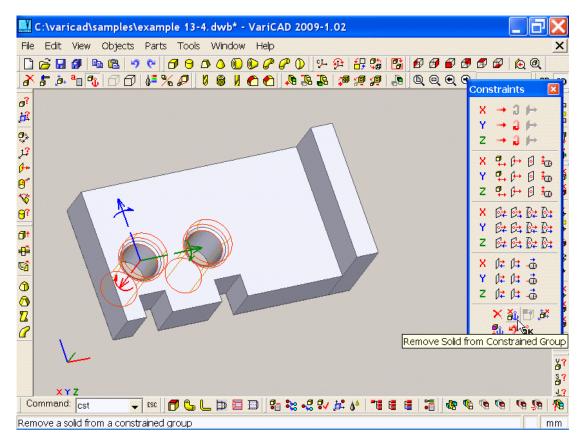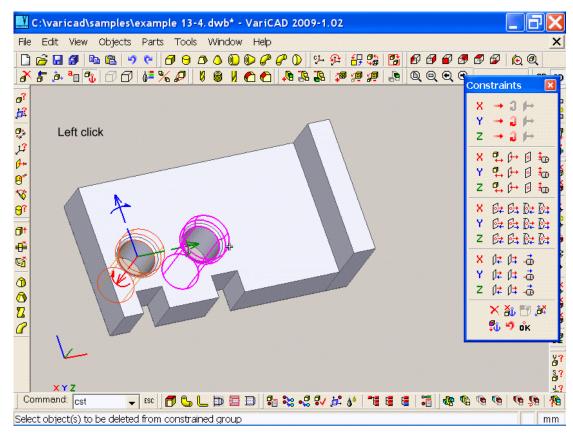
Constraints defined in the previous example contain parametric values. Here, new values are selected.
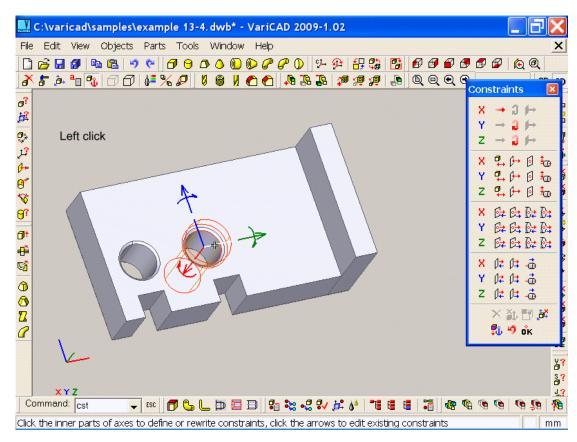
The solid was rebuilt after changes of parameters. The pitch between the grooves at the bottom is defined by geometric constraint. After the change, the pitch is different. The pitch between the holes remains constant. Moreover, both holes are constrained as one group. If you need to fix both holes at the grooves below individually, the hole at the right side needs to be removed from the constrained group. Then, define its constraints separately.
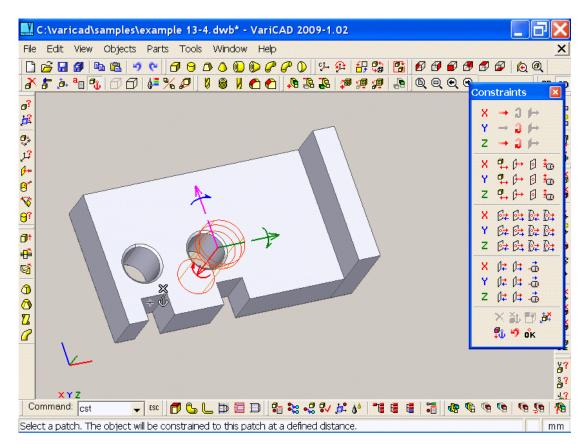
Selecting a removal from the constrained group
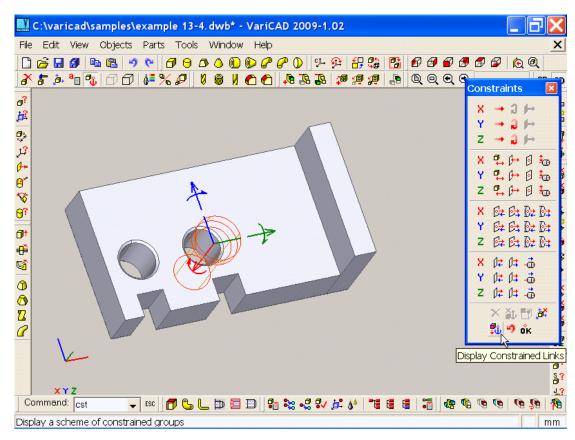
Selecting an object to be removed from the constrained group
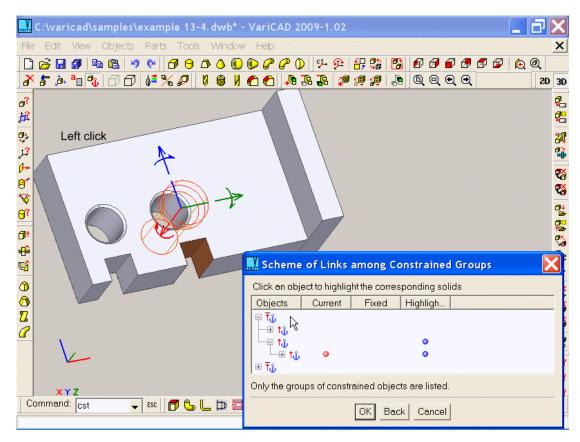
Definition of constraints for the second hole

A constraint at a distance from the patch was selected. Because the detected patch belongs to an object constrained in a different order, the selection is blocked automatically. If a constraint was performed, such a solid would not be rebuilt properly.
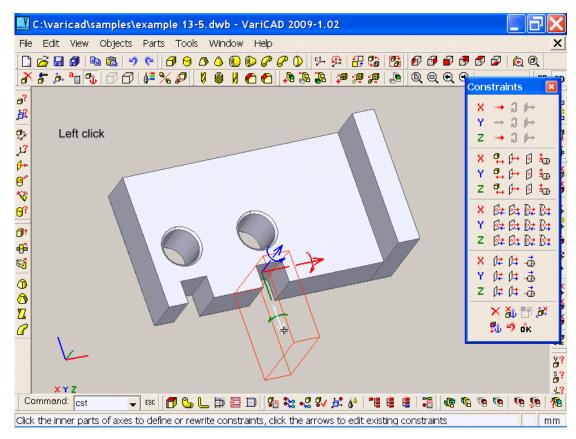
You may display a scheme of constrained links. The scheme highlights links between objects. Blocked selections are more obvious, if you click an object in the list. You can see an anchored object as well as an object the selected one is anchored at. Obviously, you cannot anchor an object A to an object B if the object B is already anchored to the object A (directly or via a chain of multiple constraints).
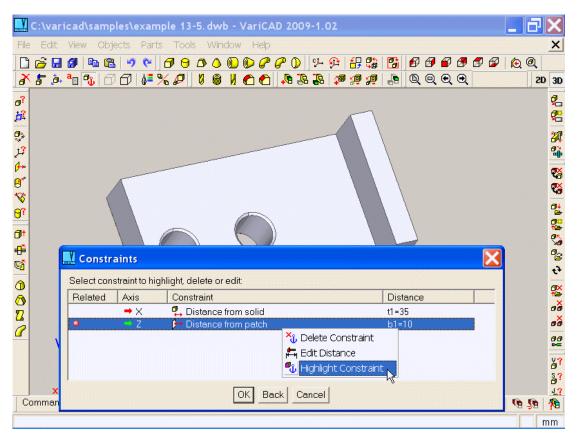
Scheme of links among constrained groups. Selected objects are highlighted.
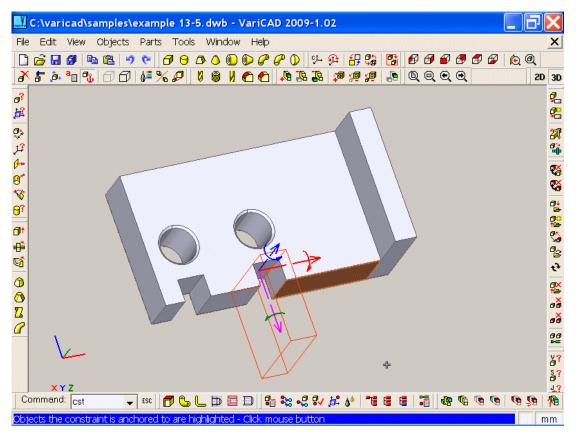
Editing Constraints, Rebuilding Solids
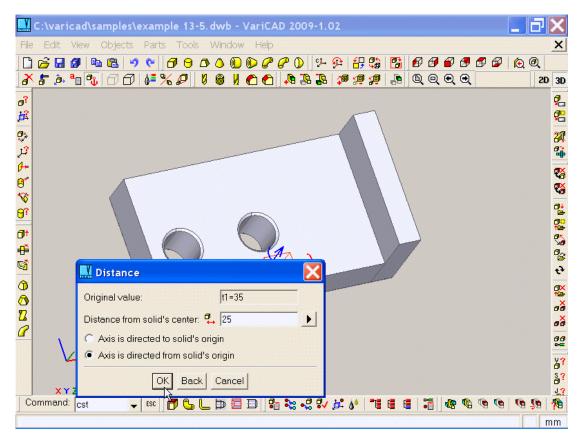
Selecting a constraint to be edited
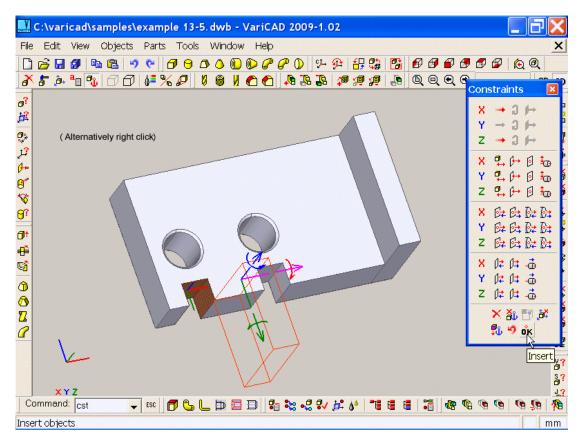
Highlighting of constraint is selected – you may highlight a selected constraint to see either constrained objects or anchors.
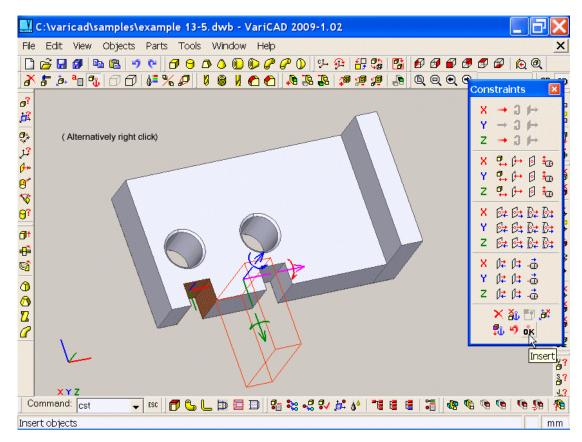
Constraint is highlighted
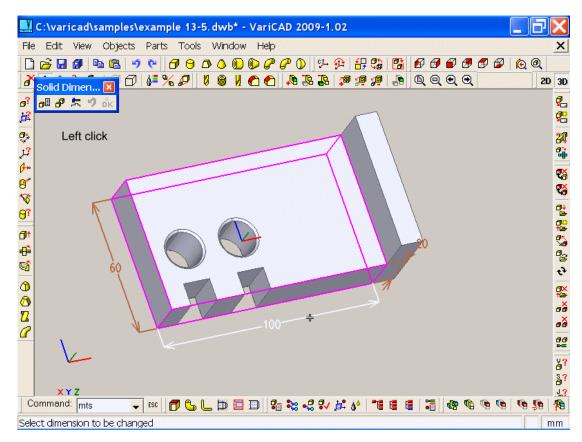
Second constraint was selected for distance editing. This constraint defines a pitch between groves. A new value of the pitch is defined.
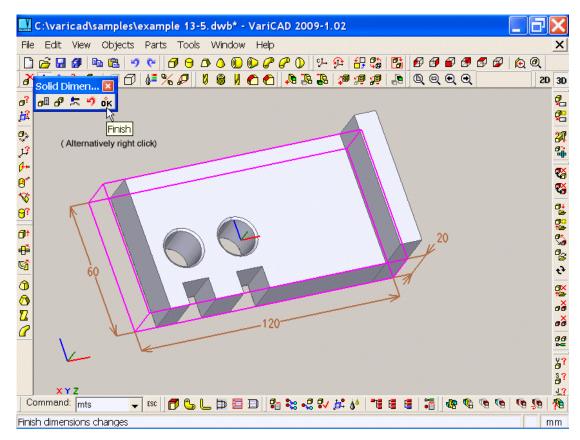
Constraint change also transforms corresponding objects.
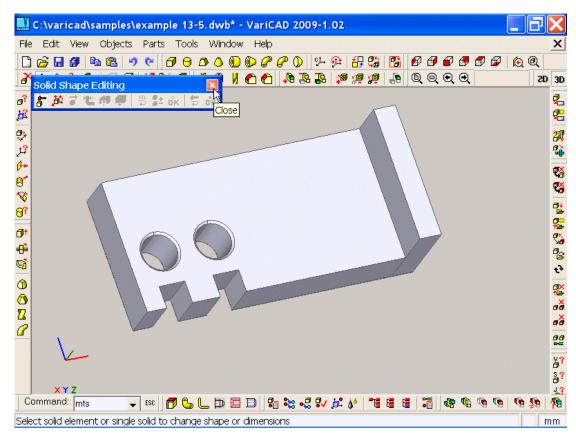
All constrained objects are transformed. Here, movement of the hole at the right side is constrained at the groove below. After the groove changes its position in X axis direction, the hole is automatically moved.

A change of the dimension of the box is selected.

The shape of the box is changed. It is extended to both sides in X axis direction. Without any geometrical constraints, the rest of all objects would remain at the current positions.
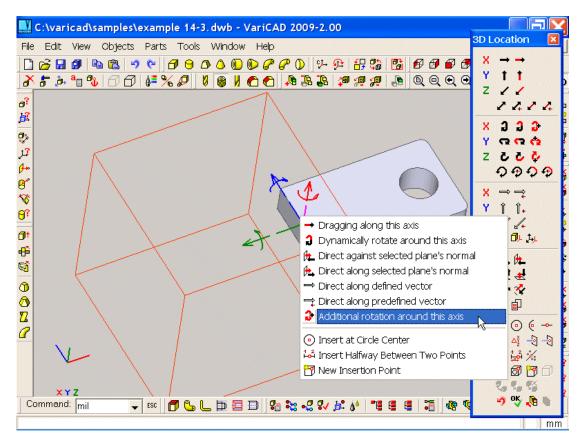
All shapes and positions were changed. The box at the right side is constrained also to the right patch of the edited box. After shape changes, this object is moved to the right automatically. The holes and grooves are constrained also to the left patch of the edited box. Consequently, they are moved to the left after the shape changes.
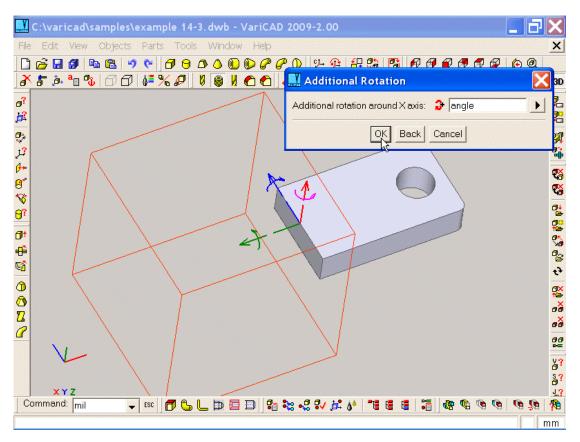
Constrained Angle

VariCAD allows you to perform additional rotation of transformed objects around an axis. Such rotation may be beneficial for transformation of solids. Objects are rotated and the axes directions are not changed.
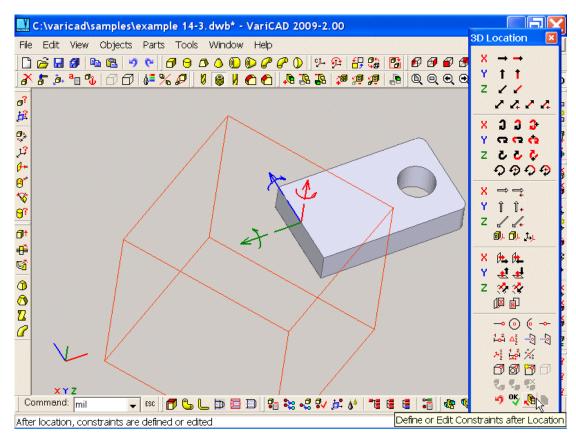
Another very convenient usage of the additional rotation around axes is a possibility to constrain objects under an angle. In such case you have to define an angle with a parametric value and the objects must be constrained. Whenever the angular parameter is changed, the constrained objects are rotated according to the new angular value.
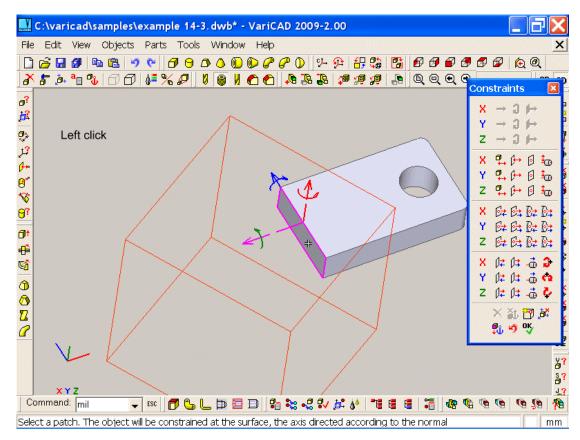
Function "mill" is performed. Cutting box is additionally rotated around the X axis.
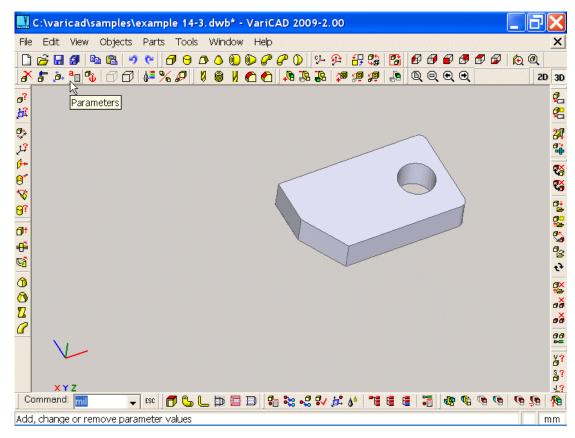
As value of rotation, a parameter named "angle" is selected.
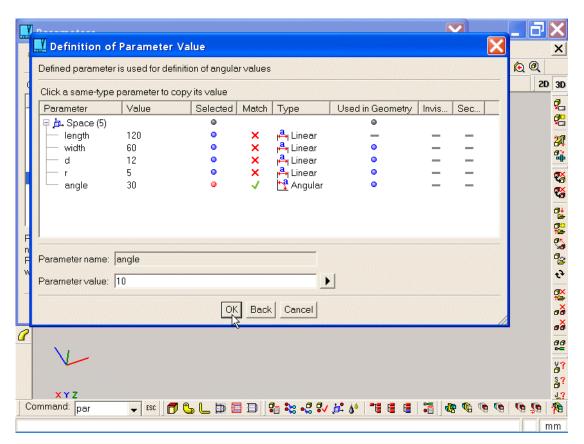
Value 30 degrees was assigned to the defined parameter. Here, the object is already rotated around the X axis with angle of 30 degrees.
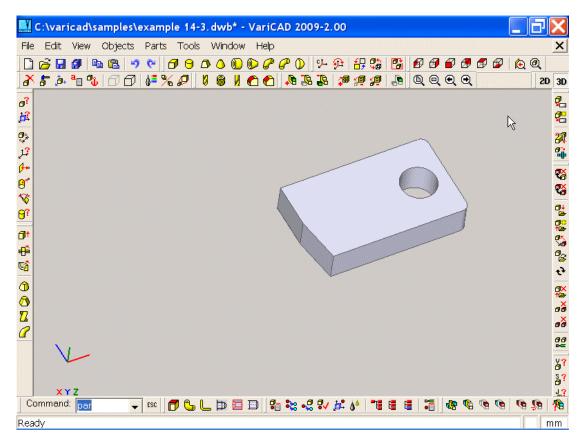
A constraint is defined – Z axis perpendicularly to a selected patch. The defined constraint allows further proper rotation of objects, if the value of the angular parameter is changed.

Solid cut under an angle as a result of previous operations.

A new value of the angular parameter is defined. Other parameters in the parameter table cannot be used for value copying – they are linear parameter, the changed parameter is of angular type.
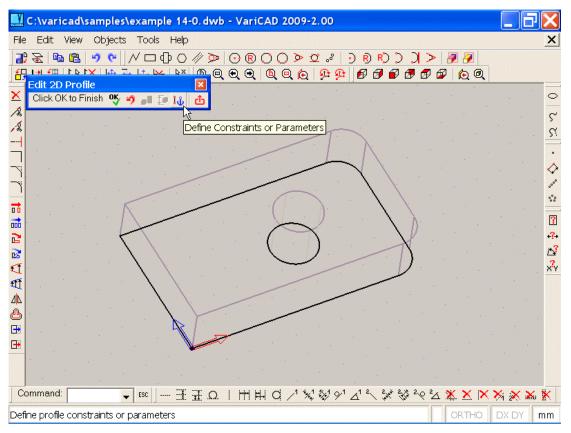
Solid after parametric change of the cutting angle

Parameters and Constraints in Solid Creation Profiles
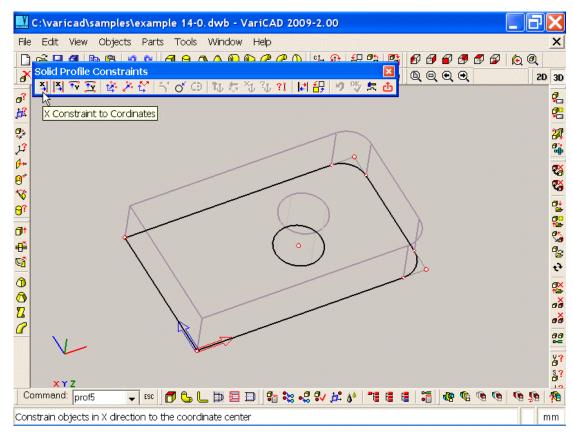
You can define constraints for a set of vertexes of a 2D creation profile. Vertexes can be constrained relative to another vertex or to the center of the coordinate system. You can also define radius of circles or arcs as a parametric value.

The vertexes are endpoints of lines, circle centers, intersections of lines or arcs (in case of filleting or blending) or interpolation points of NURBS curves.
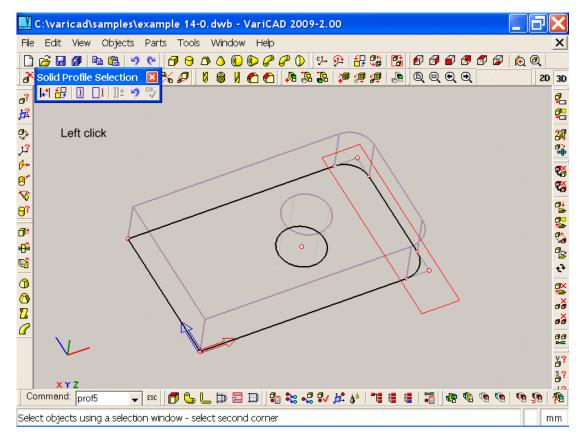
Using parameters and constraints in lines and circles
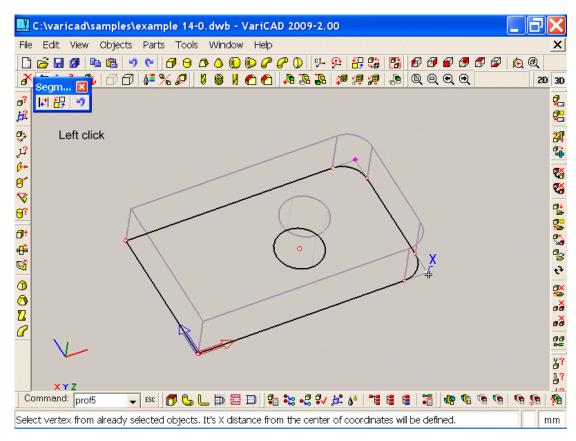
To define constraints and parameters within a solid's creation profile, first select a solid for editing. Then switch 2D edit mode to constraints definition – click the corresponding icon in the edit toolbar.
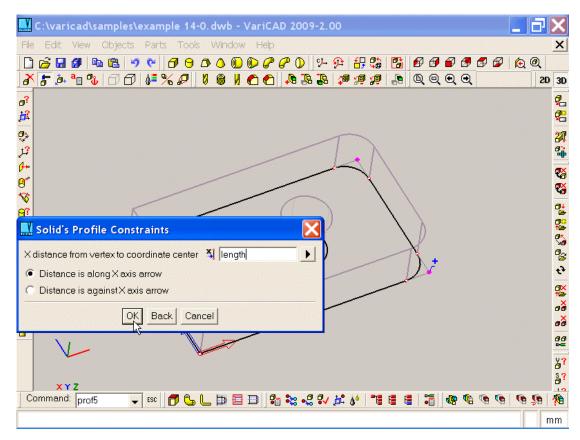
Selection of a method of constraining
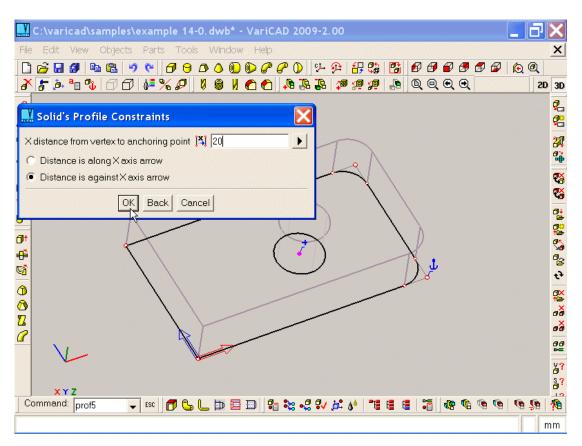
Selection of vertexes for constraint definition. Selected vertexes will be constrained to the coordinate center at a distance.
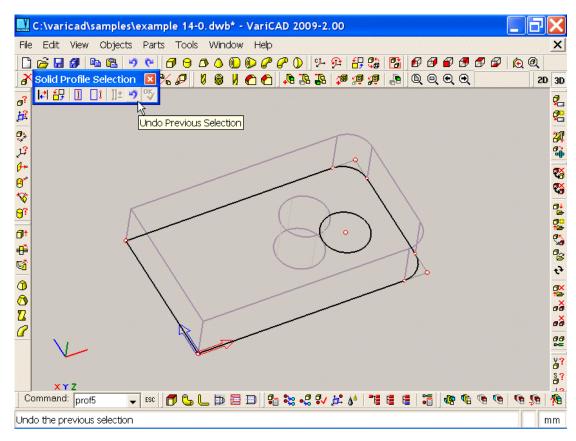
Reference point is defined. The constrained distance will be distance between the coordinate center and the selected reference point.

Distance is defined. Instead of a constant value, a new parameter named "length" is created. In the following step, a numeric value will be assigned to this parameter.
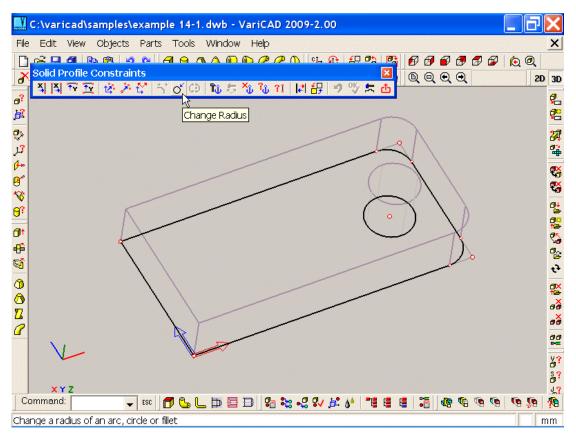
In this example, the circle center is constrained at X direction to the selected vertex (marked with the anchor).
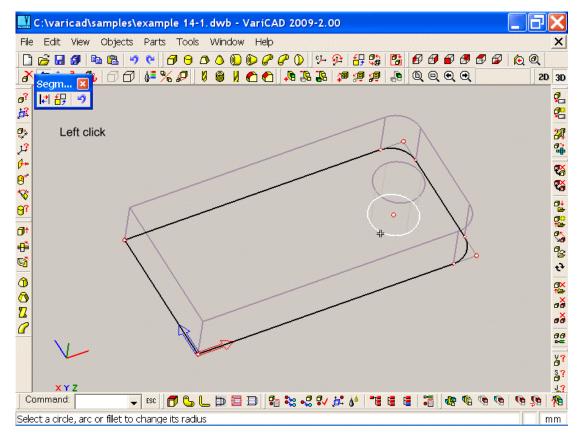
After distance definition, the circle is moved. Distance from the center to anchoring point was defined shorter than the previous distance.
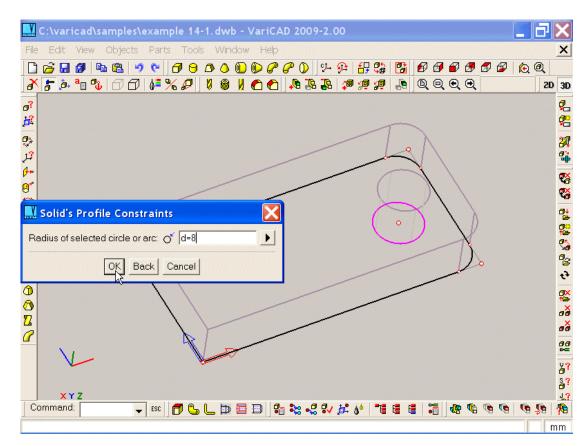
Circles and Fillets Radii
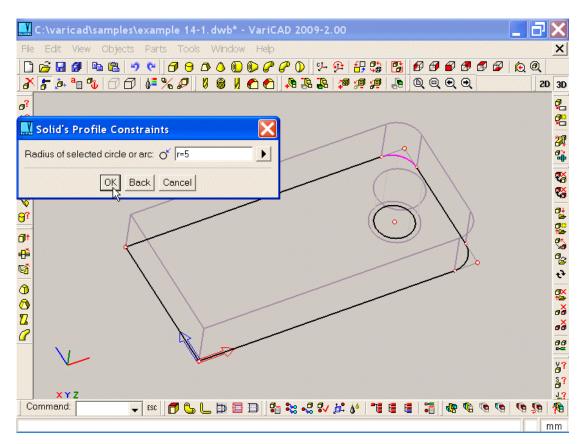
Selecting change a of circle or arc radius
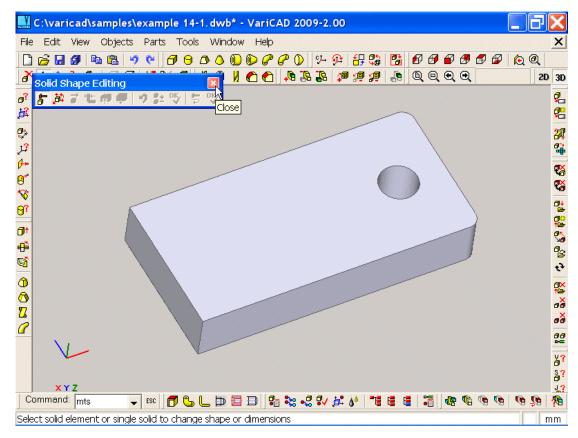
Selecting a circle for radius change

Radius definition, the parameter is used
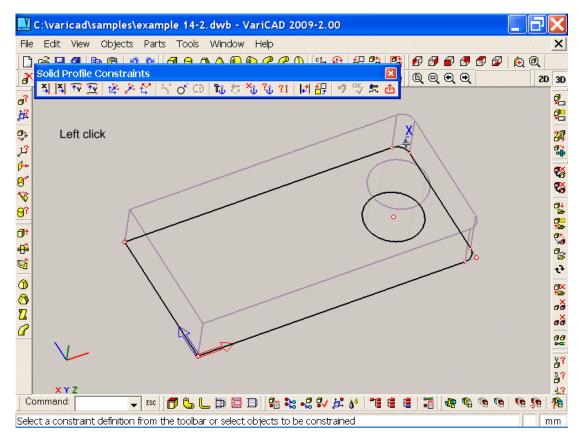
Definition of a radius of a fillet. The radius of the circle has been changed; the new value is smaller than the previous value.
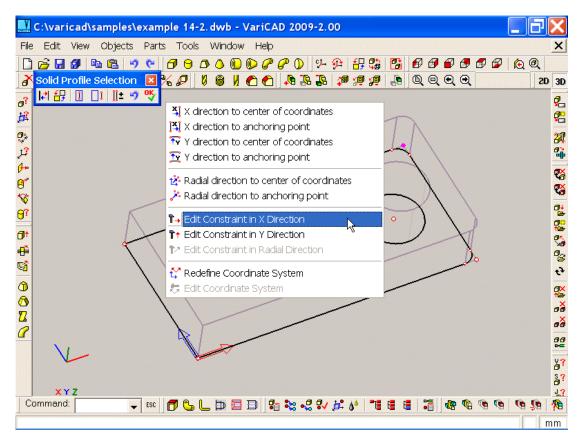
Circle and fillets after changes

Editing an Existing Constraints
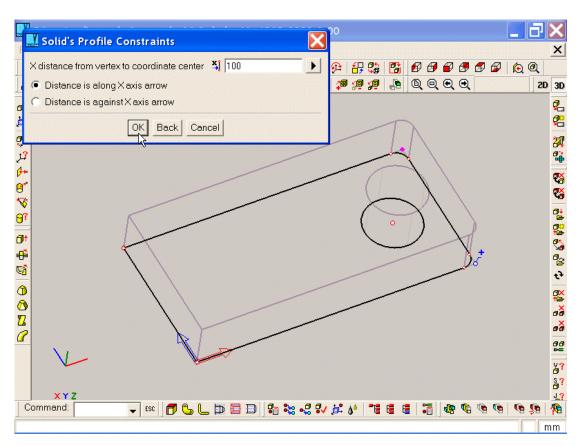
Different method of selection is used in following examples. You can click an object (vertex, chamfer, fillet, arc or circle) and then select an action from the pop-up menu.
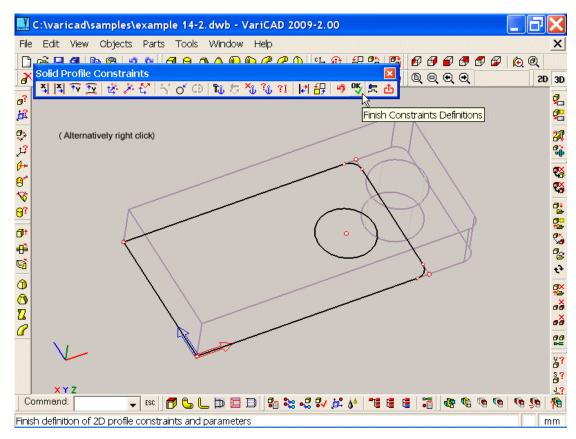
Selecting a vertex

After finishing selection, an action is selected from the pop-up menu
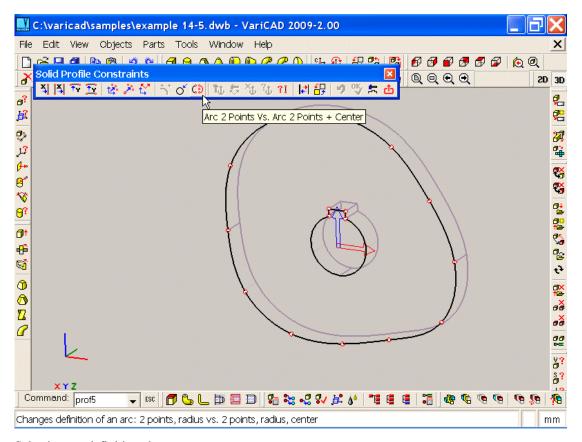
A new distance is defined for the group of constrained vertexes.
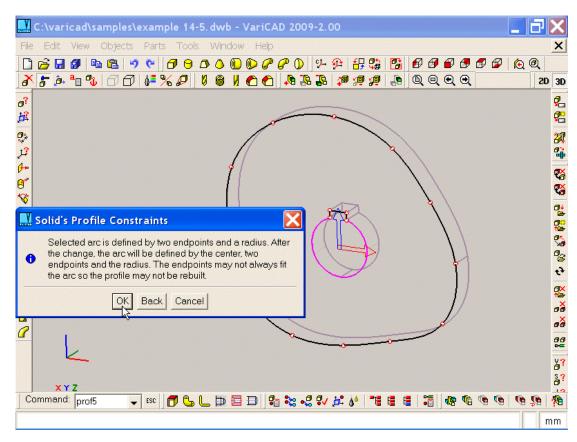
The profile after change of the constrained distance. In this example, the two right vertexes are constrained at a distance to the center of the coordinate system. The center of the circle is constrained to the vertex from this group. If the constrained distance of the right vertexes is changed, the position of the circle is changed together with the new length.

Circular Arcs
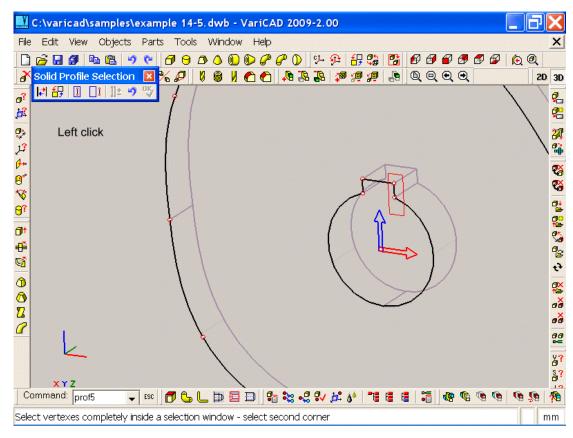
By default, arcs are defined by two endpoints and radius. If necessary, you can add the arc center to the definition. In such case, the endpoints must be explicitly calculated if they are constrained - otherwise they may not be at the radius distance from the center.
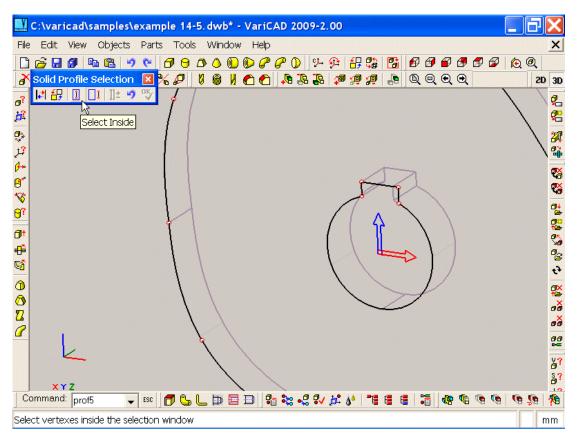
Selecting arc definition change
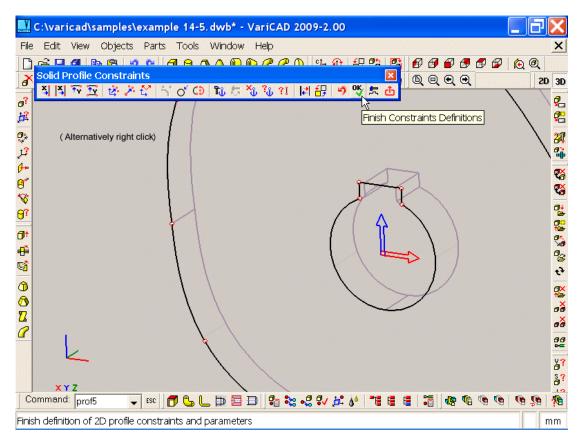
Confirmation of change after the arc was selected

Selecting vertexes for a constraint in X direction to the coordinate center
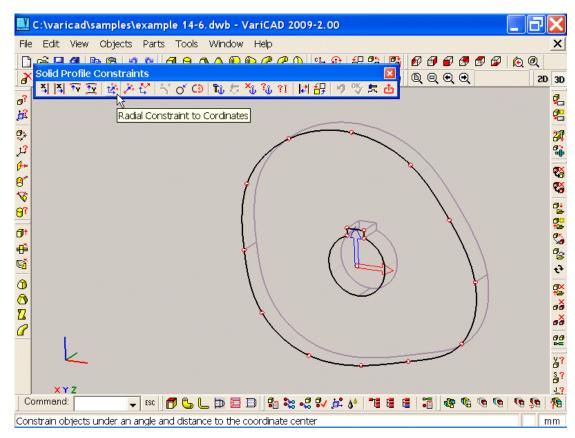
A new distance for the vertexes was defined. Because the vertexes are constrained in one direction, the vertex common for the arc and the line is recalculated automatically. Distance in X direction is defined for the constraint, the vertex must be also at the radius distance from the arc center. Location of vertex is calculated automatically.
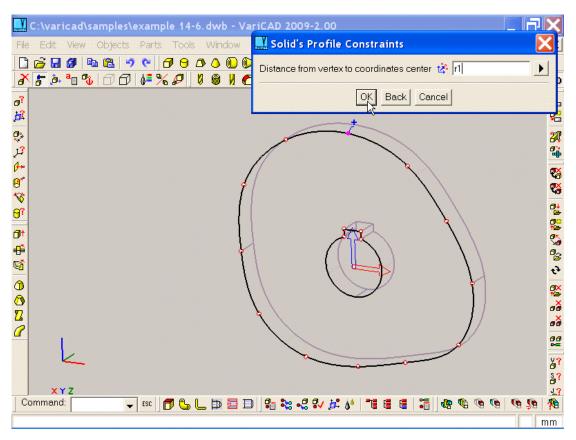
Opposite vertexes were constrained similarly.

NURBS Curves
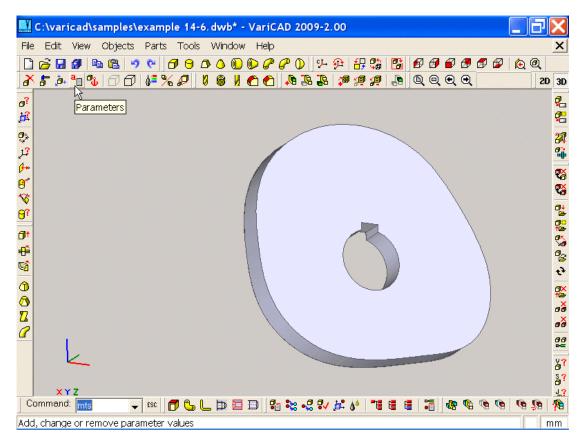
NURBS curves can be modified by parameters and constraints, if you constrain the curves interpolation points.
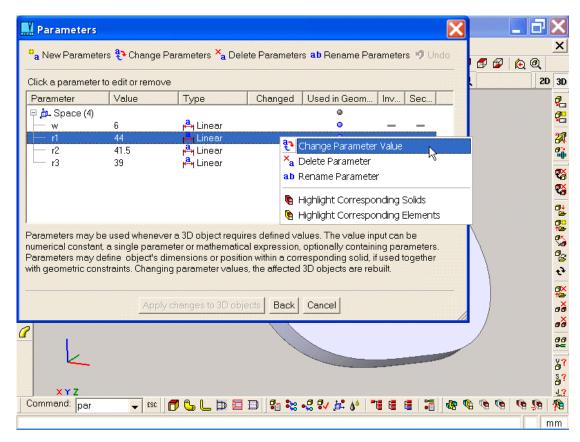
Selecting a method of constraint
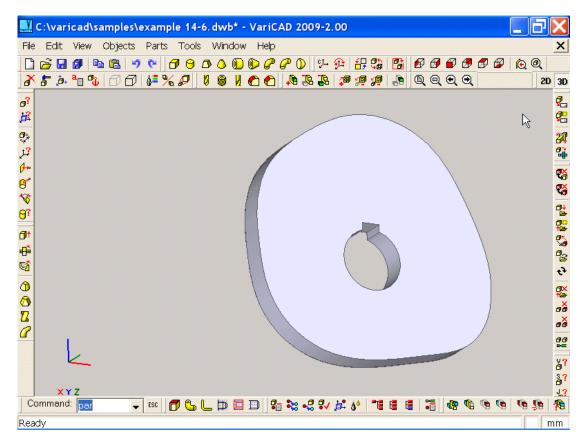
Definition of a distance to the center of coordinates. The distance is defined as a parameter. In following steps, the parametric distance is defined for multiple interpolation points.

The cam before change of the shape – value of parameters was not changed yet
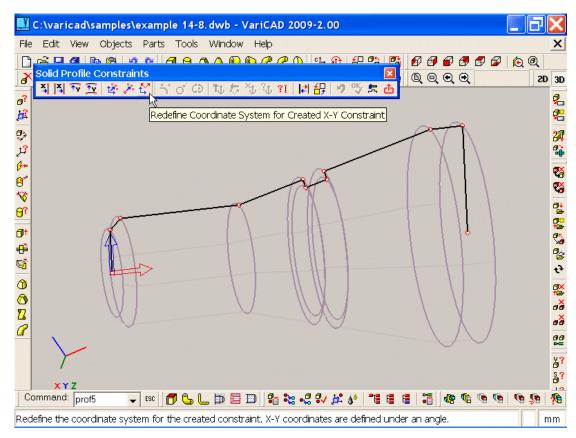
Changing distances of points from the center – the parameters are modified
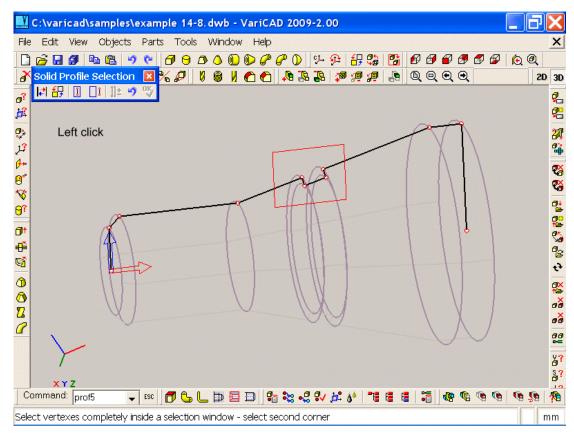
The cam after parametric change of the shape

User Defined Coordinate Systems
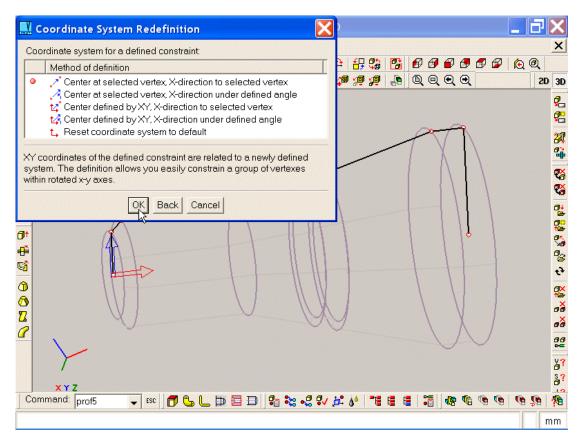
You can redefine the coordinate system of vertexes to work more conveniently with them. The coordinate system allows you to move the origin or to rotate XY axes.
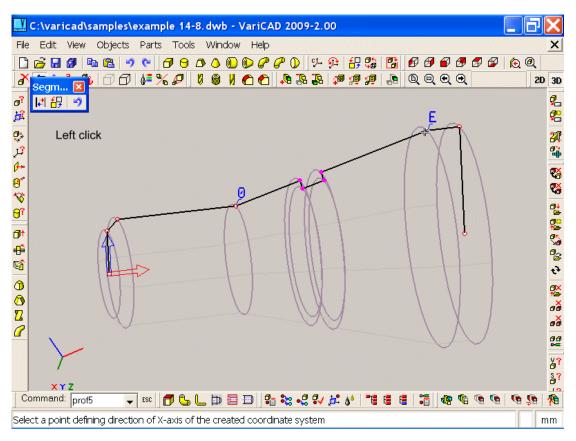
Selecting redefinition of the coordinate system
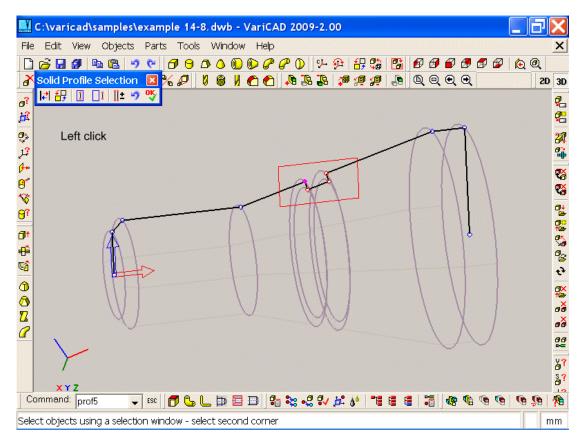
Vertexes selected for change of the coordinate system
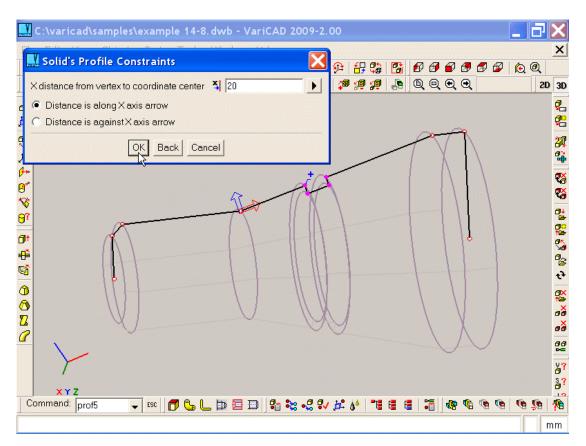
Method of coordinate system redefinition
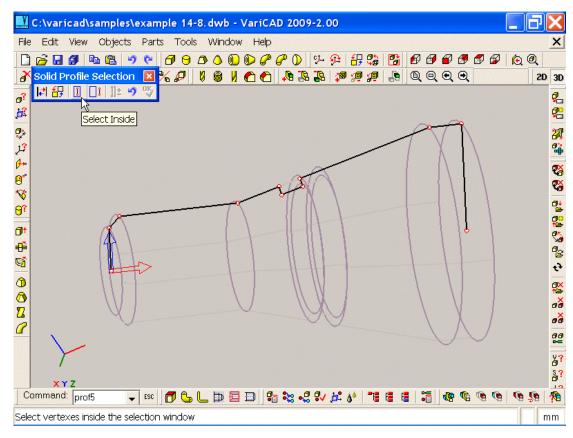
In previous step the coordinate center was selected (a corresponding vertex is marked with '0'). A vertex defining direction of X axis is being selected.
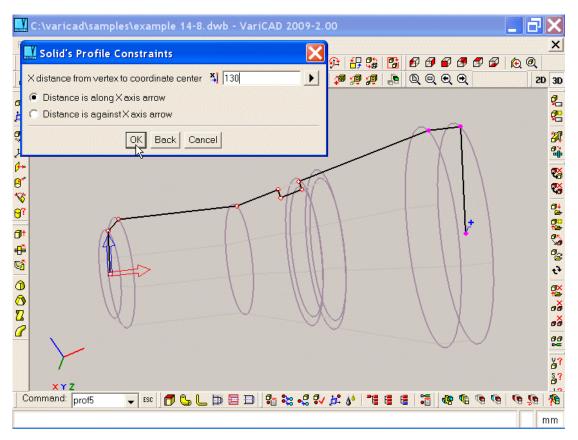
Selecting vertexes for constraint definition. If a vertex is selected and the vertex belongs to a group of vertexes with redefined coordinates, you cannot select vertexes with different coordinate systems – otherwise the constraints cannot be defined. In this situation, the not-selectable vertexes are redrawn in different colors.
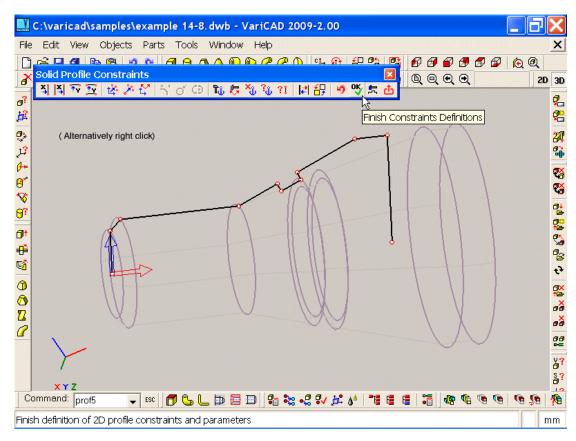
Constraint distance is defined.

The vertexes were moved according to the newly defined constraint.

Another constraint is defined – this time for the right three vertexes. A new distance to the coordinate center is defined as shorter than the previous distance.

After performing the new constraint, vertexes were moved. Because one of them defines direction of X-axis from the previously redefined coordinate system, corresponding vertexes are transformed, too.
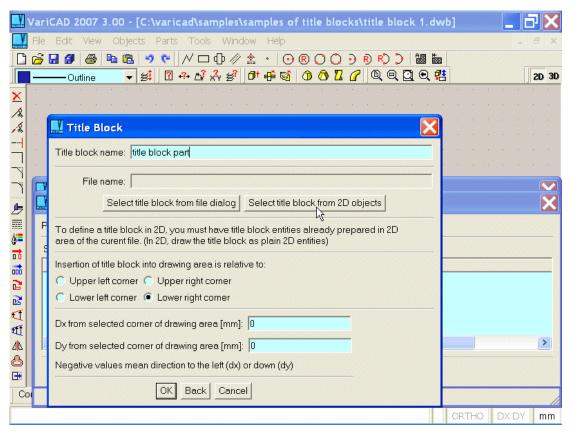
Bill of Materials, Solid Attributes, Title Blocks

Title Blocks Definition
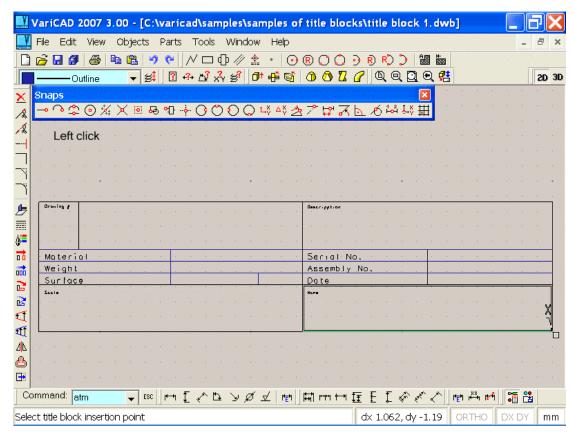
Before you will use the attributes, BOM and title blocks permanently, you should consider their proper configuration and change the settings according to your customs. For more information about settings, see *BOM, Attributes and Title Blocks Settings (page 149*.

Title Block Definition

First, create a title block in 2D as plain 2D objects. In this example, the title block is defined from such an existing drawing.
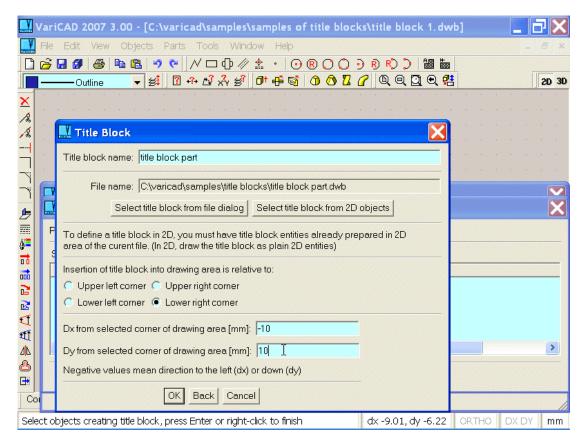
Title block name definition

Title block insertion point definition

After the insertion point is defined, select objects of the title block.
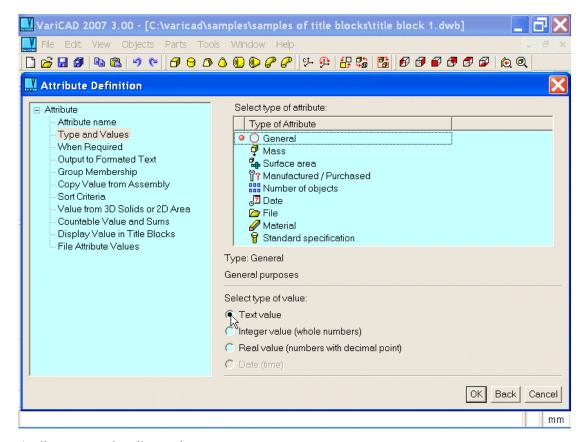
 Insertion point relative to the selected corner of 2D area
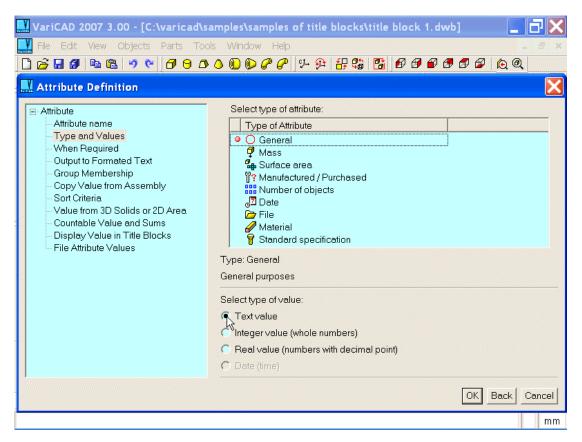
Selected 2D objects are saved into the defined file. Then the position of the title blocks insertion point relative to 2D area is defined. The title block is ready for use.

Attributes Configuration
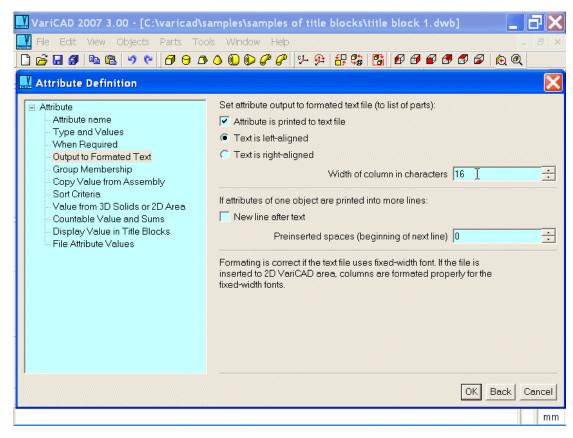
In this example, a new attribute "dwg. no." is created. The attribute type is defined as "General" type, value is "Text" value, the attribute is configured to be printed into formatted text and to be displayed in the title block. The attribute will be required for all manufactured parts.
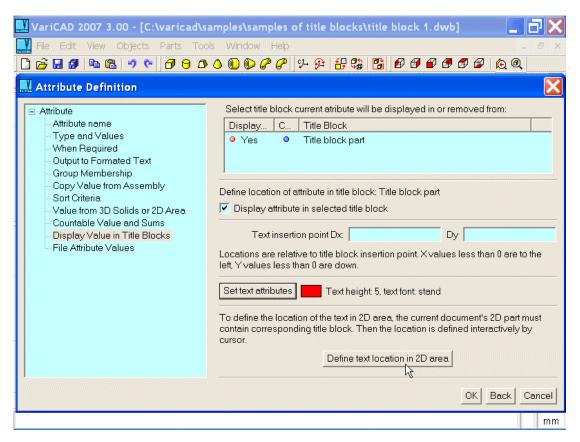
Attribute type and attribute value
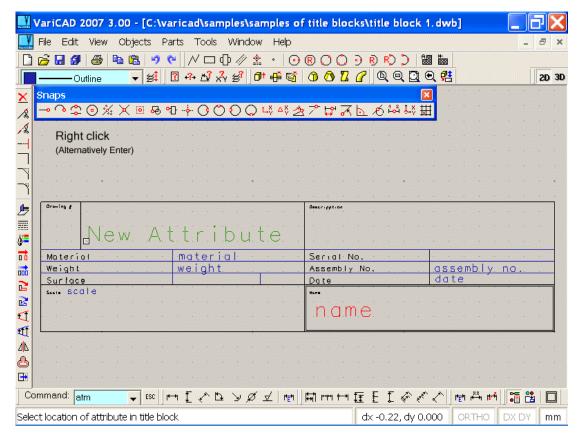
When a new attribute is required

Definition of output to formatted text files (used as a list of parts)
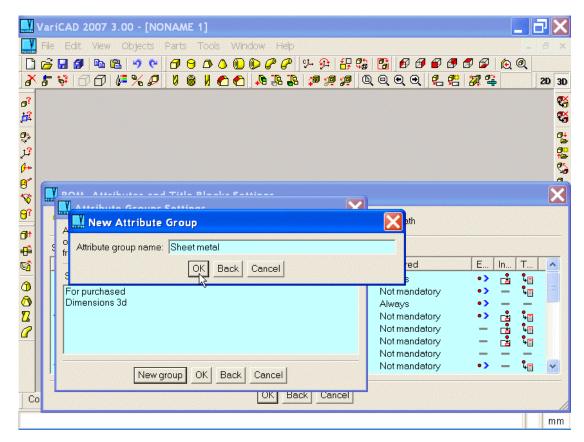
Definition of output to the title block
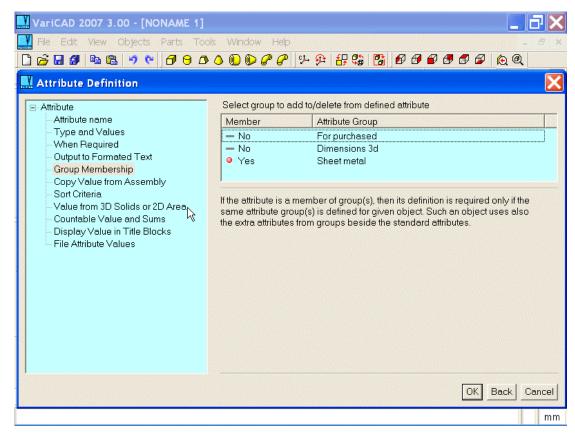
Attribute position in the title block

Attributes Groups

The 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.
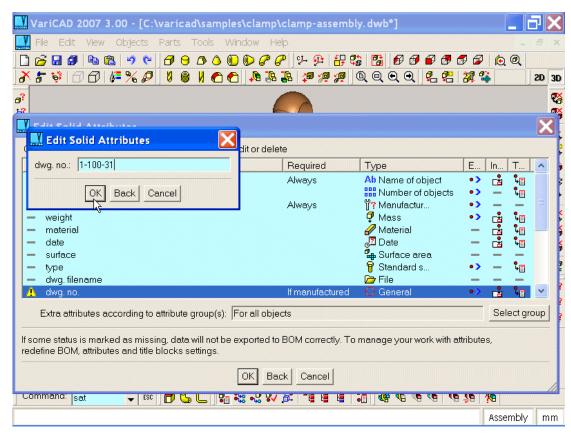
New attribute group definition

Group membership for the defined attribute
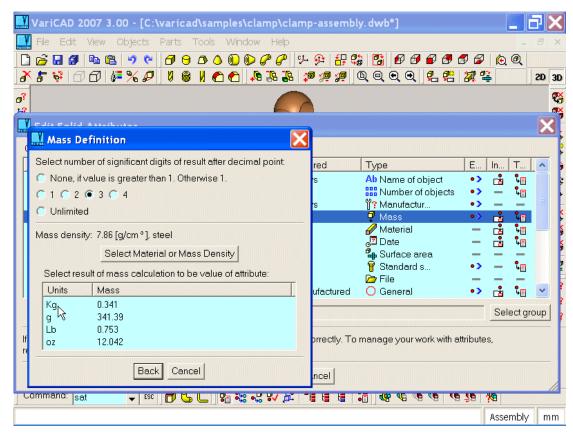
Solid Attributes

This example shows you how to define solid attributes of various types.
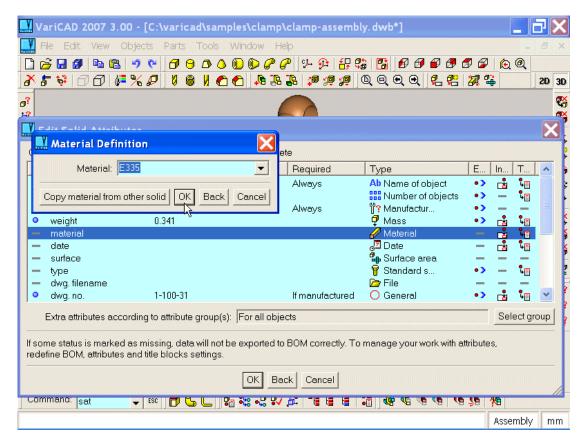
Definition of the attribute "dwg. no."

Attribute "dwg. no." was configured as "Required for Manufactured Parts". If such an attribute is missing, the warning sign is displayed in the first column of the attributes list. This attribute is the attribute of the type "General", value is text.
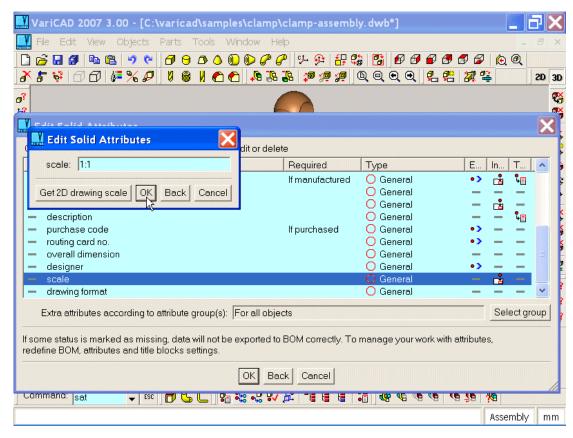
 Definition of attribute "weight"

The attribute "weight" was configured as of the type "Mass". During its definition, you can use volume and mass calculation from 3D module. Select the result from the list of results calculated for various units (like kg, lb ...)
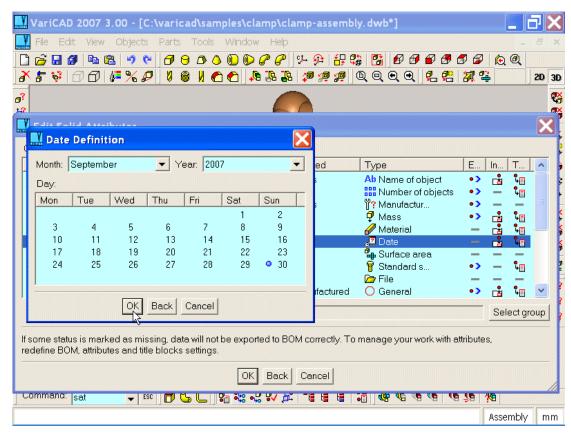
Definition of the attribute "material"

You can copy value of the attribute "material" from another solid (as in this example) or you can select from previously used materials.

Definition of the attribute "scale" (scale of 2D area)
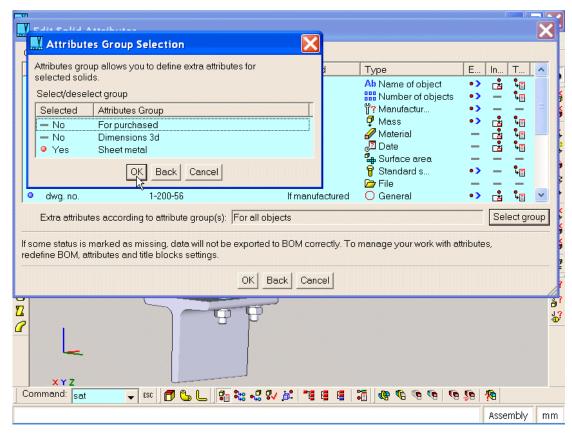
The attribute "scale" was configured to obtain value from the corresponding file - from its 2D part drawing scale.

Definition of the attribute "date"
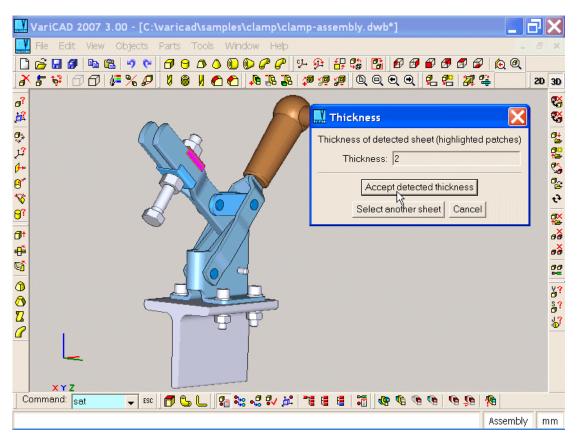
Extra Solid Attributes from Attribute Groups

In this example, the attribute group "Sheet metal" is assigned to the solid the attributes are defined for. Then the extra attribute "thickness" is required.

Definition of the attribute "thickness"

The attribute thickness was configured to allow sheet metal thickness measurement.
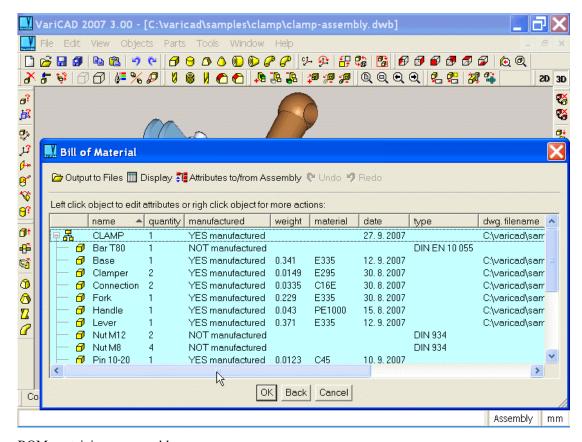
Sheet metal thickness measurement
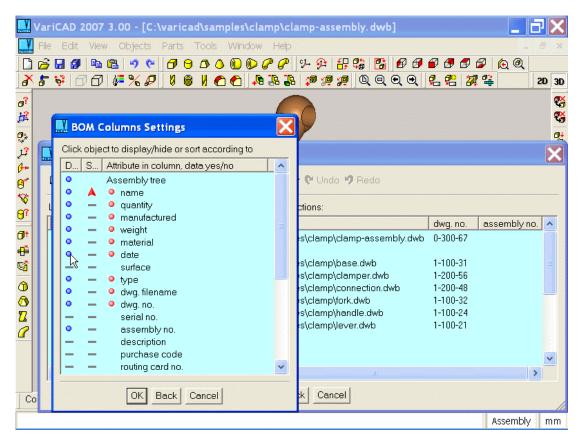
Working with BOM

Displaying BOM Objects, Filling Title Blocks

BOM contains a legible list of assembly parts, their names and attributes. BOM allows you to edit solid attributes or export data to various types of files.
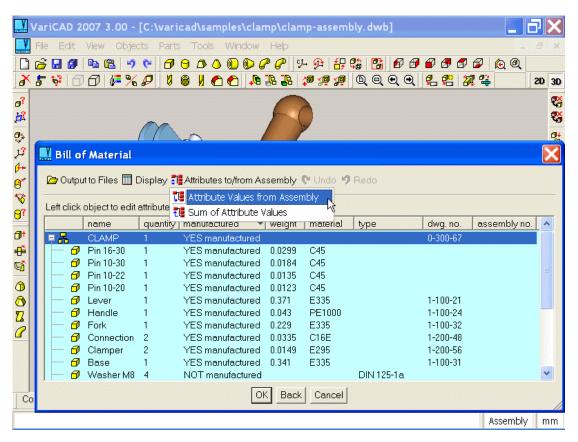
BOM containing an assembly
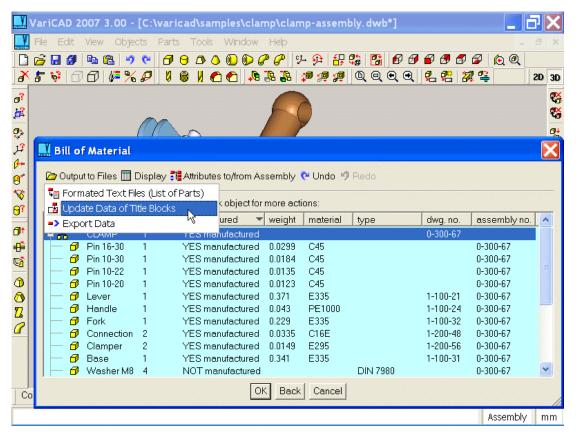
Display of columns settings

This function can be invoked by right click at column headers or from the pull down menu. You can also left click the column header. Then the list will be sorted according to attributes in this column.
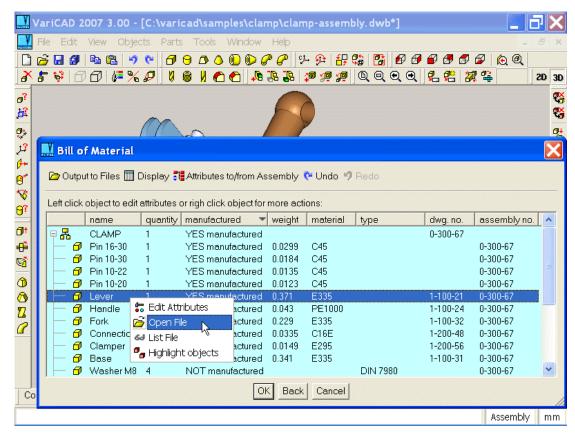
Copy of assembly attributes "dwg. no." to part attributes "assembly number"
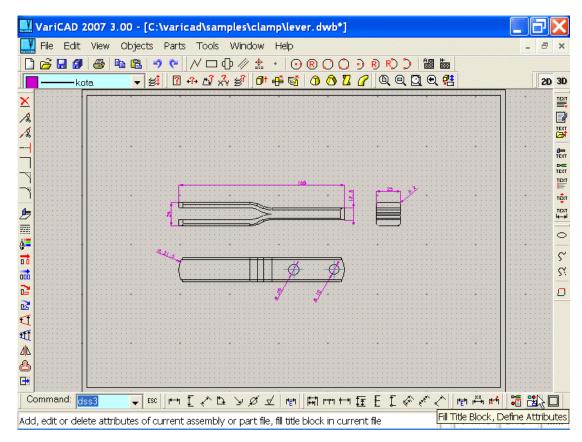
 Updating data of title blocks
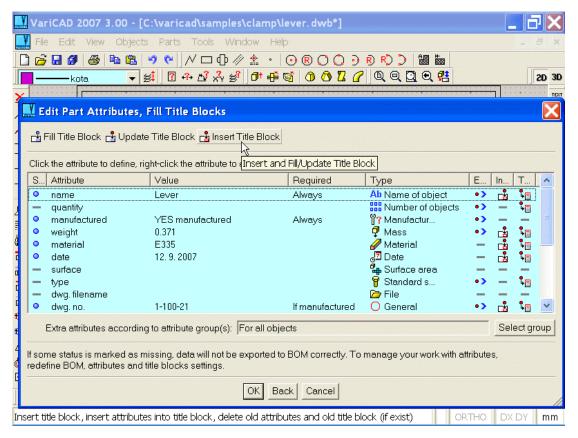
All attributes are copied into corresponding part files. Then the files are ready for filling title blocks. If the part files belong to part-assembly link, the data are updated always automatically. Attributes "assembly number" are updated as a result from the previous step.

Open the corresponding file containing a part
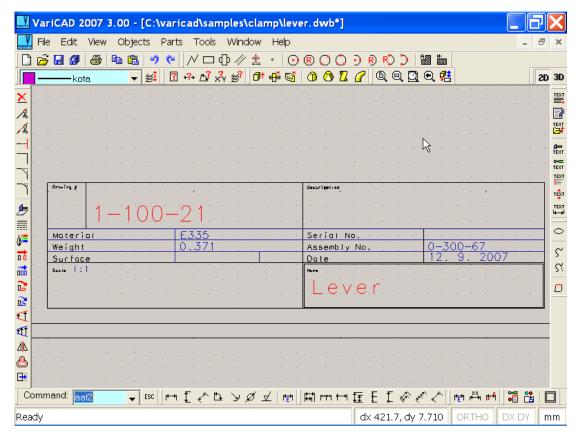
In 2D, select the function "Fill Title Blocks"

Automatic title block insertion and filling
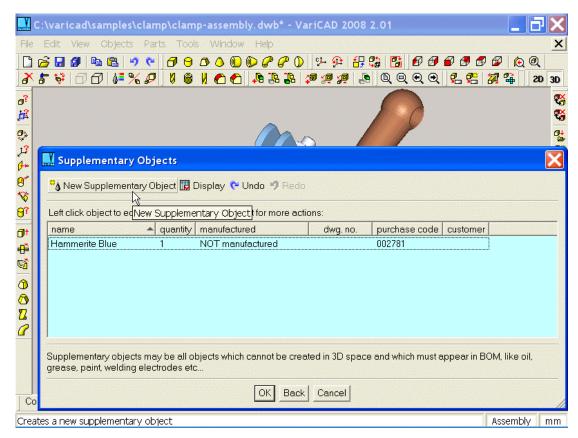
In this example, the title block is automatically inserted and automatically filled. If the title block was inserted and filled previously, the old data are removed first.

Title block automatically filled

Working with Supplementary Objects
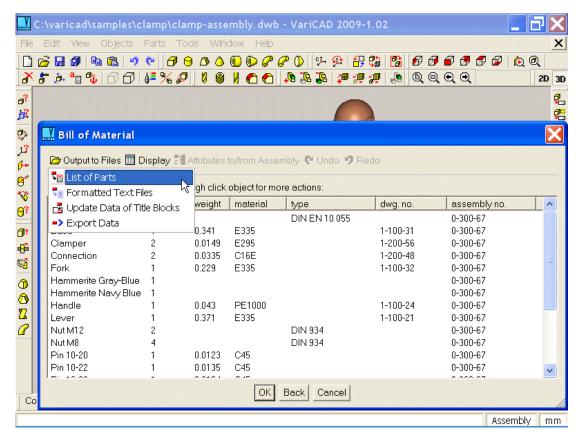
You can create a list of supplementary objects like paint, oil, grease, welding electrodes or other similar items. Such objects cannot be created in 3D modeler, but they must appear in BOM.
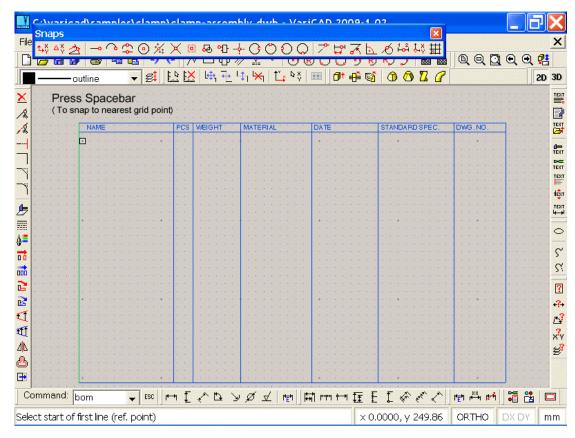
Creating a supplementary object

Creating List of Parts
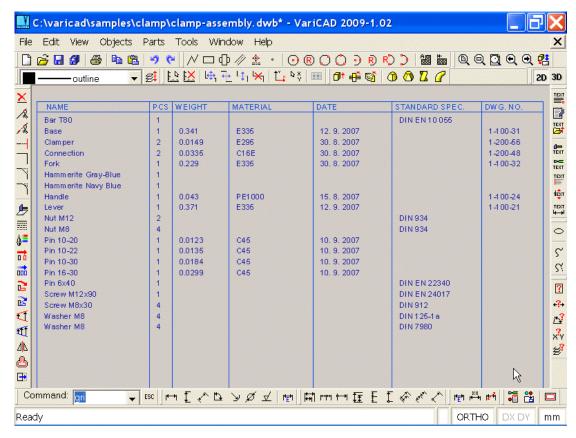
In this example, a list of assembly parts is created from BOM directly. You have to define only the insertion point of the first line of the text table, the position of the last line and the table lines are inserted automatically. If the list is longer than the defined space (the vertical difference between the two points), you can continue to insert the remaining part of the table at a different location. Configuration of table lines (records) allows you to change text attributes and text locations for each object separately. For instance, a record may contain multiple lines and variety of text styles.

BOM, selecting output to a formatted text table
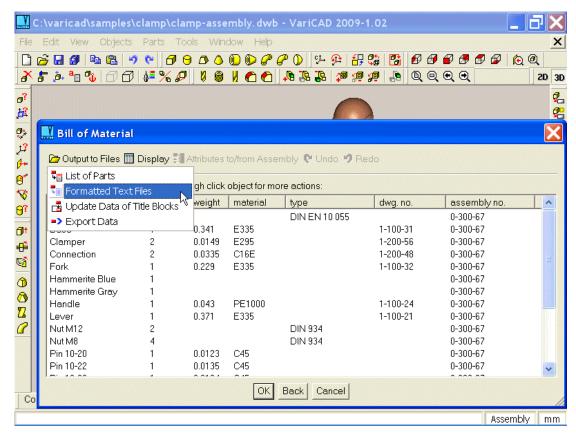
Defining text insertion into the prepared table
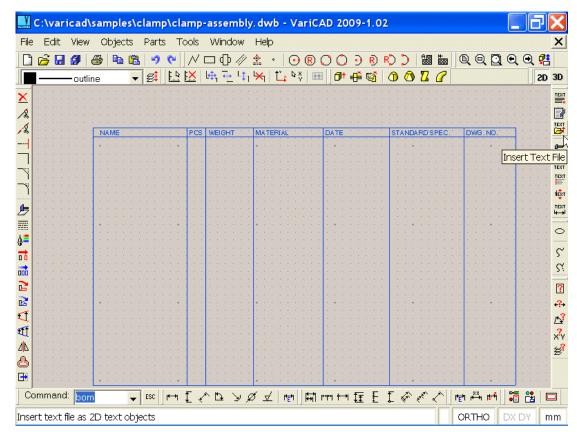
Filled table

In this example, a list of assembly parts is created via a text file. BOM saves the list into a formatted text file. The text file is inserted into 2D area. The 2D drawing contains a prepared table. Width of columns and text attributes of text objects correspond to width of columns of solid attributes, see *Definition of output to formatted text files (page 386)*. The text file is available also for other programs. On the other side, this method works correctly only for fonts with fixed width and configuration is more difficult than in the previous example.
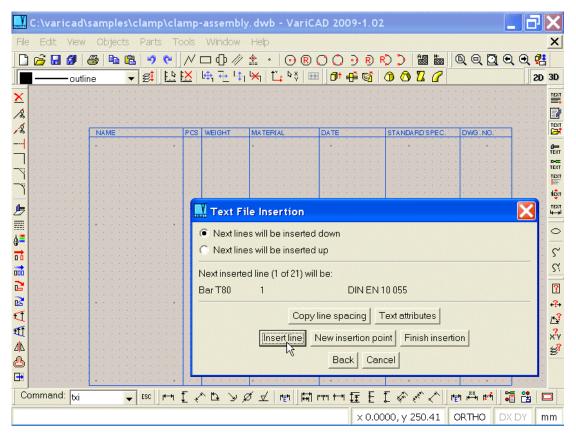
BOM, selecting output to a formatted text file

Select a file name, the list of parts is saved.
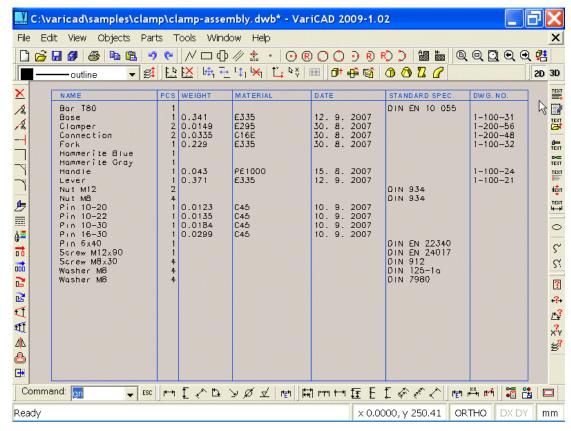
Insert the text file into 2D area

The drawing contains the prepared table. Select a location of the first line properly. In this example, the lower left corner of the first inserted line is at the highlighted grid point.
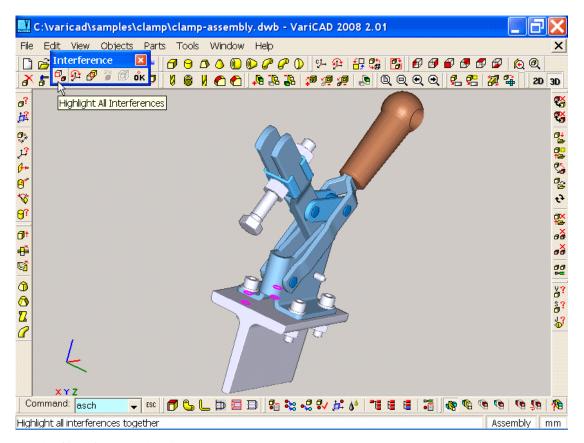
Inserting each line of the text file
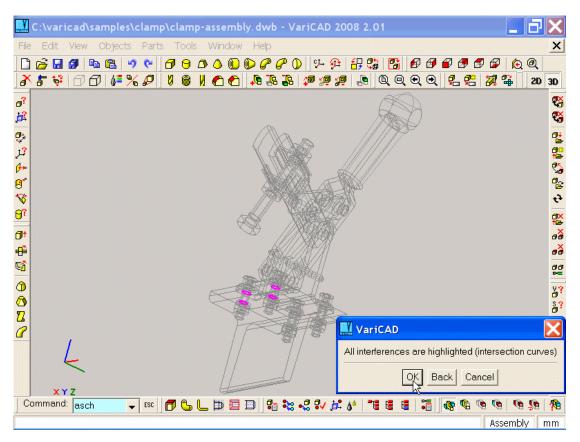
Filled table

Crash Tests (Interferences)

Checking interferences reveal colliding objects, more identical objects at a same location or smaller objects engulfed by larger ones. For more information, see *Interference between Solids (page 142)*.

Result of interferences checking

Highlighted interferences
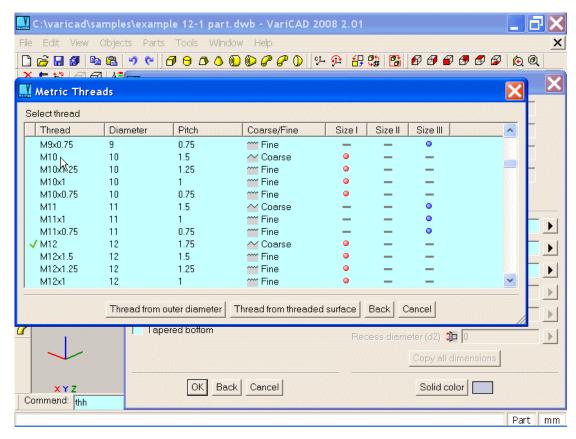
After checking, you can zoom each situation and change colliding solids' shapes or positions. In this example, the two screws were inserted and no corresponding holes were created into the T-bar.

3D Threads, 3D Sections, 2D Drawing in 3D
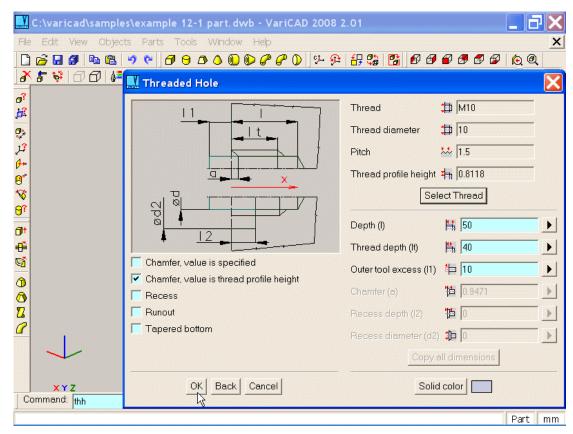
Creating 3D Threads

You can create threaded holes, insert threaded cylinders (used often as threaded ends of shafts) or create a thread at an outer cylindrical surface. If the current units are millimeters, you can select threads according to ISO (metric threads). If the units are inches, the threads are selected from list of Unified National Screw Threads (according to ANSI). For more information, see *Threads (page 139)*.
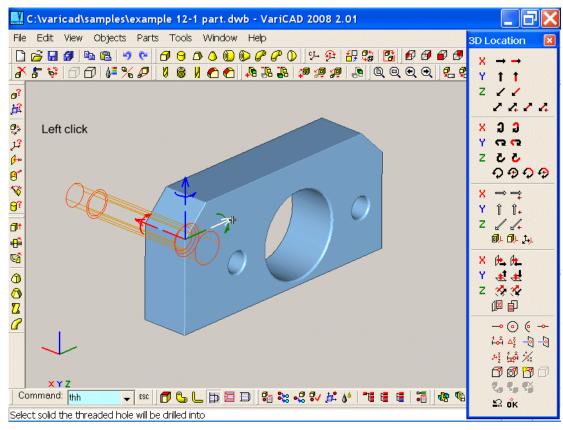
This section of the tutorial shows you creation of a thread in the part used in the assembly example.

Selecting a thread
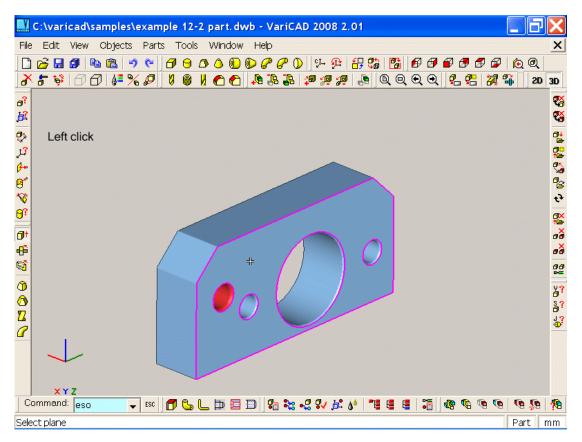
Definition of a threaded hole
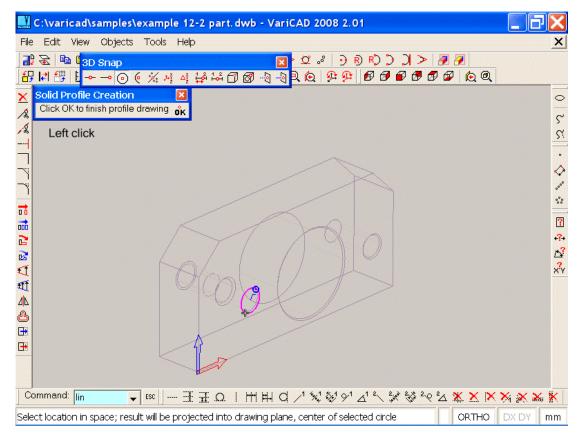
Insertion of a threaded hole

2D Drawing in 3D

In 3D, you can draw a profile further used for extrusion or rotation to create a new solid. Such a solid is pre-inserted into 3D space according to profile's location. This method can be, for instance, used for creation of a solid used as a section tool (like in this example). For more information, see *2D Drawing in 3D (page 90)*.
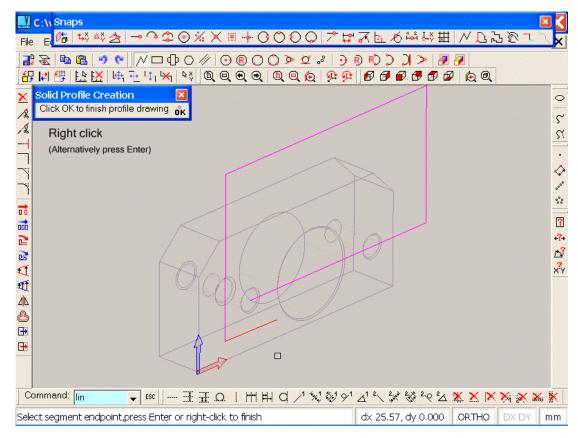
Selecting a drawing plane
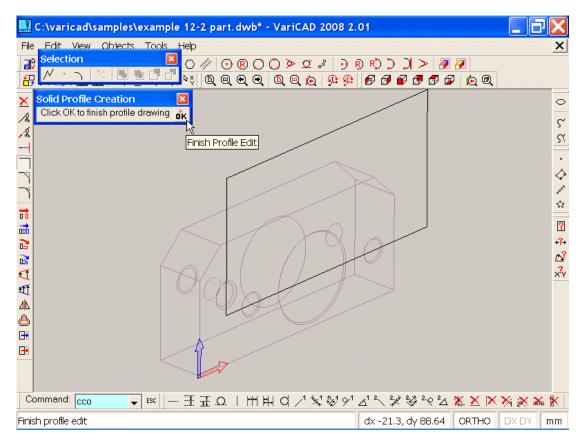
Location projected from 3D into the drawing plane
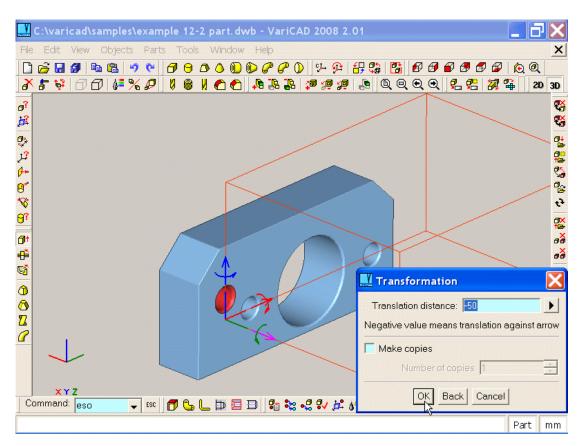
2D was switched to "Ortho" mode. Outline of the section tool is created right according to an existing 3D solid. To define profile's location, use the projection of 3D points – here is the projection of a circular edge center (intersection of axis of the hole and the drawing plane, more exactly). The profile is created as a multi-line and trimmed with the function creating a corner.

Drawing a profile
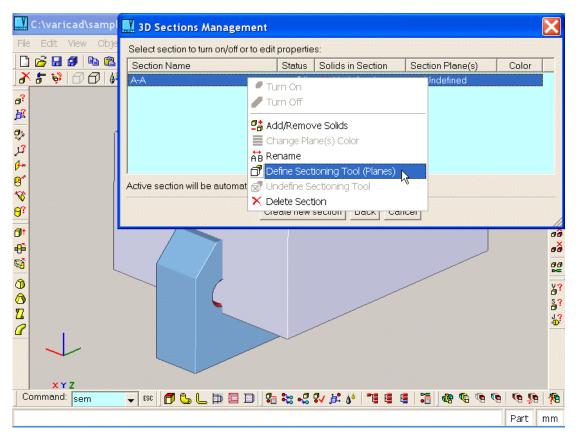
Finishing a profile
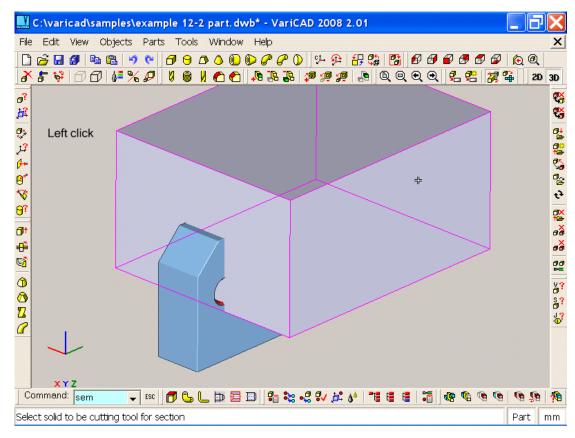
Inserting a new solid extruded from 2D profile
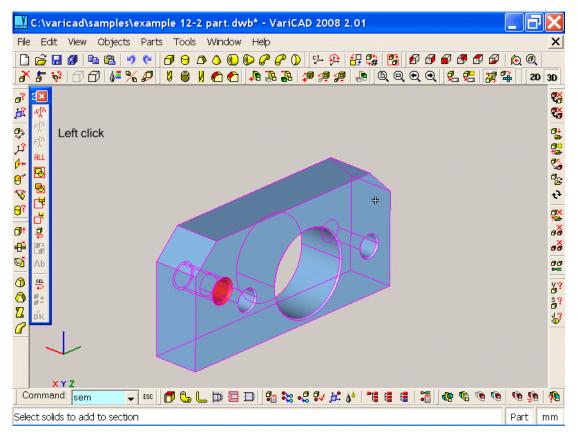
Creating 3D Sections
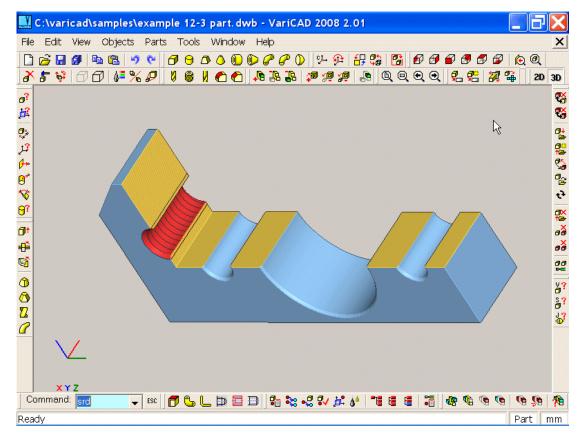
Section tool definition

Selecting a section tool

Selecting solids to be sectioned
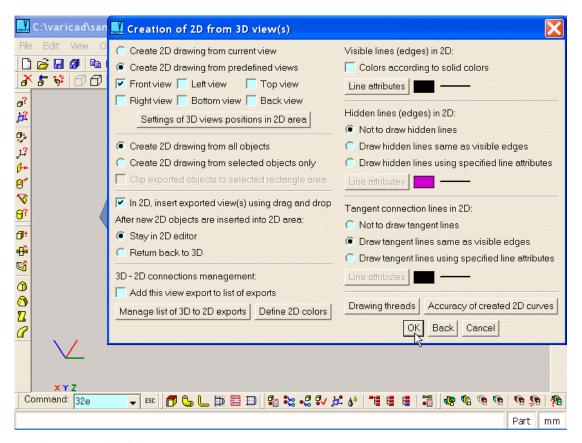
 Sectioned solid

The thread was redrawn with the function "precise rendering". Optionally, you can turn on drawing of the thread helix at threaded surfaces during the precise rendering.

2D Drawing from 3D Containing Sections and Threads
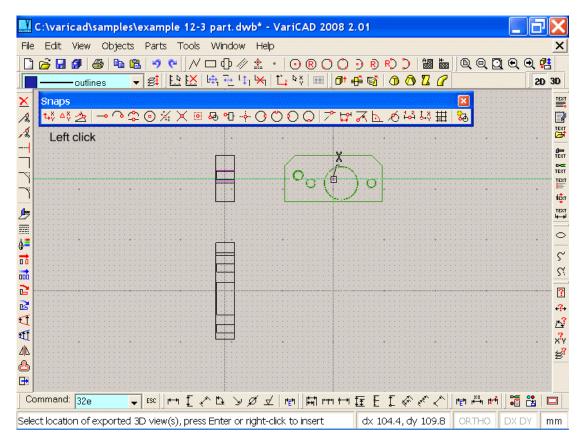
If a 2D drawing is created from 3D views with threads, the threaded surfaces are redrawn according to the current configuration. The configuration is available in the 3D export dialogue window. This example shows 2D creation of all outer or inner threads' diameters according to ISO standards.

Active 3D sections are exported into 2D as cut solids. If the particular export is inserted into a list of exports and if the 2D creation is repeated, the corresponding sections are turned on or off the same way as they were switched during the first export creation.
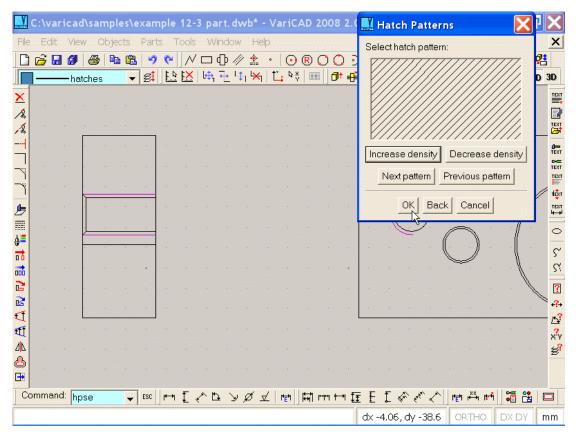
3D view export definition
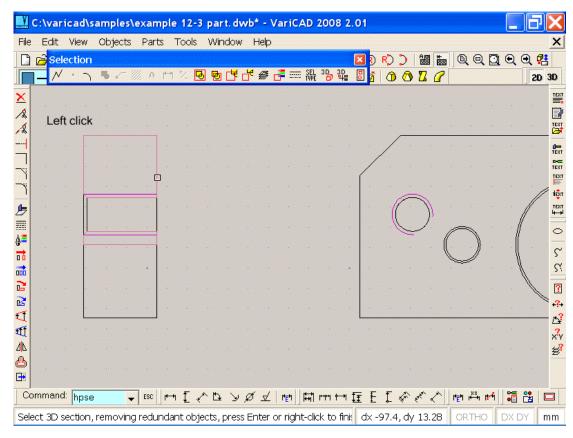
Front, top and left views were exported separately. Before the left view export, the section was switched off. The exported views are positioned at intersections of construction lines in 2D drawing area.
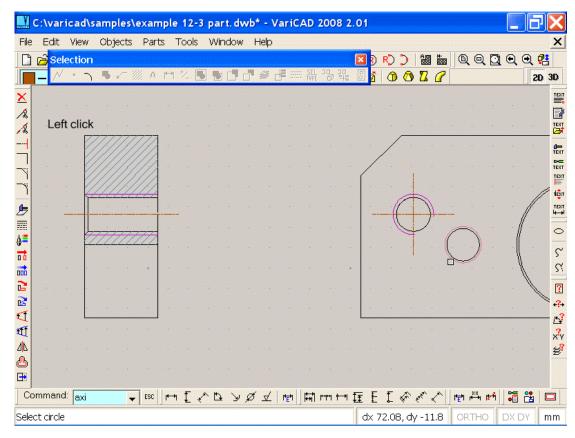
Positioning the created view

Hatching the sections

Section outline detected at once

Creation of axes

Index