Manual

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BeckerCAD 12

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1 Functional Range

1.1 Manual / Online Help

The manual contains notes for the installation and general commands, necessary to handle the program.

The complete documentation of all 2D and 3D commands of the different performance levels

BeckerCAD 2D

BeckerCAD 3D

BeckerCAD 3D pro

can be found in the Online Help. The Online Help is started using the menu command ***?, Contents***.
The Online Help in the program has the following advantages,

- immediate access to the help assigned to a specific command by pressing the ***F1*** key, while the cursor is placed on the icon.
- direct access to chapters, indices or key words.
- to store user relevant pages as favorites.
- to use cross references between chapters.
- that the description of some of the commands is supported by videos.

The complete printable manual can be found on the *BeckerCAD* installation CD in the folder Doku.

To get a first impression about the ease of use of ***BeckerCAD*** some example projects are part of the Online Help and manual as well.

1.2 Marking of the Individual Programs

The programs referred to below contain more commands than previous ones.
The assignment of commands for the individual programs is indicated as follows in the Online Help and the PDF file:

BeckerCAD 2D

As the commands for this program are also contained within the other two programs, they are not indicated here.

BeckerCAD 3D

The additional commands, which are available in this program are indicated below the title of the respective chapter by

from ***BeckerCAD 3D***

BeckerCAD 3D pro

The additional commands, which are available in this program are indicated below the title of the respective chapter by

from ***BeckerCAD 3D pro***

2 Installation

The installation of *BeckerCAD* is done with the program „SETUP.EXE“, which can be found in the root directory of the installation CD.

2.1 Hardware and software requirements

- At least 1 GB RAM (= working storage) or more.
- from AMD AM3 Phenom II, Intel Core i3
- CD/DVD-ROM
- Operating system Windows8 / 8.1® or Windows 10®.
- Graphics card with at least 1024 x 768 resolution, 32bit colours; 3D or Open GL graphics accelerator for CAD applications such as e.g.: graphics card with *nvidia* graphics processor.
- Monitor from 19’’ onwards
- Three button mouse or another control device accepted by Windows
- USB interface, alternatively for USB software protection adapter and for connection to plotter, printer, mouse and keyboard.

Connection of plotter

BeckerCAD is using the standard Windows plotters and printers. The installation of the printer/plotter is carried out exclusively with a suitable Windows printer driver from the respective manufacturer of your device or using the Windows own hardware recognition.

2.2 Program installation

Installing the current version in a folder of an earlier version

When installing a new version into the directory of a previous version it is mandatory, to deinstall the previous version first.

When this is to be done, you first have to make backup copies of the files listed here, that are supplied as part of the software for the new version and may have been edited already:

- STANDARD.TPL, STANDARD.SAT (standard template)
  This is found in the folder ...\USER\TPL

- Erase the files containing general defaults for the user interface: UIBECKER.SYS
  These are in ...\USER or in the directory which you specified as the working directory, when you connected the program.

Administrator access rights

The installation should be carried out by the administrator or with administrator rights for the workstation. The user should be given administrator rights for installation. After the installation is completed, the rights can be reduced again.
Additionally, is should be ensured that user profiles without administrator rights should have access to all installed components e.g. ODBC32, ODBC16.

Installation

In order to install *BeckerCAD*, firstly start your computer with the operating system Windows Windows 8® / 8.1® or Windows 10®.

1. Start the installation program „SETUP.EXE“ from the CD-ROM and follow the instructions.

2. Confirm the predefined destination folder for the program directory by clicking the ***Next*** button or define a new folder using the ***Browse*** button. The predefined destination folder is added as default to a selected directory. If the destination folder doesn´t exist, please confirm to create the folder.

   The target directory should not be controlled below a Windows folder. We recommend to install the program in the root directory of a partition, as suggested in the dialog.

2.3 Deinstalling the program

The deinstallation program can be used to remove all the program constituents installed for *BeckerCAD*.

We recommend that the previous version of this program is deinstalled before installing the new version in the same folder!

If necessary make a backup copy of the files listed in the previous chapter.

1. Click on the **Start** menu and select *Programs*.

2. Select the *BeckerCAD* designation for the program group specified at installation.

3. Select *Deinstall BeckerCAD.*
4. Then follow the commands which are displayed on screen.

   Once the Windows-deinstallation program displays a message that specific components are no longer required (e.g. files with the extension DLL), a prompt appears whether these should be erased. Answer the prompt **for all these components with No**.

The deinstallation program removes all folders and files apart from those which are set up by the user or by *BeckerCAD* after installation.

2.4 BeckerCAD Python

The BeckerCAD Python programming interface enables you to load and execute programs in BeckerCAD you have programmed in Python.

The user defined programs can be started from the menu command ***Extras, Python Script***.

3 Program control

3.1 Hardware requirements

The hardware requirements for *BeckerCAD* can be found in the attached manual or please ask your respective dealer.

3.2 Start and quit program

Starting the program

Once the installation program is complete and the computer has been restarted, the programs you have installed, can be started in the following ways:

1. Open the Windows Start menu.
2. Select ***Programs***
3. Select the designation of the program group you specified at installation for *BeckerCAD.*
4. Click on the program command button.
5. When you start up *BeckerCAD*, the start-up assistant is opened.

   By clicking on the corresponding command button, you can select here whether you wish to generate a new 2D drawing or a new 3D model and then you can define the necessary settings to do this.

   If you want to open an existing file and process it, click on this command button.

   Using an additional command button, you can call up the online help, open the example project or exit the assistant.
6. The first time you start up the program, you will see the 3D window for processing a **3D Model**.
   If you do not wish to generate or process a 3D model, but rather a **2D drawing,** switch to the 2D drawing window using i.e. the key combination *Ctrl Tab*.

   Please note:

   > If you see the 2D drawing window when you start up *BeckerCAD,* open the STANDARD.TPL template file, use for example *Ctrl Tab* to switch to the 2D drawing window and save the file.

You will find additional information on this and other settings in the chapter on ***Defining, program settings***.

7. If you wish to generate a new 3D Model or a new 2D drawing right at the start, define the necessary settings using the ***Define*** and ***Settings*** menu commands.

   You do not need to do this if you have already defined template files (*.TPL) containing the required settings.
   In this case, you can use the settings stored in it, by calling up the menu command ***File, New model*** and then by selecting one of these template files.

   You will find additional information on how to do this in the chapter ***Defining program settings, Defining model specific settings***.

Quitting the program

There are a number of options available for quitting a work session of *BeckerCAD*:

- Using the menu command ***File, Exit***
-  By clicking this system command button in the main *BeckerCAD* window.
-  By clicking this system command button in the main *BeckerCAD* window and then selecting the menu command ***Close***.

If objects have been previously generated, a prompt appears stating whether the model data or template data should be saved under the name which is displayed in the top line of the main window.

Yes
The model or template is to be saved under the displayed name.

No
The model data or template data is to be lost.

Abort
It is possible to return to processing the model or template in order to work on them further or, if necessary, to save the data under a different name using the ***File, Save As*** menu command.

Please note:

Exit the program without first closing the individual graphics windows (2D drawing, 3D model). The status of the closed window is saved with the model / template.

Add On Programs

Optional or user defined add on programs, using the COM interface and entered in the registry, can now comfortably be administrated, started and closed using the menu ***Extras, Additional Applications***.

For every Add On program an auto start can be activated, to start the add on automatically with the *BeckerCAD* start.

3.3 Screen and general settings

In the following sections of this chapter you will find a description of the various components making up the *BeckerCAD* screen that are available as soon as you have started the program for the very first time.

You can alter any of the internal program settings that you already find in the screens and then save them for your own use.
Further guidance is to be found in the chapter entitled ***General commands, Program control, Determine program settings***

Main window

The main *BeckerCAD* window contains the following components:

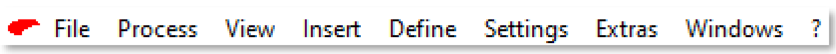
Header

As soon as you have started the program for the very first time this line contains MODELL.MOD as the name of the model, a description of the program, the entry "3D Model" and, on the extreme right and extreme left; the Windows system buttons.

When you are working on a different file (a model or a template), the entry MODELL.MOD will be replaced by the appropriate name.

When you change from the 3D view window into the 2D drawing window, which can be done by pressing the Ctrl and Tab keys at the same time, the entry in the screen header will change from "3D Model" to the name of the drawing in current use, its scale and its dimensions.

Menu bar

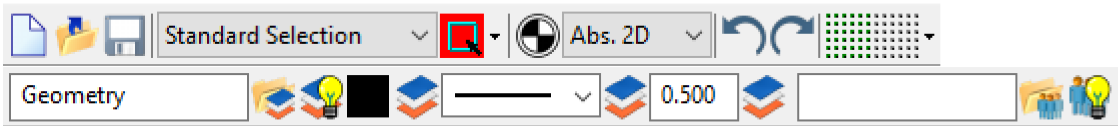


The menu bar contains headings of the pull-down menus, i.e. menus which are "pulled down" by clicking on the heading, therefore opening the menus.

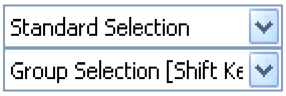
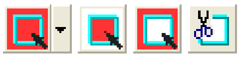
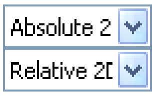
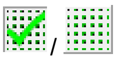
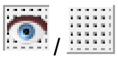
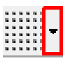
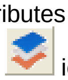
These menus contain lines which are used to load menu commands or to branch into further menus.

Most of the menu commands in the pull-down menus are always available. Some however, can only be used in the 2D drawing window or 3D window.
Menu commands which cannot be selected are displayed in grey, not black.

Main toolbar



The main toolbar contains commands which are required frequently, which can be loaded both in the 2D drawing window and the 3D view window. The following are available from left to right:

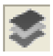
	File, New Model
	File, Open Model
	File, Save
	Standard selection using the Shift and Alt key Structured group selection using the Shift key
	Selection mode: All, Inside, Outside, Clip
	Define origin for local coordinate system
	Activate global/local coordinate system
	Undo last command
	Redo last status
	Activate / deactivate currently set drawing grid
	Draw currently set drawing grid yes / no
	**Grid selection menu** of the created grid with the menu command ***Settings, Drawing Grid***
	Display of the current layer for 2D and 3D objects. 2D objects created with the commands in the menu ***Draw 2D*** are placed on the displayed layer. If the display attributes are taken over from this layer, this is displayed by the  icon after the three following text boxes.

Opens the layer selection to define another layer as current layer for geometry.

Layer selection for 2D objects by identifying a 2D object belonging to this layer.

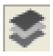
Colour selection for 2D objects which have been generated using the *Draw 2D* menu.

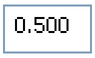
 The colour is taken over from the current drawing layer.

 The colour is allocated to the objects themselves.

Line type selection for 2D objects which have been generated using the *Draw 2D* menu.

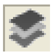
 The line type is taken over from the current drawing layer.

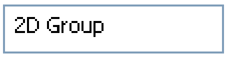
 The line type is allocated to the objects themselves



Line width definition for 2D objects which have been generated using the *Draw 2D* menu.

 The line width is taken over from the current drawing layer

 The line width is allocated to the objects themselves

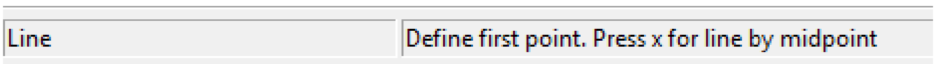


Group selection for 2D objects.
By using groups, the drawing can be structured logically.

Opens the group selection to select another group as active group.

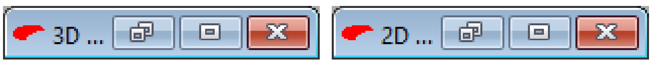
This is for group selection for 2D objects by identifying of the 2D objects in it.

Prompt line



In this line at the bottom of the main window, messages are displayed which show the command selected, e.g. drawing a line.
Next to this, is a message stating which action now needs to be carried out for the respective command, e.g. define the first point of a line if necessary; using the commands within the Snap Menu, which can be called up via the middle mouse button.

Window symbols



These will depict the windows that have been minimised to form symbols: 3D view window, 2D drawing window and the dialog box for the ***Define, Group menu*** command.

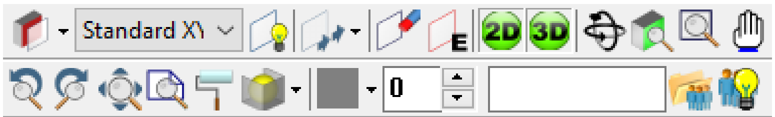
Using the system command buttons, each of these symbols can then be enlarged, maximised or closed.

3D view window

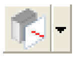
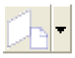
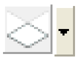
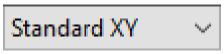
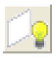
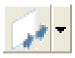
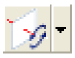
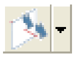
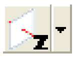
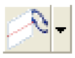
After you have started the program for the very first time, a 3D view window of the model space will appear in which you can create 3D objects (solids, 3D polygons) and 2D objects as well to help with your work in 3D.

The 3D view window contains the following sections:

3D top toolbar



The 3D top toolbar contains commands, which can be used to select and edit workplanes (WPs), to determine the view on the model space and to preset the solid display. From left to right, these are:

	Define WP using face/centre of gravity
	Define WP using face/edge/axis
	Define WP
	Define WP or WP set
	Restore standard workplanes
	Select WP from list
	Select WP by identifying frame
	Move WP in itself
	Rotate WP in itself
	Move WP along the Z axis
	Invert WP
	Rotate WP about any axis
	Erase WP
	WP settings
	Activate/deactivate 2D mode
	Activate/deactivate 3D mode

Rotate dynamically using middle mouse button, also zoom dynamically, set viewpoint

View on face/WP

Enlarge window

Vary View

Display previous view

Display next view

Display all 3D objects

Display entire WP (original) in top view

Redraw

/ Hide / Unhide Solids

/ Hide / Unhide all other Solids

Colour selection for solids generated next

Material Selection for newly defined Solids

0 Define transparency for solids generated next

3D Group Current 3D group

Opens the group selection to define the current 3D group.

Activate 3D group

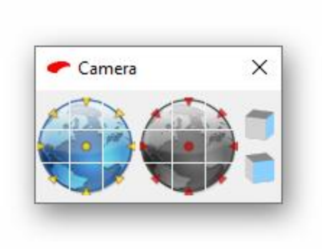
3D toolbar

This toolbar can be used to load 3D CAD Menu commands in the top area of the left column and 2D CAD Menu commands in the lower area.

They only refer to objects within the respective 3D view window.

These two commands for reselecting and erasing objects are independent of the selection of a special toolbox. They are always available.

Toolbar - Camera



The Camera toolbar contains commands for selecting the top views of the 3D model. If this toolbar does not appear, you can activate it by using the ***View***, ***Camera*** command.

2D drawing window

When you start the program for the very first time a 3D view window will appear. You can press the two keys Ctrl and Tab at the same time to change to the 2D drawing window.

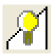
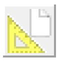
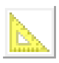
A 2D drawing window contains a view of a 2D drawing in which the 2D objects already generated are displayed.

If the 2D drawing window is not displayed full screen, the name of the current drawing, the scale of the current partial drawing as well as the dimensions of the drawing, will be displayed in the heading line.
If the window is displayed full screen, this will be shown in the heading line of the main window.

The 2D drawing window contains the following components:

2D top toolbar

The 2D toolbar contains commands for selecting the current partial drawing as well as modifying the view:

	Starts the print dialog window
	Show / hide status of partial drawing
	Activate partial drawing
	Activate partial drawing
	Access to current partial drawing
 / 	Access to defined partial drawing / list of partial drawings
	Access to all partial drawings
	Display enlarged window
	Vary view

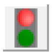
 Display previous view

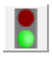
 Display next view

 Display all 2D objects

 Display entire drawing (original) in top view

 Redraw

 Update model view

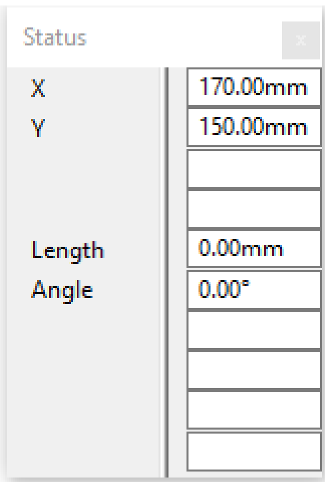
 Model view is current

2D toolbar

This toolbar can be used to load 2D CAD Menu commands. They only refer to objects within the respective 2D drawing window.

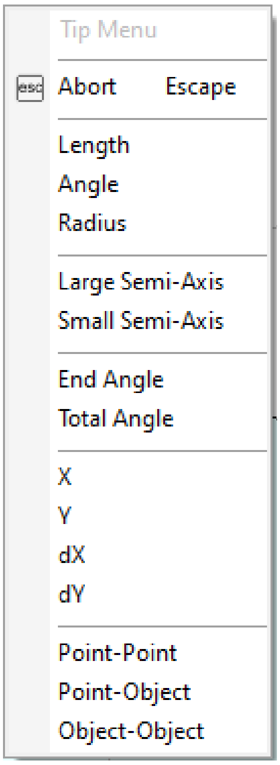
If working in a 3D view window, the 2D toolbar is part of the 3D toolbar.

The Status dialog box



The text boxes within the Status dialog box can be used when drawing 2D objects, when generating solids and processing these for display and entry of numerical values (point coordinates, angle, length, diameter, etc.).

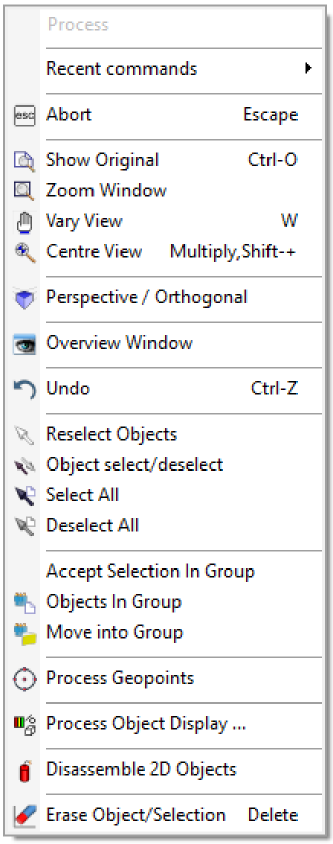
Pick menu



If the 2D drawing window is to contain data relating to an existing 2D line object so that a new object of the same kind can be defined, you can use the Pick menu.

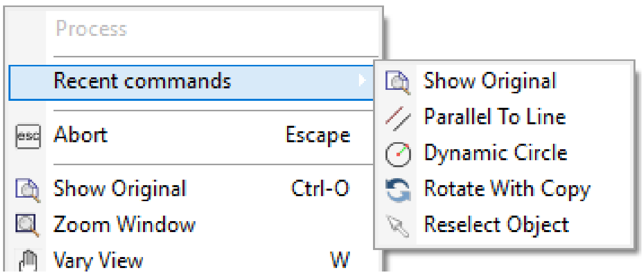
Another method is for you to drag the cursor with your mouse into one of the active input boxes in the Status dialog box and then to open it by clicking the middle mouse button.

Edit menu



If no command is active whilst the cursor is in the drawing area, you can use the middle mouse button to load the ***Edit*** menu which contains commands that are needed frequently, but are also available at other positions.

In this menu the last used commands from the current session are also offered.



User defined symbol bar

The symbol bar defined by the user will not be presented on screen the very first time the program is started.
This symbol bar is specified in detail with the help of the ***View, Toolbar*** menu command and displayed at one edge of the main window.
Guidance is to be found in the chapter entitled ***General commands, Apply user defined symbol bar***.

3.4 General commands

Except for those commands where you have to enter values or texts via the keyboard, BeckerCAD is completely controlled by the mouse.

Use the mouse to control the cursor  on the screen.

The way in which the commands of the individual program areas are started and ended is described in the following sections.

Some commands, such as opening the point definition menu, are triggered by the middle mouse button of a 3-button mouse.
If you only have a 2-button mouse, you can use the right mouse button by holding down the Ctrl key on the keyboard at the same time.

Display help

Information concerning terms, commands and interconnections in *BeckerCAD* as well as the current program versions which contain modifications, can be displayed using the ***?, Contents*** menu command.

If the cursor is on a line of a pull-down menu by which a menu command can be loaded or it is on an icon in the toolbar, press the function key **F1** in order to load the respective help text.

Menu bar, toolbar

By moving the cursor to the menu bar in the main window or to an icon in a toolbar or to a menu heading, the command "lying behind" can be started by pressing the left mouse button (= click).

The commands contained in the menu bar contain underlined characters e.g. *__P__rocess, Objec__t__ Display*.
Accordingly, the ***Process*** menu can be opened using the key combination **Alt P**, or the key combination **Alt P T** can be used to open the command ***Process, Object Display*** directly.

Symbol toolbar

After clicking on an icon in the left column, as for a submenu, the icons of the respective commands are displayed in the right column (exception: *Erase Object/Selection,* which can be loaded directly in the left column).

By clicking on the icon in the right column, the respective CAD command is started.

They can be quit using the right mouse button.

Working with the Status dialog box

The Status dialog box is for displaying and for entering numbers such as point coordinates, lengths, sizes of angles.

When a number is entered you can use the following methods:

- Enter the exact numeric value
- Transfer a numeric value from 2D line objects that have already been drawn.
- Enter a formula.

<u>Activate Status dialog box and select line</u>

For the input of numeric values or formulae, the cursor must be in the Status dialog box. There are two options of doing so:

- **Using the mouse**
  Using the mouse, move the cursor into the required text box.
  By simply clicking on the text box, it then becomes the input text box.
  By double-clicking, the text box is activated. At the same time, the existing entry is highlighted and can be written over directly.
- **Using the keyboard**
  Press the spacebar. This results in the change from the drawing area to the last active text box in the Status dialog box and vice versa.
  The required text box within the Status dialog box can be reached

using the cursor control keys ↑, ↓ or using **Tab**, **Ctrl-Tab**.
The existing entry is highlighted and can be written over directly.

Please note:

If the status window does not appear after you have pressed the space bar or selected the menu command ***View***, ***Status Dialog Box*** ..., click on the ***Arrange all dialogs Windows*** option in the ***Windows*** menu.

The Status dialog box can be docked on to the program window to the left, right, top and bottom. In order to do so, move the dialog box over the border of the program window, until it visually changes its form and then let it fall. The docked on Status dialog box can be released again by clicking on the strip and keeping the mouse button pressed to loosen the dialog box from the program window.
Minimise the Status dialog box using the icon, close it using the icon. Once you have minimised the Status dialog box in the locked on state, you can open it again by double-clicking next to the strip in the program window frame.

If you wish to move the Status dialog box on the border of the program window without it being docked onto it, keep the Ctrl key pressed before releasing the Status dialog box on the required position.

The size of the Status dialog box can be modified by moving the cursor to the edge of the Status dialog box until a black double arrow appears. At this moment, press the mouse key and drag the Status dialog box to the required size.
The vertical division between the names in the Status dialog box and the entry text boxes can be modified by moving the middle vertical line.

Setting options for the Status dialog box, e.g. the number of decimal places for the display, as well as the background colour for text boxes that must be edited or even if you wish to define a transparency degree for the Status dialog box, can be found in the menu bar under ***Settings, Options*** on the ***User interface*** option card.

Entries in Status dialog box

The Status dialog box fields can be used to enter numerical values or formulae. In the case of real numbers you can use either a comma or a point as the decimal separator.
If the text box contains an entry as a whole number and decimals, the entry will be rounded to the closest lower whole number.
Each entry must be confirmed with Enter ↵.

Accept numeric values from existing 2D line objects

If the model being developed is a circle that has already been drawn, that is a line forming the circumference, you can use the following method:

1. Position the cursor inside the text box of the Status dialog box that is to contain the new numeric entry, e.g. the ***Length*** text box.
2. Open the ***Pick*** context menu by clicking the middle mouse button.
3. Select the line in the menu so that you can accept the numeric value, e.g. ***Length***.
4. Identify the 2D line object according to which of them has the numeric value that you are going to transfer: in this case it is the circle.

Alternatively, you can use this method:

1. Position the cursor inside the text box of the Status dialog box that is to contain the new numeric entry, e.g. the ***Length*** text box.
2. Type the abbreviation for the numeric value that you are going to transfer, e.g. the abbreviation for length is ***tl***.
   Instead you can type the abbreviation used as part of a formula (see below), e.g. *tl /2*.
3. Identify the 2D line object according to which of them has the numeric value that you are going to transfer: in this case it is the circle.

   Please note:

   Instead of identifying a 2D line object you can click the middle mouse button to open the Pick menu so that you can transfer a value other than a length.

The Pick menu can also be used in conjunction with other commands, e.g. for extending a line on an absolute value. More information regarding this can be found in chapter: Extend/shorten lines to absolute value.

The Pick menu contains the commands listed below so that you can transfer numeric values from 2D line objects:

Command	Abbrev.	Meaning
Cancel		This closes the Pick menu before the transfer of a numeric value.
Length	tl	Length of 2D line objects
Angle	tw	Angle formed by a line away from a picked endpoint

Radius	tr	Radius of a circle
Large/small semi-axis	tra, trb	Large/small semi-axis of an ellipse
End angle	twe	Angle formed by a circle or elliptical arc at a picked point
Total angle	twg	Total angle formed by a circle or elliptical arc
X, Y	tx, ty	The X or Y coordinate of the picked endpoint of a 2D line object
dX, dY	tdx, tdy	X or Y distance of both endpoints of a 2D line object
Point-Point	tapp	Distance between two endpoints on a 2D line object
Point-object	tapo	Smaller distance between a point and an endpoint on a 2D line object
Object-object	taoo	Smallest distance between two 2D line objects

Entering a formula

When you are entering a formula you can use the following operators or commands in addition to numbers and abbreviations to transfer numerical values:

+	Addition
-	Subtraction
*	Multiplication
/	Division
x!	Factorial of x
abs(x)	Absolute value of x
round(x)	Rounding of x to the next whole number
sig(x)	Biggest whole number <= x
sqr(x)	Square of x
sqrt(x)	Square root of x
exp(x)	e^x
ln(x)	Original logarithm of x
pi	Pi
	Angle commands (Radian measure)
sin(x)	Sinus of x Example of the determination of sin(30°): sin(pi*30/180)
cos(x)	Cosinus of x

tan(x)	Tangent of x
arcsin(x)	Arcussinus of x
arccos(x)	Arcuscosinus of x
arctan(x)	Arcustangens of x
sinh(x)	Sinushyperbolicus of x
cosh(x)	Cosinushyperbolicus of x
tanh(x)	Tangenshyperbolicus of x

Please note:

If the Status dialog box is closed, it can be opened again if, after loading a command which requires the input of a numeric entry, the spacebar is pressed or loading the menu command ***View, Status*** dialog box. Using the menu option ***Settings, Options*** there are further options for formatting the Status dialog box regarding the actual appearance of it as well as regarding the formatting of entry and display values.

Dialog box

Within **dialog boxes**, below the heading, various option cards can be activated - if they exist - by clicking on the respective card, or text boxes can be given new or modified values by simply clicking on the respective position in the dialog box.

If the Accept command button is clicked, value modifications are activated without the dialog box being closed.

By clicking on one of the command buttons OK or Close, the dialog box is closed and the values entered are then activated. If a dialog box only contains the ***Close*** command button, the values determined become valid immediately. Such a dialog box can be kept open or reduced in size to a symbol which can later be opened when required.

If the entries which have been made are not to be kept, click on the Cancel command button.

2D drawing window, 3D view window

In the 3D view window you can generate and work on 3D objects, in the 2D drawing window, you can generate and process drawings.

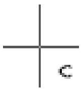
To do this you should select the corresponding window as the active one each time.

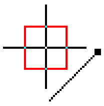
If these windows are displayed as full screens, you can use the **Ctrl Tab** combination to switch between windows.
You can also use the ***Window*** menu command to directly activate the window required.

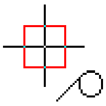
If the windows are not displayed as full screens, you can also click on any area within one of these windows to select it as the active window.

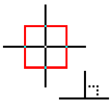
When **generating and processing objects** in one of these windows the cursor can have various tasks depending on the previously selected command and therefore receives various additional symbols:

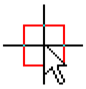
 Cursor to define an enlarged display

 Cross-hair cursor to define points

 Cursor to snap points on objects

 Cursor with tangential conditions

 Cursor with perpendicular conditions

 Cursor to snap objects

Keyboard commands

View	
Ctrl Tab	Change between 2D drawing window and 3D view window
F5	Redraw
+	View, zoom +
-	View, zoom -
*	Determine drawing centre for zooms +/-
W	Vary View
Ctrl-o, Ctrl-O	Show Original
F6	3D: camera, view from front left

F7	3D: camera, view from front right
F8	3D: camera, view from back left
F9	3D: camera, view from back right

Cursor key up ↑	The window slider moves up, the view is moved down.
Cursor key down ↓	The window slider moves down, the view is moved up.
Cursor key left ←	The window slider moves left, the view is moved right.
Cursor key right →	The window slider moves right, the view is moved left.

When pressing the ***Ctrl*** key at the same time the view is moved in bigger steps.

Page up ↑	The window slider moves up in big steps, the view is moved down.
Page down ↓	The window slider moves down in big steps, the view is moved up.

By additionally pressing the ***Strg-key***, the slider is moved in big steps to the left or the right.

(* To move the view with the keys, the slider must be visible. The movement steps are depending on the size of the slider)

Snap Menu	
Middle mouse button Ctrl right mouse button	Opens the Snap Menu, if a point is to be determined for an action.
H, h	Move cursor horizontally from last defined point
V, v	Move cursor vertically from last defined point
Ctrl+a, Ctrl+A	Autosnap for next point
E, e	Snap endpoint
M, m	Snap midpoint
Z, z	Snap centre
Q, q	Snap quadrant point
S, s	Snap intersection
Ctrl-Shift-S	Catch intersection point 2 lines
Ctrl-Shift-M	Catch midpoint of 2 points
Ctrl-Shift-P	Relative to the next point to be defined
Ctrl-Shift-R	Relative to last point
Ctrl-Shift-F	Relative to footpoint

P	Snap point
R, r	Snap reference point text
Ctrl-Shift-Y	Symbol reference point
N, n	Snap next point on line
G, g	Snap centre of gravity
t, T	Tangential for the next point
l, L	Perpendicular for the next point
C, c	Define cursor point
F, f	Activate snap mode
A,a	Activate generally auto snap mode
Ctrl-L, Ctrl-l	Activate generally perpendicular mode
Ctrl-T, Ctrl-t	Activate generally tangential mode
Strg-2	Origin 2D
Strg-3	Origin3D

Status dialog box	
Spacebar	Activates the drawing area or Status dialog box
↑, ↓	Line selection in Status dialog box
Enter, ↵	Confirm entered value in Status dialog box
	Direct selection from input text box
Ctrl-p; Ctrl-P	Switch between polar and cartesian coordinate entry in the 2D window
Ctrl X	X / DX
Ctrl Y	Y / DY

3D Selection Mode	With keyboard support
h	The solid the cursor is placed on is hided.
H (Shift+h)	The solid, on which the cirsor is placed on remains visible, all other solids in the current view are faded out.
u	The last hided solid ist unhided.
a	All previously hided solids are unhided.
d	The solid the cursor is placed on gets transparent. Points, lines and faces in the background can be identified.
f	The solid the cursor is placed on gets transparent. Points, lines and faces in the background can be identified. All other solid are hided.

F (Shift+f)	Deactivates / Activates the transparent display of the previously with f transparent defined solid.
y	Leave the transparency mode and fades all solids in.

General commands	
Del	Erase object/selection
Esc	Abort command
F1	Load online help
F2	Save model
Shift F2	Save model with the name SAVE.MOD
Ctrl F2	Save model as …
?	Information 2D/3D object
Ctrl+c Ctrl+C	Print to the clip board with the settings defined in the ***Print*** dialog window.
Ctrl+z; Ctrl +Z	One undo step is done for the last command.

Additional commands	
B, b	Switch between complete display and block display when positioning symbols dynamically
Shift (⇑)	If the key is kept pressed: - an active grid is temporarily deactivated - in tangential and perpendicular mode, the cursor can be moved over the ends of lines and arcs - when selecting/deselecting using the cursor, the group belonging to the identified 2D object can be determined - when positioning diameters or radius dimensions, to deactivate the 15° grid
Ctrl	- Used to collect objects when selecting/deselecting. The list of collected objects is confirmed using the left mouse button or rejected using the right mouse button. - With autopan, multiplied the scroll width by the scroll factor
Alt	- Used to select a group and the subgroups and objects in it during selection

X	- Within the framework of *symbol positioning* as well as within the framework of the *Transform , Dynamic Move/Dynamic Copy* command, the objects found on the cursor are mirrored on the X-axis - Makes it possible during dimensioning to switch between radius and diameter dimension. - toggle between the inner and outer angle, while the angle dimension is on the cursor.
Y	- Within the framework of *symbol positioning* as well as within the framework of the *Transform, Dynamic Move/Dynamic Copy* command, the objects found on the cursor are mirrored on the Y-axis.
Left arrow key ←	- Within the framework of *symbol positioning* as well as within the framework of the *Transform , Dynamic Move/Dynamic Copy* command, the objects found on the cursor are rotated in an anticlockwise direction by the angle defined in the Status dialog box.
Right arrow key →	- Within the framework of *symbol positioning* as well as within the framework of the *Transform , Dynamic Move/Dynamic Copy* command, the objects found on the cursor are rotated in a clockwise direction by the angle defined in the Status dialog box.
Ctrl+arrow key left ←	Makes it possible during the placement of symbols and during ***Transform***, ***Move Dynamically*** and ***Move with Copy Dynamically*** to rotate the grafic being on the cursor by 90° anti clockwise.
Ctrl+arrow key right →	Makes it possible during the placement of symbols and during ***Transform***, ***Move Dynamically*** and ***Move with Copy Dynamically*** to rotate the grafic being on the cursor by 90° clockwise.
Up arrow key ↑	- Within the framework of *symbol positioning* as well as within the framework of the *Transform , Dynamic Move/Dynamic Copy* command, the objects found on the cursor are scaled larger by the factor defined in the Status dialog box.
Down arrow key ↓	- Within the framework of *symbol positioning* as well as within the framework of the *Transform , Dynamic Move/Dynamic Copy* command, the objects found on the cursor are scaled smaller by the factor defined in the Status dialog box.
Ctrl+R	- Within the framework of *symbol positioning* as well as within the framework of the *Transform , Dynamic Move/Dynamic Copy* command, the objects found on the cursor are rotated dynamically about the cursor point .
Ctrl+S	- Within the framework of *symbol positioning* as well as within the framework of the *Transform , Dynamic Move/Dynamic Copy* command, the objects found on the cursor are scaled dynamically.

User defined key commands

The individual user defined key commands can be stored in a seperate file and with new versions only this file has to be copied in the new directory.

Create a text file and store it in the User directory. The file must be named **User_Hotkeys.txt**.

In this file enter the user defined key commands accordingly:

CTRL-s "op.gen.line"

CTRL-S "op.gen.line"

Example for the command Draw Line on the keys Strg+s and Strg+S.

Please note to write commands in apostrophe and to use **SHIFT-** and **CTRL-** for key combinations.
The internal command name can be found using the dialogue window ***View***, ***Toolbar***. The name will be displayed in the area ***Command*** if a command is selected in the command tree. (Apply user defined symbol bar and menus).

The user defined key commands are read after the internal key commands to insure that the user definitions have priority to the internal definitions.

After installation, the program directory */Bin* contains the file Hotkeys.txt, in which all application-independent keyboard commands are listed, and the file BeckerCAD.txt, from which the special keyboard commands are read.

Examples of key names (Virtual Key Codes) :

TAB	CLEAR	RETURN	PAUSE	CAPITAL
ESCAPE	SPACE	PAGE_UP	PAGE_DOWN	END
HOME	LEFT	UP	RIGHT	DOWN
SELECT	PRINT	EXECUTE	SNAPSHOT	INSERT
DELETE	SLEEP			
NUMPAD0	NUMPAD1	NUMPAD2	NUMPAD3	NUMPAD4
NUMPAD5	NUMPAD6	NUMPAD7	NUMPAD8	NUMPAD9
MULTIPLY	ADD	SEPARATOR	SUBTRACT	DECIMAL
DIVIDE				
F1 … F24				

Even more key combinations are available now, as SHIFT+M is no longer interpreted as capital 'M' , but really as SHIFT+M.

Please note:

The file **User_Hotkeys.txt** is not part of the delivery. It must be created by the user.

Keyboard combinations such as SHIFT + M are not evaluated as a capital letter 'M', but as a real key combination SHIFT + M.

Gesture Recognition

With the help of the gesture recognition any commands can be started by user defined mouse movements (gestures)

To execute a gesture no other command should be active. Press the left mouse button during the complete gesture. Move the mouse according to the route defined for a command as described below.

To connect a command to a specific gesture do the following:

1. Open the dialog using the menu ***Settings***, ***Gesture Recognition***.

2. In the structure tree on the left of the dialog all commands for the gesture recognition are listed. Select one of the commands by double click or drag&drop into the table.

3. In the column ***Gesture*** define the route by comma (,) seperate directions, the mouse should be moved to start the command. The following directions can be used:

N	North	Cursor up
NO	Northeast	Cursor diagonal right up
O	East	Cursor right
SO	Southeast	Cursor diagonal right down
S	South	Cursor down
SW	Southwest	Cursor diagonal left down
W	West	Cursor left
NW	Northwest	Cursor diagonal left up

Double click in the field or mark the field and confirm with ***F2***. Confirm the route with Enter.

Example: To start a command by one time up and down with the cursor, enter in the first field ***N,S*** (without blank).

4. Define in the column ***Context*** , in which grafic window the gesture should be recognized:

2D/3D = 2D und 3D window
2D = only 2D window
3D = only 3D window

5. In the field ***Parameter*** a path and a file name for an external program call can be entered for example. Usually this field is left empty.
6. Activate the option ***Gesture Recognition active*** in the dialog to use the above defined gestures.
7. With ***Sensitivity in pixels*** the minimum cursor movement on the screen is defined before recognizing a gesture.
8. If the option ***Display route*** is active, the direction will be shown in the command line.
9. An acoustic signal is given when recognizing a gesture if the option ***Acoustic support*** is active. The length and the frequency of the sound can be defined in the according fields.
10. If the option ***Remember last commands*** is active, the last commands and icon bars are stored and can be activated by special gesture types:

    Repeat last commands: S,O,W,O,W,....
    Example: Three commands were executed by icons.
    With the gestures
    - ***S,O*** the last command,
    - ***S,O,W*** the command before the last command,
    - and so on
    are repeated.

    Open last icon bars: N,O,W,O,W,...
    With the gestures
    - ***N,O*** the last toolbar is activated
    - ***N,O,W*** the toolbar before the last toolbar
    - and so on
    are repeated.

Only the commands started by icons and toolbars can be repeated by this special gestures.
In the field ***Number*** the number of commands and changes between toolbars can be defined.

Please note:

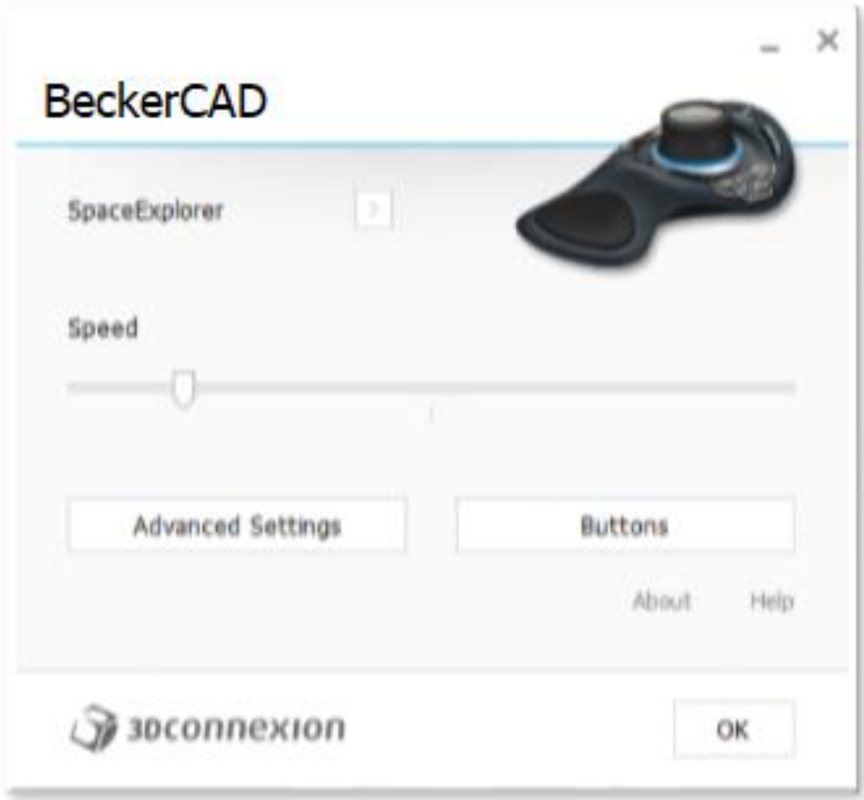
The defined gestures are stored in the file GESTURE.TXT in the currently defined TLW directory.

3D-Mouse: Setting up

3Dconnexion’s family of SpaceMouse products provide a comfortable and natural way to interact in *BeckerCAD*.

How to install the 3D Mouse:

- Make sure the 3D Mouse is connected to a USB port of your PC. If you have a wireless 3D Mouse, please connect the supplied USB cable during the initial installation process.
- After installing the 3Dconnexion software, you can access the 3Dconnexion Properties panel to configure your 3D Mouse. At the top of the panel you see the active application. ***Any changes you make to 3D mouse settings are relevant to this application only***.
  You can access the 3Dconnexion Properties panel via the icon in your notification area (systray), 3D Mouse Home or the "Menu" button on your 3D Mouse (by default this is the left button on 2-button devices).



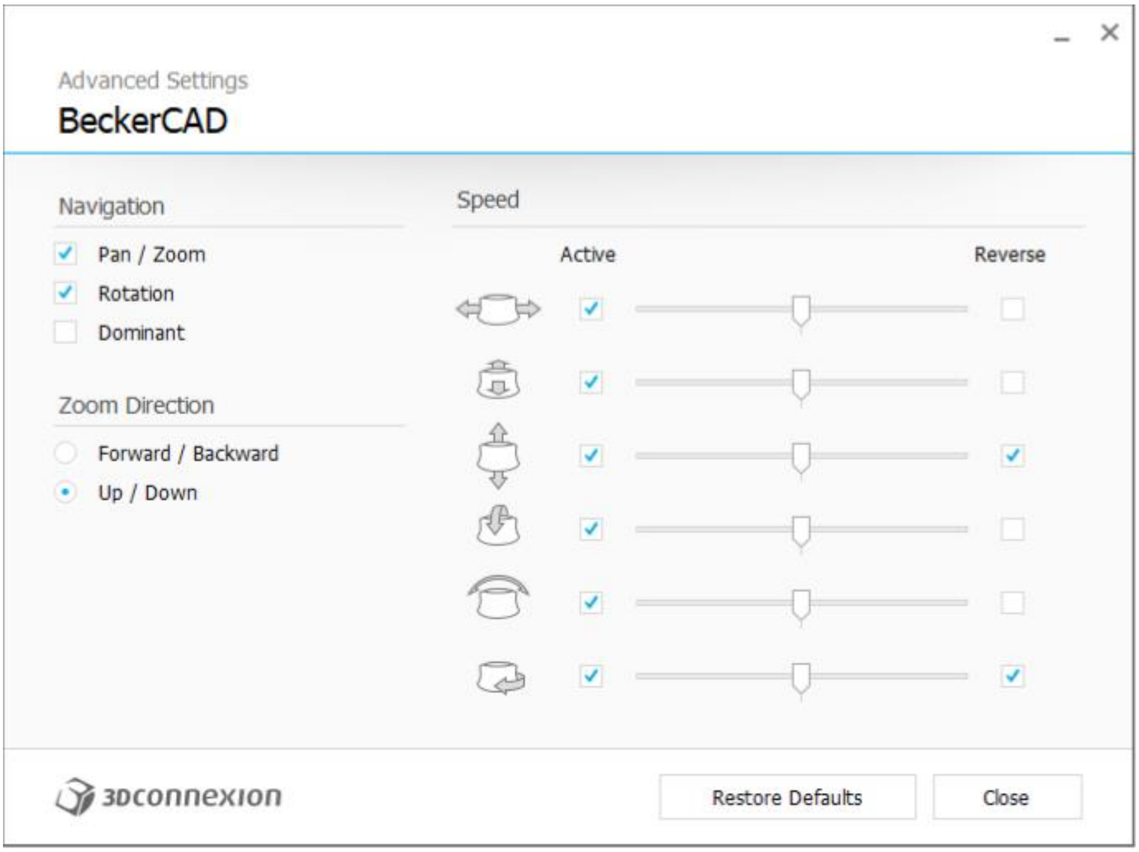
- **Speed:** This slider sets the overall speed for your 3D Mouse. In other words, it changes the amount of force or torque that needs to be applied to the 3D Mouse cap to move an object, scene, or image.

- **Advanced Settings:** This is where you can configure specific 3D Mouse navigation settings. See the Advanced Settings section.
- **Buttons:** Click this button to review or customize which commands, macro's or radial menus are assigned to your 3D mouse buttons. See the Buttons section.

3D-Mous: Advanced Settings

The ***Advanced Settings*** panel allows you to configure 3D Mouse settings for the active application (displayed at the top of the panel).

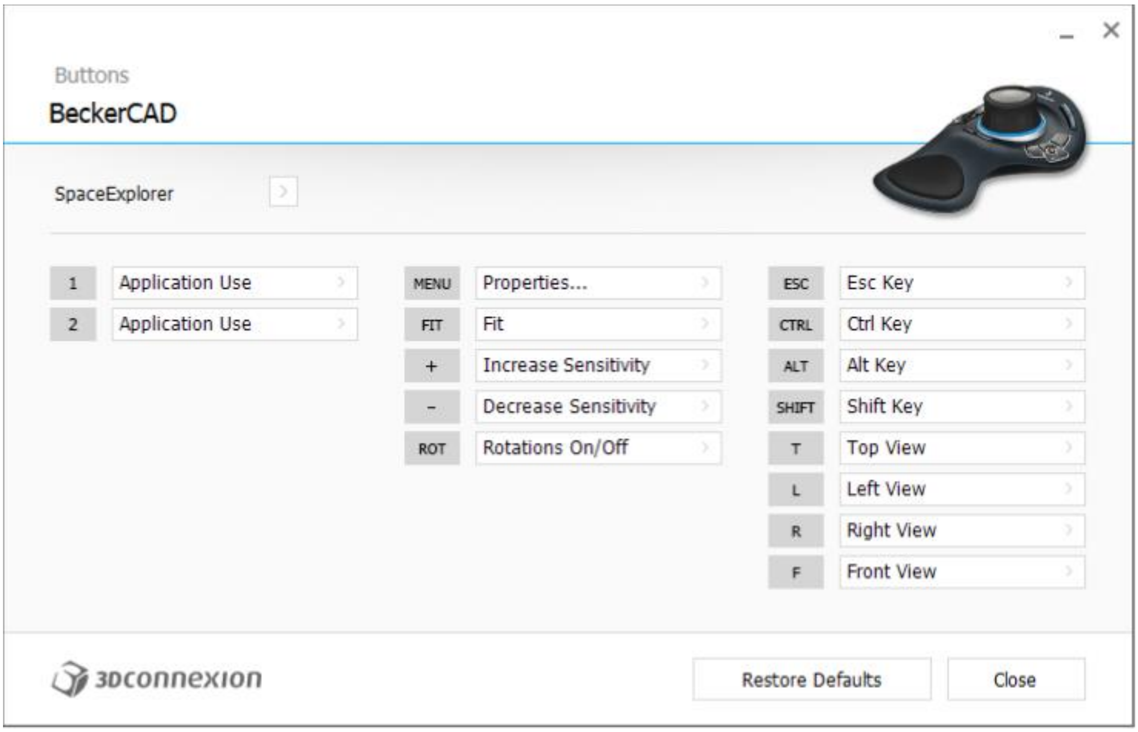
Changes you make only affect the active application, so it's easy to configure your 3D mouse to work just how you want it in each application. If necessary, activate the *BeckerCAD* program window.



3D-Mouse: Buttons

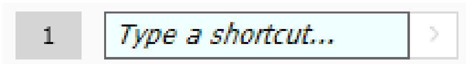
The Buttons panel within 3Dconnexion Properties allows you to assign commands, Macros or Radial Menus to the CadMouse buttons.

The active application is displayed at the top left of the panel and any changes made are only applied to that application. If necessary, activate the *BeckerCAD* program window.



Assigning "Quick Macros" to buttons

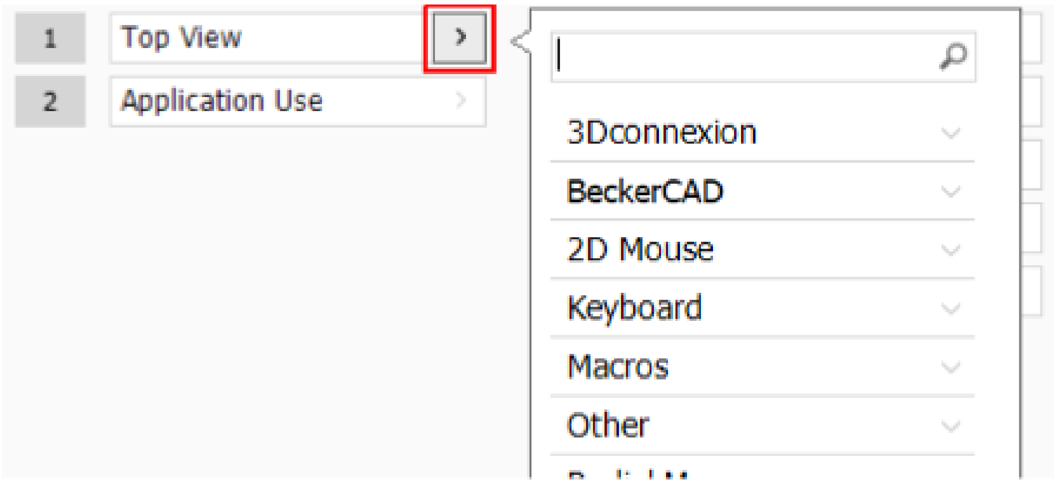
To quickly assign a keystroke or macro to a button, simply click on the text in the button field



and press the required key or key combination. For example, in the screenshot below, clicking on „CTRL + O“ (Function: *Show Original*) would make the field active and the keyboard command pressed would be assigned to button number 1.

Assigning Commands using the flyout window

Clicking on the ">" arrow on the right hand side of the button field opens a flyout window for that CadMouse button:



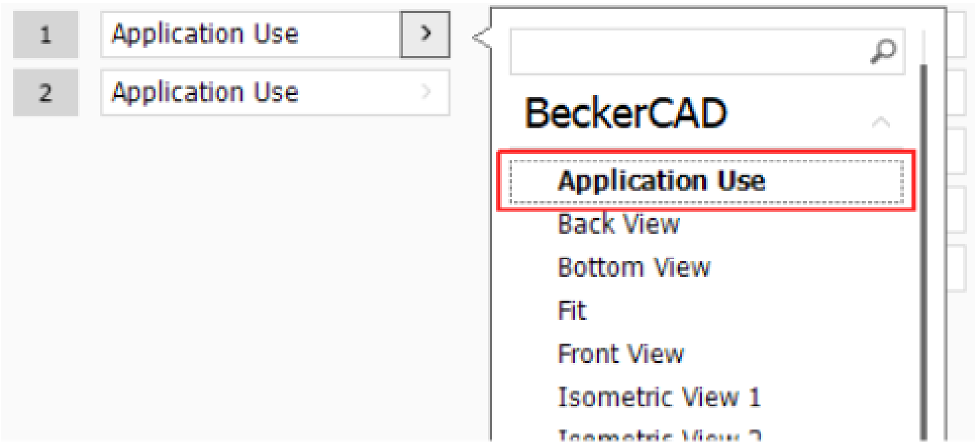
The category of the command that is currently assigned to that button is highlighted in bold.

The flyout window allows you to browse or search for commands in different categories such as 3Dconnexion or application commands.

Assign a *BeckerCAD* command to a key

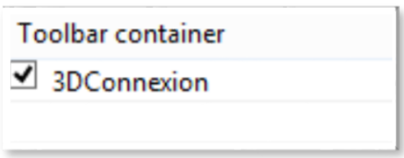
To assign a *BeckerCAD* command to a key of the 3D-Mouse please follow this stepps:

1. Select a ***BeckerCAD*** menu option from the flyout dialogue on a key, ***Application Use***.



2. Switch to the *BeckerCAD* program window and select the menu command ***View, Toolbar***.

3. In the area ***Tool Container*** click the icon ***Create new tool container*** and enter the name ***3DConnexion*** for this icon bar.



4. In the list of the icon bars mark the icon bar ***3DConnexion*** and select the command in the list of commands on the right side. Click the icon ***<< Add*** or choose the command by double click.
5. On the 3D-Mouse click the key which should process this command.
   In the dialogue ***Toolbar,*** in the area ***Function*** a remark is entered, which should not be changed.
   If the icon bar contains several commands, mark the related symbol first.
6. Close the dialogue ***Toolbar*** by clicking the ***OK*** icon.

Further information to ***Toolbar*** and ***Commands*** can be found in the chapter Apply user defined symbol bar and menus.

Creating Radial Menus

To create a new Radial Menu, expand the "Radial Menu" category and click on "New Radial Menu".

This brings up the Radial Menu Editor where you can assign a name and commands to the different sections. Your new Radial Menu will be automatically assigned to the button from where you opened the panel.

Please note:

No direct commands can be assigned to a radial menu.

3.5 Apply user defined symbol bar and menus

In addition to the symbol bars that are generally available to all users you can define your own that are for activating symbols to load commands from the 2D or 3D toolbar or even external programs. This symbol bars will be displayed at the edge of the main window.

You can also start functions via ***user defined symbol bars***. The commands can be assigned to any key and can be started with this key when no other commend is active. User Defined Menus are also defined using the symbol bars definition. The general approach to define the User Defined Menus can be found in the chapter.

General procedure for defining and activating symbol bars

1. When you want to define your own symbol bar, you must start by loading the ***View, Toolbar*** menu command.

Next, complete each of the following steps in turn. You will find a detailed description of them in separate sections of this chapter.

2. Open or create a toolbar file in the area ***Current Toolbar***.

3. Define at least one toolbox with a name.
4. Insert symbols that can be called up from inside *BeckerCAD* or for loading external programs in a toolbox.
5. Activate the ***active symbol bar*** presetting.
6. Accept the settings by clicking the ***Accept*** command button.

Saving and loading a symbol bar

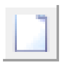
After you have ended your work with the program, and after you have closed the dialog box used for defining symbol bars, the program will save the position and size of the symbol bars just defined in the file UIBECKER.SYS.
The next time you start the program, the symbol bars will be offered.

Create new Toolbar

If you wish to work with different toolbars, a new toolbar can be created very quickly and can be stored with all contained icon bars in a TLW file.

Please follow this steps:

1. Select the menu command ***View, Toolbar***.

   In the area Toolbars the name of the last used or stored TLW file is listed. Save, if you have made changes.
2.  Click the button ***Start new toolbar definition***.
3.  Create at least one icon bar by clicking the displayed icon in the Tool container area and give a name to the new ***Tool container*** in the list of ***Tool containers***.
4. Click the displayed icon ***Save TLW-file as*** in the ***Current Toolbar*** area and enter a name for the new file.

Process Toolbar

To process an existing Toolbar please follow this steps:

1. Select the menu command ***View, Toolbar***.
2. If the Toolbar is listed in the ***Toolbars*** area then doubleclick on the file name. The file is displayed in the ***Toolbar container*** list and the icon bars are offered to be processed.

3.  Alternatively click on the displayed icon ***Open TLW file***.
4. Select a file in the ***Open file*** dialogue window and click the ***Open*** icon.

Define toolbox

A user defined symbol bar cannot itself be used to insert actual symbols: The symbol bar has to contain at least one toolbox in which you insert the symbols.
Use the procedure described here to create your toolbox:

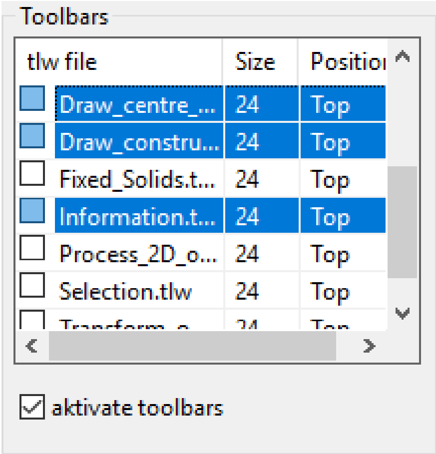
1. Select the ***View, Toolbar*** menu command.
   This step opens the dialog box for defining symbol bars.
2. As described in the chapter Process Toolbar, open the desired TLW file**.**
3. In the area ***Tool container*** click the displayed icon ***Create new tool container*** and enter a name in the list.
   Alternatively Toolbars can be added at the end of the list or inserted above a marked entry using the context menu in the list of toolbars.

   Please note: The icon bars inside the toolbars are listed according to the sorting in the list. The sorting can be changed using the context menu.
4. You can add an info in the field remark. This info has no function outside of this dialogue window.
5. Next, type the name of a symbol in the ***Symbol*** text box, or first click the  icon and then select from the files with ICO as their extension.
   The symbol that has been selected will be displayed at the beginning of the toolbox.
6. You may want to define additional toolboxes to insert them in your toolbar, and this can be done just be repeating Step 2. – Step 4.
7. Which of the toolboxes in the symbol bar is actually available to you can be determined by clicking the adjacent radio button.

Copy toolbars from TLW file

Use the function ***Insert tool containers from selected bars*** in the context menu to transfer toolboxes from other toolbar files (*.TLW) to the currently loaded toolbar.

1. Define a new toolbar or open an existing toolbar, where another tool container should be added.



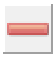
2. Mark in the list of tool containers the one, which should be added to the current toolbar. Press the ***CTRL*** key to select several TLW files.

   Using the context menu, you can add more TLW files.

3. In the list of tool containers in the upper left of the dialog start the context menu and select ***Insert tool containers from selected bars.***

Erase toolbox

When you want to erase a toolbox from a user defined symbol bar use the following method:

1. Select the ***View, Toolbar*** menu command.
   This will open the dialog box for defining symbol bars.
2. Select a TLW file by doubleclick in the ***Toolbars*** list or open a file by clicking the  icon in the ***Current Toolbar*** area.
3. Next, you will see the list containing all the toolboxes. Mark the toolbox that you want to delete.
4.  Click the displayed icon ***Delete selected tool window*** in the ***Tool container*** area or select the respective option from the context menu.

Insert symbols for loading *BeckerCAD* commands

Provided that you have a user defined symbol bar containing at least one toolbox, you can insert symbols for loading commands from *BeckerCAD*.

Use the method described here:

1. Tag one of the toolboxes in the list containing all the toolboxes. This will be the one into which you are going to insert the symbols for loading *BeckerCAD* commands.
2. In the command tree on the right tag the command that you want to insert:

   This is done by clicking the ⊞ icon next to the command branch that you want to open; or click the ⊟ icon to close a branch that is already open.

   In one of the branches already open tag the command that you want to insert.

   Below the command structure tree the last executed commands are listed. When selecting a command here, it is displayed in the structure tree. Even with the dialogue window open, commands can be executed and are shown in the list.
3. Click the ***<< Add*** command button or select the command by double clicking.
   The command is added to the toolbar at the end of the list.
   If a symbol is linked to the command, the former will be displayed in the toolboxes symbol bar.

   Mark a command and click the ***<< Insert*** button to insert the command above the marked command in the list.
4. commands not linked with a symbol will be indicated by the icon shown here.
   You can, however, insert a symbol of your own. Do so by typing the symbol name in the ***Symbol*** text box that is in the ***Function*** section, or by clicking the icon and then selecting a file with ICO as its extension.
5. Type an additional note in the ***Remark*** text box, but only if you want to. This text will be displayed as a QuickTipp. This is helpful especially if you call an external program, e.g. a Python script.
6. You may want to insert additional commands. This can be done by repeating Step 2 – Step 5.
   If the setting ***Add to list end*** is active, each new symbol will be inserted at the end of the toolbox. If this setting is not active, the new symbol will be inserted before the symbol tagged in the toolbox.

Please note:

If commands are already inserted in a toolbar, they can be searched in the command tree by clicking the icon ***Search***. Mark a toolbar and click on the icon for the respective command, then click the ***Search*** button.

If a command is marked in the command tree, in the area command besides the command name also the internal command name is displayed. This internal name is necessary to execute a command using the User defined key commands.

Search functions / command name (F3)

In addition to the options listed above with the button ***Search*** and ***the last functions***, you can search for functions or command names within the structure tree as follows:

1. Click in the structure tree at the top right and press ***F3*** or select ***Find*** from the context menu.

2. Enter the search term in full or in part in the dialog box. Start the search using the ***Find next*** button. After each further click on this button the next search result is displayed.

Match whole text only
There must be no further letters in front of and behind the search text.

Match case
The search is case-sensitive.

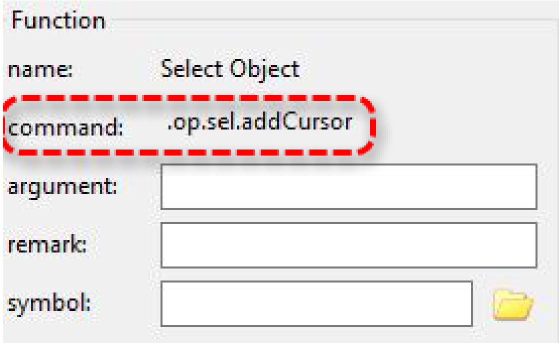
Match regular expression
With regular expressions, any complex search queries are possible. With special characters, search terms can be put together so that not only a single term is found, but also precisely defined combinations of terms.
Examples for the search with regular expressions can be found at the end of this chapter.

Wrap around
The search is continued at the beginning of the list up to the selected element.

Siblings only
The search is restricted to the current structure level within the selected structure tree.

Command name (only in the toolbar dialog)
This option allows you to search for the command name of a

function.



Examples for regular expressions

For regular expression, the search text is expanded using simple controls or control elements. Here are some examples, which can only give a small overview of the entire spectrum of the search with regular expressions.

Character	Meanings	Example
.	Represents exactly one character	.ower = tower, lower
\w	An alphanumeric character	aAbBcC …
\d	A digit	0123456789
\d+	A digit sequence	
*****	No or several repititions	The precending search expression can not occur once, or occur once or several times
+	At least one repitition	The precending search expression can occur once or several times
[ ]	Character class Matches all characters within the brackets. Areas can be specified with the hyphen	[abc] finds "a", "b" and "c". [a-z] finds a lower character. [0-9A-Z] finds a digit and an upper character
( )	Describes a sequence of characters	Grouping of characters and search operators

Insert symbols for loading external files

Provided that the user defined symbol bar contains at least one toolbox, you can insert symbols to load external programs.

Files can be opened with the System Command ***External file call***. The application that is linked to the respective file type on the computer is used. For example, opening a file with the extension *. docx opens *Microsoft Word*, like double-clicking on this file in the Explorer.

Use the following method:

1. Tag one of the toolboxes in the list containing all the toolboxes. This will be the one into which you are going to insert the symbol for loading an external file.
2. Click the ⊞ icon in the command tree at the start of the line ***System Commands***.
3. Tag the line reading ***external file call***.
4. Click the **<< *Add*** command button or select the command by double clicking.
   This step will insert the command in the toolbox at the end of the list.

   Mark a command and click the **<< *Insert*** button to insert the command above the marked command in the list.
5. In the ***argument*** field, enter
   - the action what should happen to the file (for example, *open* for opening the file or *print* for direct printing)
   - and behind the action, quoted in quotation marks, enter the directory and the file name with the extension.

   Example: `open "C: \Text.txt"`
   opens the file *Text.txt* with the standard editor.
6. Next click the  icon in the ***function*** section next to the ***Symbol*** text box, if you want to have an icon displayed on the button. Select a *. ICO file from the file selection dialog.
7. If required, enter an additional note in the ***Remark*** text box. It is displayed as a quick tip when you hover your mouse over the button.
8. You may want to extend the procedure for loading external files. This can be done by repeating Step 3. – Step 7.

Pease note:
If you enter `$DIR_HOME` in the argument field instead of a file directory, the specified file is searched in the *BeckerCAD* working directory (user); `$DIR_EXE` branches to files in the *BeckerCAD* directory ‘Program\Bin’.

Example:
`open "$DIR_HOME\Textures\Boden\boden001.jpg"`
opens the image file with the standard program for picture display.

Insert symbols for loading external programs

Provided that the user defined symbol bar contains at least one toolbox, you can insert symbols to load external programs.

Use the following method:

1. Tag one of the toolboxes in the list containing all the toolboxes. This will be the one into which you are going to insert the symbol for loading an external program.

2. Click the ⊞ icon in the command tree at the start of the line ***System Commands***.

3. Tag the line reading ***load external program***.

4. Click the ***<< Add*** command button or select the command by double clicking.
This step will insert the command in the toolbox at the end of the list.

Mark a command and click the ***<< Insert*** button to insert the command above the marked command in the list.

5. Next click the icon in the ***function*** section next to the ***Symbol*** text box.

6. You will next see the file selection dialog box. Select one of the *.EXE or an *.ICO file in it.
If the file that you have selected is an *.EXE file that can be executed, e.g. C:\WINDOWS\NOTEPAD.EXE the program will find the program symbol for it and insert it in the toolbox.
The program name will be inserted in the ***Argument*** text box.

In the case of those programs that are started with external parameters, you can add to them. For example the entry C:\WINDOWS\NOTEPAD.EXE C:\TEST.TXT would result in the editor starting after you have clicked the symbol: the file named C:\TEST.TXT would then be opened.

Please note:

If the entry inside the ***Argument*** in text box is enclosed by double quotation marks ", the external parameters must be before the terminal pair of quotation marks ".

7. If required, enter an additional note in the ***Remark*** text box. It is displayed as a quick tip when you hover your mouse over the button.

8. You may want to extend the procedure for loading external programs. This can be done by repeating Step 3. – Step 7.

Please note:

If you enter $DIR_HOME instead of the drive in the Argument field, the specified file is searched for in the BeckerCAD working directory (User); $DIR_EXE branches to files in the BeckerCAD directory Program\Bin.

Remove symbol from toolbox

After you have inserted symbols inside a toolbox included in the user defined toolbar, you can remove what is no longer required in the following way:

1. Tag one of the toolboxes in the list containing all the toolboxes. This will be the one that you are going to erase.

2. Tag the symbol that is to be erased.

3. Click the ***Remove >>*** command button.

Save user defined symbol bar

You may want to work with different symbol bars. You can save any symbol bar you have just defined with its settings in a file with TLW as its extension.
The file will be put into the folder that is open and reserved for TLW files.

Use the following method.

1. Select the ***View, Toolbar*** menu command and complete the settings that are to apply to the symbol bar.

2. You will be able to see the name of the TLW file you have most recently loaded or saved inside the ***Configuration*** section.

If this is the name that you want to use, click the ***Save symbol bar configuration*** icon.

But if you want to use a different name for saving the file, click the ***Save symbol bar configuration as*** icon, and type the new name in the file selection dialog box.

Display Toolbar

User defined toolbars consists of at least one icon bar with commands from the 2D or 3D command pool or with icons to start external programs. They will be displayed at the border of the main window. It is possible to display multiple toolbars. If the same position is given they will be displayed in separate lines or beside each other.

To display the icon bars of user defined toolbars in the main window, please follow this steps:

1. Select the menu command ***View, Toolbar***.
2. The sorting of the ***TLW files*** in the ***Toolbars*** area defines the sorting of the display if the same position is given. Open the context menu in the ***Toolbars*** area to select one of the following options.

    **Add toolbar (N)**
   The selected toolbar is added at the end of the list.

    **Insert toolbar (Strg-N)**
   The selected toolbar is inserted above the marked file.

    **Open all toolbars (Shift-Strg-N)**
   Adds all leaving toolbars from the Standard TLW directory to the list of files.

3. Using the context menu you can define the size of the icons and the position of every toolbar in the program window.
4. Activate the checkbox in front of a toolbar to display the contained icon bars.
5. You can change the position of the toolbar by clicking the **U** (= up in list) key or **D** (= down in list) key.
6. Activate the checkbox ***Toolbar active***.
7. Confirm your changes by clicking the ***Apply*** button if further changes should be made or the ***OK*** button, if the dialogue should be closed.

Position and size of the displayed icon bars are stored in one of the file UIBECKER.SYS when leaving the program. If the program is started again the TLW files are readed and the contained toolbars and icons will be displayed.

Remove Toolbar

It is possible to load and process multiple toolbars in the ***Toolbar*** dialogue window. To remove a toolbar please follow this steps:

1. Select the menu command ***View, Toolbar***.

2. Mark the toolbar to be removed in the list of toolbars.
You can use the standard commands to mark the toolbars, e.g. Ctrl = collecting or Ctrl+A = mark all.

3. Click the ***Delete*** key or select the option ***Remove toolbar*** from the context menu.

Please note:

If you have added several toolbars during the current session, they can be deleted in one step without marking them. For this select the option ***Remove last added toolbar*** from the context menu.

User Defined Menus

By clicking any key a user defined menu is offered at the current mouse position, where user defined commands can be started or dialogue windows can be opened.

The content of the menus can be defined in the Toolbar .

General procedure to create a user defined menu

1. To select the commands for the menu, first select the menu command ***View, Toolbar***.

2. Create at least one new toolbar.
If a name is defined for the toolbar, this name is used as a header for the menu. When multiple toolbars are created a relevant section is shown in the menu.

3. Add the desired icons for the commands or for dialogue windows into the toolbar. The names of the commands are transferred from the program into the menu automatically. Alternatively a user defined name can be used by defining it in the field ***argument*** in the ***Function*** section.

4. Save the settings in a TLW file in the current TLW directory.

5. Then the file HOTKEYS.TXT can be edited with an editor e.g. **Notepad**. This file can be found in the directory <Install-directory>\Program\Bin.
The file contains predefined entries for the keys
0 to *9*;*SHIFT-0* to *SHIFT-9*;*CTRL-0* to *CTRL-9* and

SHIFT- CTRL -0 to *SHIFT- CTRL -9* .
Enter the name of the TLW file at the end of the predefined lines without extension following the example:
0 ".sys.wnd.openToolWinMenu Standard"
By clicking the 0 key the content of the file *Standard.tlw* is offered at the current mouse position.

Please note:

To be sure to not to overwrite your changes for the user defined menus during an update installation, please save your changes in the file USER_HOTKEYS.TXT. Further informations concerning this file can be found here.

6. Save the file HOTKEYS.TXT and restart the program *BeckerCAD*.

In the program the user defined menus can now be started by clicking the according keys or key combinations.

Please note:

The keys **1** to **9** are predefined with examples. They can be expanded or exchanged according to your needs. The related TLW files can be found the the standard TLW directory.

Please be aware: If you change the sample menus, they should be renamed or stored in a directory different from the standard TLW directory, because they will be overwritten when a new installation is proceeded.

3.6 Working with windows

When starting *BeckerCAD*, you can choose in the start wizard whether you want to create a new file with a 2D drawing or a 3D model according to an industry.
In this case a corresponding template file is loaded, which selects either the 2D drawing window or the 3D view window as the current window.
The command button in the next dialog window you are shown, you can adapt the settings taken from these template files to your requirements.
Instead, you can select the option ***Close.*** This activates the STANDARD.TPL template file settings.

So you have both the required window settings and other required settings available to you at the start, you can now change the template files referred to.

You can also set up additional template files, which you can reselected as required using the menu command ***File***, ***New Model***.

Guidance on how to do this is to be found in the chapter entitled ***Determine program settings, Determine model specific settings***.

Open new 2D drawing window

For each 2D drawing, additional view windows can be opened in which different views can be displayed i.e. sections of the same drawing. When you are drawing and processing 2D objects, the 2D view windows can be changed between as required.
In this way, for example, in both 2D view windows, two different sections can be displayed enlarged and e.g. a line drawn directly from a point in the first window to a point in the second window.

Select one of the following procedures:

1. Activate the 2D drawing window of the drawing for which another view is to be opened.

2. Select the menu command *View, New 2D Drawing View*.

Or:

1. Select the menu command ***View, Model Explorer***.

2. Activate the ***Model*** option card.

3. In the tree structure, tag the drawing for which a new view window is to be opened.

4. Use the key combination ***Ctrl+Shift+n*** or open the context menu for this drawing and select the option ***New, New View***.

The new 2D view window has the same entry in the heading line as the previous.

Please note:

As an alternative to your opening a new view window you can use the ***Window, Split Horizontally*** menu command or the ***Split Vertically*** menu commend to split the current 2D drawing window. In this way you will not need so much space, because the 2D symbol bar and the 2D toolbar will be displayed only once.

If you have closed all the view windows of a drawing intentionally or accidentally, they can be opened again using the ***Model Explorer***, by selecting the description of the required drawing and use the key combination ***Ctrl+Shift+n*** or loading the ***New, New View*** option from the context menu.

Closed graphic windows are marked with the symbol  in the Model Explorer.

Open new 3D view window

If a further 3D window is to be opened for better orientation or for simplifying the definition of points which are not visible in the current 3D window for example, use the menu command ***View, New 3D View Window***.

When defining points or identifying objects, simply move the cursor to the required view window and carry out the action.

If, for example, the view in a 3D window is to be modified, activate this window by e.g. clicking on the window frame.

Please note:

As an alternative to your opening a new view window you can use the ***Window, Split Horizontally*** menu command or the ***Split Vertically*** menu commend to split the current 2D drawing window. In this way you will not need so much space, because the 2D symbol bar and the 2D toolbar will be displayed only once.

If all 3D view windows have been closed deliberately or by mistake, these can be reopened using the menu command ***View, New 3D View Window***.

Closed graphic windows are marked with the symbol in the Model Explorer.

Select current window

To select one of the opened view windows as the current window, use the following possibilities:

Full screen display

- First open the ***Window*** menu and then click the line containing the name of the window that you want.
  After you have done this windows which have been reduced to a symbol will be restored.
- You may also use the key combination *Ctrl Tab* or the menu command ***Window, Next Window*** by scanning the opened windows, until the desired window is displayed. The reduced window symbol will not be restored here.

Reduced display

In this case you may select one of the visible windows as the current window; by clicking on this window.

Symbol display

 You can use either of the system icons to restore or to maximise.

Open / Activate 3D Window

Especially if a model contains many 2D drawings, you have a quick option to activate the 3D window via the new menu command ***Windows, Open/Activate 3D window (Alt-N-3)***.
When the command is called, a new 3D view window is created if all 3D windows were closed.

Modify window size

To modify the size of the reduced window which is now restored in the main window, proceed as following:

1. Move the cursor onto the frame of the window. The cursor will now be displayed as a double arrow.

2. Move the cursor – while keeping the left mouse button pressed – in the direction of one of the arrows.

3. When you have the desired window size, release the mouse button.

Modify position of window

Proceed as follows to change the position of minimized or restored windows within the main window:

1. Place the cursor in the header line of the window.

2. Move the cursor and thus the window while holding down the left mouse button.

3. If the window has the desired position, release the mouse button.

Tile all windows

To arrange all opened windows of a model choose one of the following:

- ***Windows, Tile vertical***
- ***Windows, Tile horizontal***
- ***Windows, Cascade***

Close window

When you have decided that you do not need one or more of the 2D or 3D view windows, you can close it (or them) without losing any of the data.

You can do this in the following ways:

-  Click this system icon.
-  Open the system menu by clicking this icon on the left side of the header line. Then select the ***Close*** menu command.

Please note:

You can re-open a window that you have erased by using the methods described above to open a new 2D drawing window or a new 3D view window.

Split window

Sometimes you may want to display different views of the same drawing or of the same 3D model so that you can see more clearly what you are doing or reduce the number of separate steps you have to complete. You can solve this problem by splitting the view window that you are currently working in. When you are drawing and dealing with objects of different kinds you can change as often as you like between the windows.

Use the following method:

1. Select the ***Window, Split Horizontally*** menu command or the ***Split Vertically*** menu command.

2. The size of the partial window can be changed by placing the cursor of the splitting line. The cursor will then change to a double-headed arrow.

3. Then, keeping the left mouse button pressed, drag the splitting line to wherever you want.

Two colours are used to indicate the function of each partial window. The active 2D drawing window has a red frame, whereas the 3D view the active one has a blue frame.

You select the partial window as the active one by clicking inside it. Another method is to load a command and then specify a point or to select an object in the partial window that you want to activate.

Please note:

The commands for changing a view or for redrawing always apply

only to the window that is active.
This means that you must always click the partial window first to activate it.

Apply split to another window

You can apply the split set in one view window to another window in the following way:

1. Activate the window containing the split that you want to apply.
2. Select the ***Window, Copy split*** menu command.
3. Activate the view window to which the split is to apply.
4. Select the ***Window, Copy split*** menu command.

Remove current window split

You can remove the current split from the window by using the ***Window, Remove current split*** menu command.

Remove last window split

The program keeps an internal record of all the splits that have been inserted in a partial window.

The most recently inserted split in the view window that has been selected can be removed by using the ***Window, Remove last split*** menu command.

Remove all window splits

All the splits in the view window that has been selected can be removed by using the ***Window, Remove all splits*** menu command.

Arrange all dialog boxes

If you often have to change the resolution of your screen, e.g. because using a beamer, using the menu commands ***Windows, Arrange all*** all dialog windows can be moved in the actual primary screen.

Dialog windows like the status dialog box, which maybe appears outside, are displayed the again.

3.7 Control model display

The commands for controlling the view in the toolbar for each 2D drawing window or 3D view window can be loaded at any time.
Any command which has already been activated is not aborted if any of these CAD commands are selected.

These commands make it possible to display a different section in each of these windows or a different perspective.

Enlarge a window display

In order to enlarge the display of objects to fill the entire size of a rectangular section (window), follow this procedure:

1.  Select the menu command ***View, Enlarge Window*** .
   The cursor contains a magnifying glass as an additional symbol.
2. Using the cursor determine the first corner point of the window which is to contain the objects to be enlarged.
3. Determine the diagonally opposite corner point of the window.
   The objects which lie within the window are enlarged so that they fill the current window.
4. Redefine a window or abort using the right mouse button.

Please note:

You can decide whether the enlargement rectangle is to be defined using 2 points or by dragging whilst keeping the mouse button pressed. The setting can be made using the ***Settings, Options*** menu on the ***User Interface*** dialog box.

Enlarge display (+)

In order to display the content of the current window in an enlarged form, follow this procedure:

1. Select the menu command ***View, Zoom +***.

   A magnifying glass symbol is added to the cursor  .
2. Using the cursor, determine the fixpoint.
   This point is moved to the centre of the window. All objects are displayed enlarged by a factor of 1.2.
3. Redefine a fixpoint for further enlargements or abort using the right mouse button.

Reduce display (-)

In order to display the content of the current window in a reduced form, follow this procedure:

1. Select the menu command ***View, Zoom -***.

A magnifying glass symbol is added to the cursor  .

2. Using the cursor, determine the fixpoint.
This point is moved to the centre of the window. All objects are displayed reduced by a factor of 1.2.

3. Redefine a fixpoint for further reductions or abort using the right mouse button.

Vary View (w)

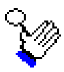
 After clicking on the displayed icon or using the hotkey ***w*** (or ***W***) the command ***Vary view*** is started.

This command can be used to ***Move view*** and/or ***increase*** / ***decrease*** and also to ***rotate*** if the 3D window is active. The condition to move a window is, that the screen window must be decreased or increased before, so that the screen window can also by moved by the scroll bars.

By double clicking in the 2D or 3D window the view can be centred. Double clicking in a 3D window also defines the viewpoint, which is used as centre point for rotation. The centre of gravity is used as centre point for rotation when double clicking on a surface. Double clicking on an edge uses the nearest endpoint, midpoint, centre or quadrant point as centre point for rotation.

 **Pan view**

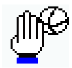
Start the command and click into the active view. Keep the mouse button pressed and move the view with the mouse. The view can be moved until it reaches the border of the screen or until the scroll bars are in there end positions.

 **Decrease / Increase view**

To decrease / increase the view in the active window choose the following:

- In case the mouse is equipped with a wheel this can be used to decrease / increase the view.

- Alternatively the view can be decreased / increased by using the ***CTRL-key*** and moving the mouse pointer up and down the screen.
- Also the view can be decreased / increased by clicking the middle mouse button until the icon appears and then move the mouse pointer up and down the screen.

 **Rotate view (only in the 3D window)**

To rotate the view in the 3D window choose one of the following:

- Use the ***Shift-key*** and press the left mouse button.
- Also the view can be rotated by clicking the middle mouse button until the icon appears and then press the left mouse button.

 **Rotate View around Z Axis (only in the 3D window)**

To rotate the view around the Z axis of the current work plane please follow one of this steps:

- After starting the command keep pressed the left mouse button and click the key combination ***Ctrl-Shift*** and move the cursor left or right.
- After starting the command click the middle mouse button as often as the displayed icon is attached to the cursor icon. Keep the left mouse button pressed and move the cursor to the left or right to rotate the view around the Z axis of the current work plane.

Change view with mouse or keyboard

The view inside a drawing or the 3D window can be changed easily during an action by using the keyboard or the scroll button of the mouse.
For this the following commands can be used:

Functions of the cursor keys

Cursor key up ↑
The window slider moves up, the view is moved down.

Cursor key down ↓
The window slider moves down, the view is moved up.

Cursor key left ←
The window slider moves left, the view is moved right.

Cursor key right →

The window slider moves right, the view is moved left.

By additionally pressing the ***Strg-key*** the view is moved faster.

Page up ↑

The window slider moves up in big steps, the view is moved down.

Page down ↓

The window slider moves down in big steps, the view is moved up.

By additionally pressing the ***Strg-key***, the slider is moved in big steps to the left or the right.

(* To move the view with the keys, the slider must be visible. The movement steps are depending on the size of the slider)

Functions of the scroll button

With the scroll button of the mouse the view can be zoomed in or out as well as moved. The functions in an overview:

Turn scroll button forwards

The view is zoomed out. The actual cursor position defines the direction of the zoom operation.

Turn scroll button backwards

The view is zoomed in. The actual cursor position defines the direction of the zoom operation.

Shift-key + scroll button forwards

The scroll bar* moves upwards, the view is moved downwards.

Shift-key + scroll button backwards

The scroll bar* moves downwards, the view is moved upwards.

Strg+Shift-key + scroll button forwards

The scroll bar* moves to the left, the view is moved to the right.

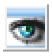
Strg+Shift-key + scroll button backwards

The scroll bar* moves to the right, the view is moved to the left.

(* To move the view using keys, a scroll bar must be visible. The step size of the movement is depending on the size of the scroll bar)

Overview window

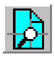
If a part of the current window is displayed in an enlarged form, the entire content of the window can be displayed in an overview window. In drawings which have a particularly high number of objects or drawings which do not have a standard sheet format, the current window can be defined as a high optimised display list. In particularly with lots of elements, the access speed e.g. in the framework of selection or snap point options, is increased considerably. A display list is managed as standard when opening or setting up a model. Only in special cases is it necessary to influence this display list with user defined settings.

In order to open the overview window, load the menu command ***View, Overview Window*** .

The commands in the overview window can be used to move the enlarged area, redefine, tag, declare to the display list or resort the drawing elements within it.

The position and size of the overview window is saved in the file UIBECKER.SYS when the program is quit.

Define window in overview window

1.  Click on the displayed icon.

2. Define a corner of the new window, drag the window open whilst keeping the left mouse button pressed until the required size is reached.
Then release the mouse button.

Move window in overview window

1.  Click on the displayed icon.

2. Within the overview window, define a point using the cursor.
The window is moved so that its centre lies on this point.

Redraw view in overview window

1.  Click on the displayed icon.

2. The content of the overview window is redrawn.

Show all

If you have later drawn objects outside the window displayed in the overview window, this command can be used to also display the objects outside the previous view.

1.  Click on the displayed icon.

2. All objects in the drawing will be displayed to the full size in the overview window.

Please note:

If the following command ***Display current display list*** is active, the ***Show all*** command is deactivated.

Show all / display current display list

The command ***Show All /Display Current Display List*** can be used to select whether the display list defined by yourself or all drawing objects are to be displayed in the overview window.

Display current DPL

1. If the  icon is displayed, the current display list is displayed in the overview window.
2. Click on the displayed icon in order to change to the ***Show all*** command. Then, the entire content of the current drawing window is displayed in the overview window.

Show all

1. If the  icon is displayed, the entire content of the current drawing window is displayed in the overview window.
2. Click on the displayed icon in order to change to the ***Display current DPL*** command. Then, the entire content of the current display list is displayed in the overview window.

Activate/deactivate frame around display list

The ***Tag current DPL*** / ***Remove DPL tag*** command can be used to display or suppress a frame around the current display list.

Display frame

1.  Click on the displayed icon.
2. The current display list is allocated a frame.

The display of the frame can be determined using ***Settings, Options*** on the ***Autopan/Overview window*** option card.

Suppress frame

1.  Click on the displayed icon.
2. The frame around the current display list is removed.

Current view in display list

In drawings which have a particularly high number of objects or drawings which do not have a standard sheet format, the display list can be defined by the user themselves. However, this is only necessary if there are a particular large amount of objects in a section of the drawing. In order to define the display list, proceed as follows:

1. Enlarge the required section using the ***Define Window*** command or using the ***Zoom Window*** command in the toolbar on the top border.
2.  Click on the displayed icon.
3. The doubled size of the selected section is defined as the display list. Using the ***Resort line objects*** command, you can considerably increase the access speed to objects in this section.

Resort line objects

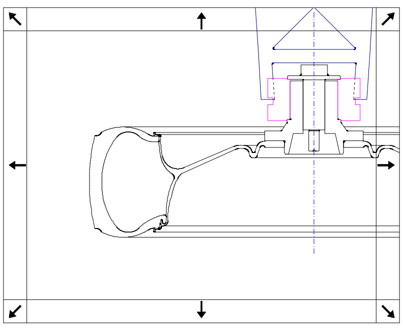
When loading a drawing or redrawing, objects are drawn in the order that they were generated. This order can be optimised for the section of your drawing, by using the ***Reselect line objects*** command to resort the list of objects in individual sections.

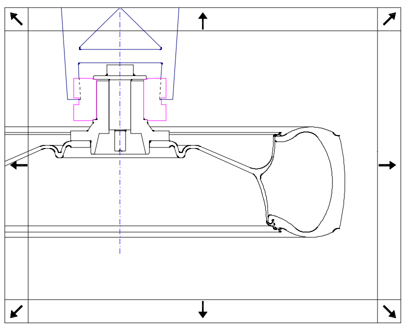
1.  Click on the displayed icon.
2. The content of the objects taken into the display list are resorted in the current drawing window. The drawing order is optimised.

Move the current displayed area automatically (autopanning)

If an enlarged window is defined, it can be moved automatically over the current drawing window.

Put the cursor near by a border of the current drawing area. After a little timeout the displayed window will be moved in this direction.





The relation of the autopanning can be influenced by settings, which can be determined using the menu command ***Settings, Options*** on the ***Autopan/Overview Window*** option card.

Centre view (*)

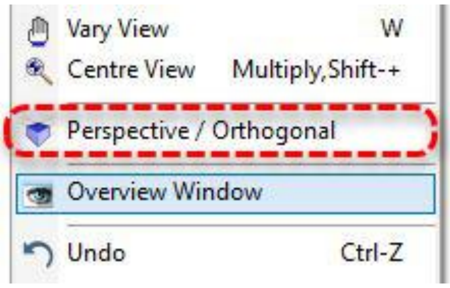
In order to move the display of objects to the centre of the current window, follow this procedure:

1. Select the menu command ***View, Centre View***.
2. Using the cursor, define a point which is to be moved to the centre of the current window.
3. Redefine a point or abort using the right mouse button.

Perspective / Orthogonal View

A perspective view is the natural view for the eye. Parallel lines run in the distance in a vanishing point.
To switch between perspective and orthogonal views in the 3D window, select the ***View***, ***Perspective / Orthogonal*** menu option

If no command is active in the 3D window, you can also change the view via the Edit Menu (middle mouse button).



Please note:

When zooming in or out of the view with the mouse's scroll wheel, the view is centered on the vanishing point or a previously defined viewpoint.

Display all objects

 This command can also be loaded using this icon in the top of the 2D or 3D toolbar.

Use this command if objects have been moved, rotated, mirrored or scaled in such a way that they are no longer visible within the drawing boundaries or model 3D space.
This redetermines and redisplays all objects.

In order to do so, select the ***View, Show All*** menu command

Please note:

In the 2D drawing window, the drawing can be displayed to fill the window by selecting the menu command ***View, Show Original***.

In the 3D view window, the model 3D space can be displayed to the full by firstly selecting the menu command ***View, Original Workplane*** and then activating the required perspective representation.

Display original

 This command can also be loaded using this icon in the top 2D or 3D toolbar.

The 2D drawing can be displayed to fill the size defined with ***Settings, 2D Drawing*** using the menu command ***View, Show Original***.

Display last view

This command can also be loaded using this icon in the top 2D or 3D toolbar.

For each 2D drawing window and each 3D view window, the last selected view status is always saved. Click a number of times on the displayed icon in the toolbar.

This command can be used to update the previously defined view status step-by-step.

Display next view

This command can also be loaded using this icon in the top 2D or 3D toolbar.

For each 2D drawing window and each 3D view window, the last selected view status is always saved. Click a number of times on the displayed icon in the toolbar.

If the command ***Last View*** has previously been used, this command can be used to update the next view status step-by-step.

Redraw (F5)

This command can also be loaded using this icon in the top 2D or 3D toolbar.

If the drawing in the current window contains inaccuracies due to processing or erase commands, the drawing can be regenerated using the menu command ***View, Redraw***.
Current settings such as selected windows, perspectives and display attributes of objects are not modified by doing so.

Select 3D standard view

In the 3D view window, one of ten standard views can be selected directly.

This can be done using the menu command ***View, New 3D Drawing View*** or by using the symbols on the *Camera* toolbox.

Use the menu command ***View, Camera*** to open this toolbox.

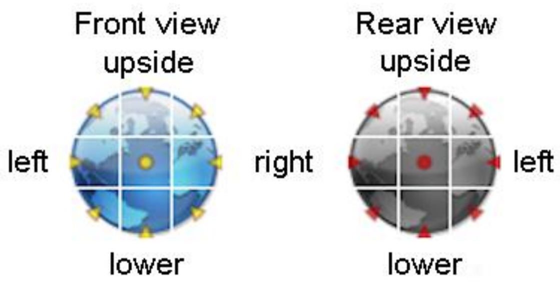
Position this toolbox at any position on the screen.

The toolbox contains symbols for activating the 3D standard views. These always refer to the planes of the global coordinate system.

These show e.g.

- the top view of the XY plane viewed from the positive Z axis,
- the front view of the XZ plane viewed from the negative Y axis,
- the left view of the YZ plane viewed from the negative X axis.

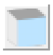
The toolbox contains the following symbols for the selection of views:



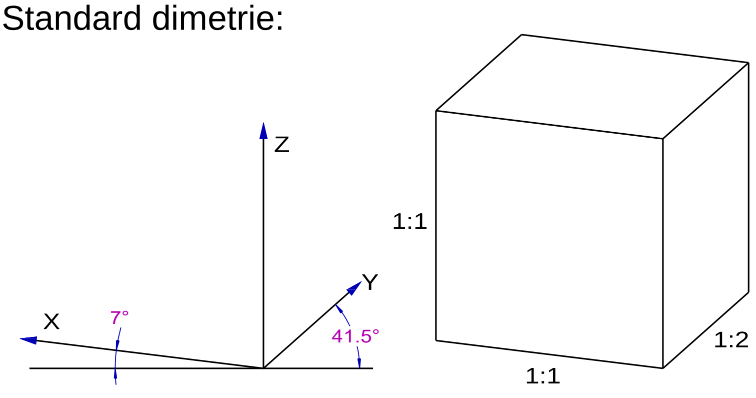
These icons can be used to select the main view directions named in the illustration and also to select the combined view directions such as from front, left and top.

For views which contains the important information in the front view, two additional icons are available for dimetric projections:

 Dimetric left

 Dimetric right

Standard dimetrie:



Please note:

The centre for each view is the focal point. This is positioned in the middle of the view window.
When defining the focal point using the command ***Dynamic Rotate/Enlarge (Define Viewpoint)*** , ensure that the previously selected section of the model remains in the area of the view window when changing the view.

Navigate View using Camera Cube

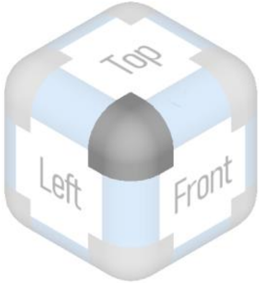
The camera cube is a 3D navigation tool that allows you to switch between standard and isometric views.

Select the menu command ***View***, ***Camera Cube*** to open the dialogue window.

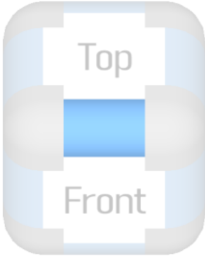
By clicking on the cube, you rotate to one of the predefined views. A click on the home button switches to the home screen. Use the arrow symbols to rotate the view. You can also open the selection dialog for dimetric views.

The settings for the animated view can be defined using the menu command ***Settings***, ***Options*** in the tab ***User Interface*** in the section ***Animated view in 3D window***. The settings also affects the command  ***View on Face / WP***.

- Activate / Deactivate animation
- Number of steps to define, how many steps in the defined time interval are made until the end view has been reached. Enter a value between 1 and 100.
- Time interval per step (ms) defines the interval between two steps. Enter a value between 1 and 1000 (ms).



Corner



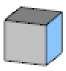
Edge

Face

Click on one of the highlighted areas to switch to the default view. Another click on the same area rotates the view 180° to the rear area.

The home button switches to the standard perspective *front left top (F6)*.

Click on one of the arrow buttons to rotate the view by 90°. If you hold down the Ctrl key at the same time, the view is rotated by 45°



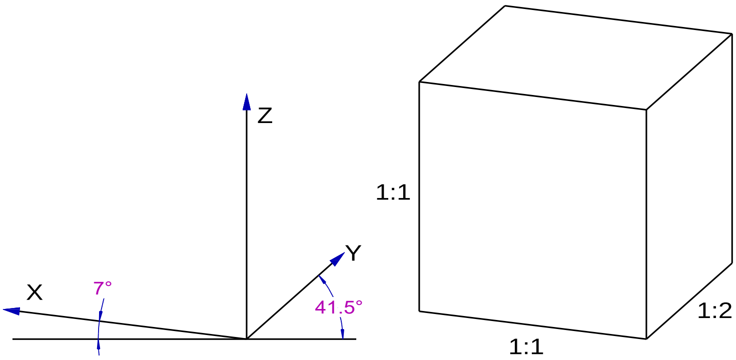
Opens the dialog ***Dimetric Views***.

Dimetric Views

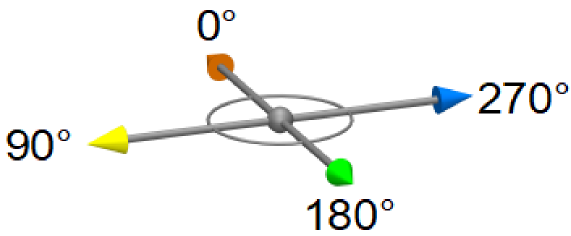
To display the model in a dimetric view in the 3D window, select the menu command ***View***, ***Dimetric Views***.

In the dimetric view, one edge of the component is drawn at an angle of 7°, the other at an angle of 42° to the horizontal.

Standardised dimetry:



The predefined dimetric views rotate the view of the model in the direction of the angle relative to the global 3D coordinate system with the angle aligned to the right or left.



 Example for the view ***Dimetric 0° left***.

Please note:

See the chapter Custom Views in the 3D Window for information on how to save additional dimetric or arbitrary views.

For the representation of the view in *BeckerCAD* the mathematically exact definition of the dimetry is used. The widths and heights of a box with the edge length 10 are shortened to $\frac{2}{3}\sqrt{2} \approx$ 9,428 and the depths are shortened to $\frac{1}{3}\sqrt{2} \approx$ 4,714. The lines of the front view are displayed at an angle to the horizontal of $\arcsin\frac{1}{8} \approx$ 7,18° and the side view of $\arcsin\frac{\sqrt{7}}{4} \approx$ 41,4096°.

Display workplane in top view

If 2D objects are to be drawn in the current WP or swept or rotational solids are to be generated from them, it is useful to use the top view of the WP as the entire "drawing area" respective to the set size.

1. Select the 3D view window, in which the top view is to bedisplayed on the WP.

2.  Click on the displayed icon in the 3D toolbar or activate the menu command ***View, Original Workplane***. Afterwards the top view of the current WP will be displayed as the original that is in accordance with the specified dimensions.

Please note:

The dimensions of a WP can be changed with the help of the ***Edit WP properties*** menu command that you can activate by clicking  din the 3D symbol bar.

Select special 3D view

 By clicking on the displayed icon in the 3D top toolbar.

This command can be used to select a special view for the 3D view window, by dynamically rotating and/or zooming the model 3D space including all objects.

Dynamic rotate with defined focal point

This command can be used to dynamically rotate the view about a defined point: ***the focal point***.

The rotation axis for a dynamic rotation runs, as long as it has not been otherwise defined using the focal point, simply through a point in the screen plane. Its position in the model space is more or less determined by a calculation procedure.
Particularly when working in an enlarged display – this can lead to objects that you wish to view from another direction being rotated out of the visible area of the view window.

For this reason, in the ***Dynamic Rotate*** command, you should define a focal point on an existing solid about which you wish to rotate the view.

Proceed as follows:

1. Define the focal point on a solid surface of a solid or by defining a point in the current work plane.
The model 3D space is moved so that the define point lies in the centre of the 3D view window and the rotation axes run through this spatial point.

If you do not wish to define any focal point, abort focal point definition using the right mouse button and then proceed as follows.

2. Using the cursor, define a point as near to the centre of the screen as possible.

3. Starting at this point, if the cursor is moved to the right or left, the model 3D space is moved in the same direction about the vertical screen axis which lay in the middle of the 3D view window when the command was activated for the first time.
If the cursor is moved upwards or downwards, the rotation takes place in the same direction about the horizontal screen axis.

4. Determine the selected view, by confirming with the left mouse button.
The right mouse button can be used to abort. The previous view is then redisplayed.

5. If the view does not fulfil your requirements, follow the procedure from step 1 again.

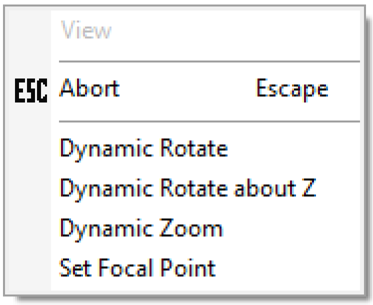
Dynamic rotation around the Z axis

This command allows the rotation of the view around a defined focal point with a rigid Z axis of the current work plane.

Please follow this steps:

1. Activate the desired Work plane, to define the rigid Z axis.
2. Start the command ***Dynamic Rotate with Focal Point*** by clicking the displayed icon.
3. Define the focal point by clicking a surface (=rotation point of the view).
   The 3D space is moved in the way, that the defined point is in the centre of the 3D space.

   If no focal point should be defined, cancel the focal point definition by clicking the right mouse button and continue as described below.
4. Press the middle mouse button to start the context menu:



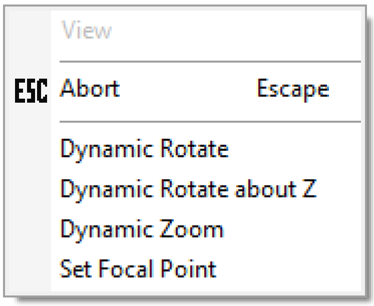
5. Select the command ***Dynamic Rotate about Z***.
6. Define a point by cursor in the centre of the screen.
7. If the cursor is moved to the left or the right, the 3D space is rotated around the Z axis of the current work plane.
8. Set the desired view by clicking the left mouse button.
   Using the right mouse button instead, the command is cancelled and the previous view will be displayed again.
9. If the view is not correct, you can proceed with step 4 again and maybe select another option from the context menu.

Dynamic zoom

1. Define the focal point on a solid surface, by identifying a solid.
   The model 3D space is moved so that the define point lies in the centre of the 3D view window and the rotation axes run through this spatial point.

If you do not wish to define any focal point, abort focal point definition using the right mouse button and then proceed as follows.

2. After starting the command, press the middle mouse button. This loads the following context menu:



3. Select the command ***Zoom Dynamically***.
4. Using the cursor, determine a point in the screen plane. This is the "handle" on which the model 3D space is "held and moved".
5. "Move the model to the back" by moving the cursor upwards starting from the reference point or "drag forwards" by moving the cursor downwards.
6. Determine the selected view by confirming the left mouse button. The right mouse button can be used to abort the command. The previous view is redisplayed.
7. If the view does not fulfil your requirements, follow the procedure from step 1 again.

Please note:

The following is valid for rotation:
If a focal point has been determined, this is kept as the reference point for the rotation axes. If a section of a model has been selected which is not close to this focal point, a new focal point should be determined.

The further away the "handle" from the middle of the 3D view window, the larger the possible rotation angle.

The following is valid for zooming:
The nearer the "handle" to the lower edge of the 3D view window, the smaller the model 3D space can be displayed.
The nearer the "handle" to the top of the 3D view window, the larger the model 3D space can be displayed.

Rotate view on solid face / workplane

 Click on the displayed icon in the 3D toolbar to start the ***View On Face/WP*** command.

This command can be used to change between a solid face or workplane in the top view.

Settings for this command can be made using the menu command ***Settings, Options*** on the ***Autopan – overview window*** option card:

- Activate /deactivate animation
- Number of steps defines how many steps are to be traced back in the animation during the set time interval, until the view is reached. Enter a value between 1 and 100 and confirm the entry with ENTER or use the arrow command buttons next to the entry.
- Time interval per step (ms) defines the time interval between 2 steps. Enter a value between 1 and 1000 (ms) and confirm the entry with ENTER or use the arrow command buttons next to the entry.

After you have started the command using the command button, identify a solid face, a 3D axis or the frame of a workplane. The 3D view is rotated in the top view of the selected solid face, a 3D axis or WP.

Presentation Mode

In the presentation mode predefined views in the 3D window can be stored to be reused in the design phase or in presentations just by double clicking them. When several views are stored, an animation can be started showing all the views.

Select the menu option ***View*** to start the ***Presentation mode***. Alternatively the commands of the presentation mode can be inserted in a user defined icon bar.

A presentation is prepared as follows:

1. If a presentation is already stored it can be selected and loaded using the icon ***Open Camera Animation ...***  .

2. To insert a new view into the presentation list, first define the new view in the 3D window and then click the icon ***Add (Ctrl+N)***.
Redo the steps for further views.
To insert a new view in between the list select a view and with the command ***Insert new camera view (Ctrl+I)*** the new view will be

inserted before the selected view.
To delete a view from the list, select the view and press the ***Delete*** key or select the command ***Delete*** from the context menu.

3. To rename a view use the command ***Edit (F2)*** from the context menu.
4. In the ***pause time*** column it can be defined, how long a view should be shown during a camera animation. Click into the field and press the ***F2*** key to edit the value.
5. The ***animate time*** defines the time between leaving the current view and start the next view in the list. Click into the field and press the ***F2*** key to edit the value.
6. To process the presentation the following commands are offered:

	Starts the camera animation
	Pauses the camera animation at the current position
	Stops the camera animation at the current position
	Replays the camera animation when reaching the final view.
	Replays the camera animation step by step, from one view to the next.

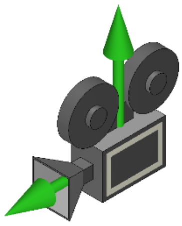
The camera animation mode always starts from the last selected view in the list. By double clicking a view in the list it can be activated immediately.

7. The current camera animation list can be stored and reused also in other models:
Click the icon ***Save camera animation*** if the current animation file should be stored using the current file name.
If the camera animation should be stored using a different name click the icon ***Save camera animation as ...*** and enter the file name in the following dialogue window.
8. Click the icon ***OK*** or ***Accept***, to store the current camera animation file with the model. When clicking the ***Cancel*** icon the changes made in the dialogue window will not be stored with the model.

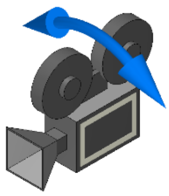
User defined Views in the 3D Window

In ***Presentation mode***, you save selected views in the 3D window so that you can select them again in any model from this dialog. In addition, you can now activate a saved view via an icon in a user-defined toolbar. There are two ways to do this. With the camera view straight, the inclination of the camera in the viewing direction is constant; with the camera view tilted, you can use the up vector to determine the angle of inclination.

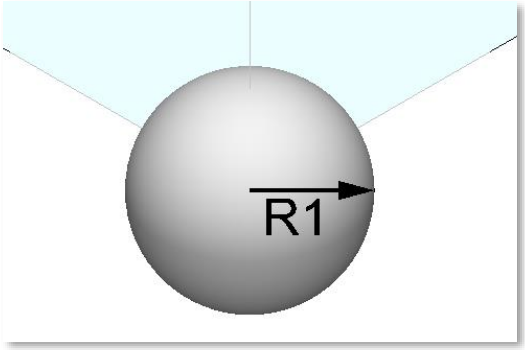
The coordinates for the current view direction and the up vector are displayed in the ***Presentation Mode*** dialog (***View*** menu, ***Presentation Mode***) and can be adopted for each individual camera view.

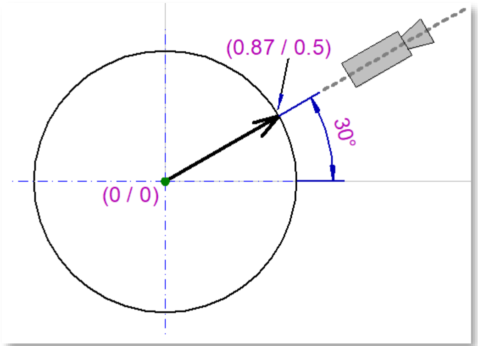
	Viewing direction	*Up-Vektor*	
 Camera view orthogonal:	**X Y Z**		

Example *(0.57735026919,0.57735026919,-0.57735026919)*

 Camera view tilted:	**X Y Z**	**X Y Z**	

Example *(0.57735026919,0.57735026919,-0.57735026919) (0,1,1)*





Please note:

The coordinates in the ***Presentation mode*** dialog for the viewing direction show the end point of a vector with length 1 whose starting point is at the absolute zero point. This vector determines the viewing direction. A viewing direction, starting from the view from the left, rotated by 30° to the right, results, for example, in the coordinates (0.87,0.5,0).

To activate a camera view from a custom toolbar, do the following.

1. Select a desired view in the 3D window.

2. Click the menu command ***View***, ***Toolbar***.

3. In the list of toolboxes, select the toolbox in which you want to insert the function.

4. In the folder ***View*** in the commands tree (on the right) you can find the commands

 ***Camera view orthogonal***

 ***Camera view tilted***

Select one of the commands and click ***<< Add*** or ***<< Insert ,*** to add the command to the selected toolbar.

5. Select the added command in the toolbar.
Enter the coordinates for the viewing direction in the ***argument*** input field (Function area) and the up vector for the tilted camera view. The values for X, Y and Z are separated by a space. Use a dot as decimal separator:

 Camera view orthogonal: **X Y Z**
Example *0.58 0.58 0.58*

 Camera view tilted: **X Y Z X Y Z**
Example *0.58 0.58 -0.58 0 1 1*

You can transfer the values from the ***Presentation Mode*** dialog one-to-one as described above.

6. In the ***remark*** field, enter a name for the view that will be displayed as a tooltip if the mouse pointer remains motionless over the icon for a short time.

3.8 Selection/deselection of objects

For a number of commands for processing existing objects (modify display attributes, transform, erase, etc.), you can determine whether the objects to be processed should be selected either before the command is started or after.

Both of these procedural possibilities are described as **selection - action** and **action - selection**.

Depending on the procedure selected, there are different procedural possibilities for the commands, e.g. for moving objects:

If following the **action - selection** principle, the move vector should first of all be determined. Then the objects should be selected. In this way, the move is carried out practically statically. The selected objects are only taken over in an action list for the duration of the command execution.
If following the **selection - action** principle, accept the objects in the selection list. After determining the start point of the move vector, the selected objects can be moved dynamically. The objects remain in the selection list until they are removed from it.

Selection - action

- With this procedure, the objects to be processed are taken over into the **selection list** or removed from it before starting the respective command.
  Objects which are contained in the selection list are highlighted. Double elements at the same or a very near position are listed in a text box when identified. The correct element can be choosen within the text box. By moving the mouse pointer in the box the related elements are marked in the drawing in the selection colour.
- Then load the command with which all the objects in the selection list are to be processed.
- After executing the command, the objects remain in the selection list until you remove them. This is not valid for the command ***Erase object/selection***

Action - selection

- This procedure can only be used if the selection list is empty.
- Start the command.
- If only one object is to be processed, identify it using the cursor. Double elements at the same or a very near position are listed in a text box when identified. The correct element can be choosen within the text box. By moving the mouse pointer in the box the related elements are marked in the drawing in the selection colour.
  If a number of objects are to be selected, take them over in the **action list**. The objects taken over in the list are highlighted. No objects can be removed from the action list. This happens automatically once the command is quit.

Please note:

-  Using the cursor in the 3D window, only 2D objects and/or 3D objects can be selected - depending on the activated 2D mode and/or 3D mode.
- Using the cursor, only objects can be selected of the type which are linked to an active layer and which have been activated in the object filter.
- 2D objects can be selected in the active WP when you are in the 3D view.
- Selected objects are highlighted in the colour which has been determined as the ***selection colour for 2D/3D objects*** using the menu command ***Settings, Options***.
  After you have selected groups, their objects will be displayed in the ***Selection colour for groups***.

Determine settings for the identification of objects using the cursor

If objects are to be identified using the cursor, as an optical aid, you can allow the objects that are touched by the cursor to be highlighted in colour.
The setting ***Activate cursor highlight*** can be activated using the menu command ***Settings, Options*** on the ***Standard settings*** option card.

- Using the menu command ***Settings, Object Filter***, object types can be determined which are not to be active, and are therefore not accessible for identifying with the cursor.

- Using the ***Layer structure tree*** inside the ***Model Explorer*** you can determine that layers are not active, therefore, objects which are linked with these layers cannot be identified.
- In the 3D window, in the main toolbar using the 2D/3D mode, you can determine whether 2D objects and/or 3D objects are to be active:
  If one of the two is not active, the respective objects cannot be identified using the cursor.

  This setting applies only to the commands that can be used for selecting both 2D objects and 3D objects. In the case of commands that can be used only with 2D objects or only with 3D objects they will have no effect.

If layers, groups or allocated attributes are used to select or deselect objects, the above mentioned limitations are not valid.

Determine settings for object filter

Using the menu command ***Settings, Object Filter**,* you can determine which object type should be permitted for selecting or deselecting **with the cursor**.

Active object
By tagging the switch, you may determine which categories and/or object types should be active for selection or deselection by means of the cursor. In this way, you can decide whether entire groups, e.g. all 2D line objects are to be active or not, or whether only individual objects e.g. only lines or circles are to be selected or not. Untagged categories/object types cannot be selected or deselected by means of the cursor.

Activate all objects
If this setting is active, all objects – independent of individual settings for the object filter- can be selected or deselected by means of the cursor.

Tag all
Tags all object types whose groups are active.

Untag all
Untags all object types whose groups are active.

Selection if Ambiguous
When several objects were found while identifying in a command, the objects were listed. By identifying the objects in the list, the geometric objects are marked by colour and so the desired object

can be choosen by a mouseclick.
When this option is deactivated, always the object will be found, which is nearest to the cross hair cursor inside the red snap box.

Copy mode for objects

Only copy selection

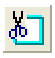
If you copy objects (lines, circles, text, etc.) with this presetting, all copied objects are saved in the active group. If the originals lay in different groups, the copied objects then lie in one group.
If however, you copy groups (incl. subgroups), new groups with the same descriptions are generated on the same structure plane. A group structure remains.

Also copy subordinate groups for each object

If an object lies in a group, this setting will not only copy the object but also the group in which the object(s) lies. The new groups are generated on the same structure plane as the originals and the group structure is kept.
For each copied group, a new group with the same description and structure is also set up.

Also copy complete tree for each object

If you use this setting to copy an object from a group, a new group for the copy is generated on the same structure plane.
However, if a number of objects are copied from different groups with this setting, all the objects within the subordinate groups in the hierarchy are also copied up to the common node point.

Selection mode: Clip 

With this setting, the reaction of certain types of 2D objects may be influenced by activating the ***Selection mode***.

Single object (extend)

This setting influences the selection of circles, ellipses and splines with the command ***Process 2D Objects, Extend***:

Setting	**Result**
Active	Moved if lie entirely or partially in defined window
Inactive	Moved only if lie entirely in defined window

Complex objects (extend, transform, erase)

These settings influence the selection of dimensions, standard parts, model views, section views, symbols, text, fit dimension tables, geometry calculations for the 2D commands: ***Extend, Move, Rotate, Mirror, Scale*** as well as ***Erase object/selection***.

Setting	Result
Active	Considered if lie entirely or partially in defined window
Inactive	Considered if lie entirely in defined window

Please note:

The settings in the Object filter have priority over the settings for the ***Clip selection mode*** .

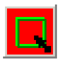
If it is only the 2D or 3D modes which are activated in the view window, then the settings of the object filter will take priority i.e. 2D or 3D objects may be selected/deselected by means of the cursor.

The settings for the object filter are not taken into account if objects are selected or deselected using means other than the cursor, e.g. using groups or layers.

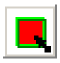
Selection mode

The command ***Clip selection mode*** at the upper program screen was extended. It affects all 2D line objects (e.g. Lines, Circles, Ellipse, Splines), which are selected by defining a window, e.g in the commands ***Reselect Object, Erase Object / Selection*** and also in the commands ***Transform*** / ***Move dynamically*** und ***Move with copy dynamically***..
The last choosen definition will be saved if the program is left and restarted.

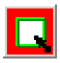
The following definitions can be made:

 **Selection mode: All**

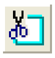
All 2D line objects touched or inside a selection by window are completely selected.

 **Selection mode: Inside**

Only those 2D line objects are selected which are completely inside the selection window.

 **Selection mode: Outside**

Only those 2D line objects are selected which are completely outside of the selection window.
Important:
Using this definition all the elements outside of the window are selected and maybe deleted if you choose so, even those elements which maybe are outside of the visible area of your model.

 **Selection mode: Clip**

2D line objects are clipped at the frame of the selection window. The Clip frame is shown in a green rectangle. The following objects cannot be clipped:

Dimensions, Hatches, Symbols, 2D standard parts, finish characteristics, Form- and Position tolerances, welding symbols Using this command a hatch can only be deleted by mouse click not by selection window.

Select/deselect using partial drawings

In the 2D drawing window, 2D objects can be generated in separate work areas, so called partial drawings (please see the chapter entitled ***2D Commands, Drawings and partial drawings***).

If, in the 2D drawing window, 2D objects are selected or deselected with one of the commands described in the following, you can previously determine in which partial drawing the selection/deselection is to take place.

When you use the cursor to select/deselect, one of the following icons in the icon bar of the 2D window shows the current status of the selection options.

<u>Select/deselect in the active partial drawing</u>

 If you have decided to select only inside the active partial drawing, activate this setting by clicking the icon.

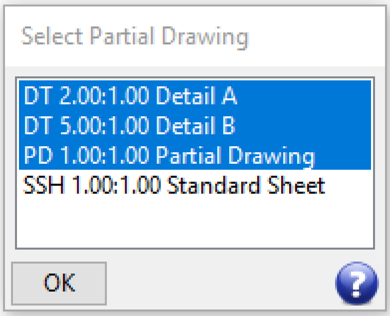
<u>Select/deselect in all partial drawings</u>

 If all partial drawings are to be taken into consideration for the selection, activate this setting by clicking on the displayed icon.

<u>Select/deselect in selected partial drawings</u>

 When the selection procedure is to be restricted to the selected partial drawings, activate this setting by clicking on the displayed icon.

Using those in connection with and also activated icon , you open a list in which you can select partial drawings you wish to be able to access during the selection procedure, e.g.:



Tag all the partial drawings that are to be taken into consideration during the selection procedure by clicking them with the cursor.
You can remove the tag from a partial drawing by clicking a second time.

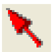
Please note:

In the 3D window the 2D objects are drawn on workplanes. The selection/deselection procedure of this object can be completed only in the active workplane.

Reselect objects

 After selecting this icon, the ***Reselect Object*** command is started.

This command removes all objects from the selection list.

 If a selection exists, the icon for this command is presented in red to prevent any objects selected outside the visible screen section to accidentally be manipulated or erased.

Afterwards you can use the following options to insert objects into the selection list or to remove them.

1. Provided that the 2D drawing window is active, specify the partial drawings to be considered in the selection procedure by typing an entry in the dialog box.

2. Next select one of the options listed below for including objects in the selection list:

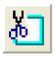
   Individual

   Identify each object separately using the cursor. Double elements at the same or a very near position are listed in a text box when identified. The correct element can be choosen within the text box. By moving the mouse pointer in the box the related elements are marked in the drawing in the selection colour. The list view can be deactivated in ***Settings, Object Filter*** in the option ***Ask for objects if selection is ambiguous***.

Using window

Using the cursor, determine a corner of the window.
Keep the left mouse button pressed and drag the window so that it contains all the required objects.
Confirm the size of the window by releasing the left mouse button.

Using window with clip (2D object)

 Activate the ***Selection mode:Clip*** in the main toolbar.

Define the window in the same manner as described above.

You may also determine it constructively:
Via the middle mouse button open the Snap Menu for both corner points and define them using the available commands.

When reloading the command, the selection mode will be reactivated in its previous state.

With this form of selection, you may select parts of 2D line objects for the commands ***Extend, Move, Rotate, Mirror, Scale*** and ***Erase object/selection***.
Deselection is not available.
In the 3D view window, it is only available if the top view of the current workplane will be displayed.

Please note:

> With ***Settings, Object filter***, you may, in addition, determine the level of performance for certain types of 2D objects (e.g. circles, standard parts, text).
>
> If a selection list is already available, this will be reduced by the definition of a window.
>
> Finally, objects that were available in the window definition will be selected.

All the objects in a group

You can select all the objects in a group and its subgroups by keeping the ***Shift key*** pressed down while identifying if the ***Standard Selection*** is active.

If the ***Group Selection*** is active a context menu appears when identifying an object with the pressed ***Shift*** key. In this menu the following can be selected:

- **only the objects in the group and in all subgroups (*Elements of:)***
- **the group and all subgroups**

- **the next higher group(s) in the structure and all subgroups**

<u>**Group with objects**</u>

You can not only copy or remove the objects in a group and the group itself: you can also select groups:
When you keep the ***Alt key*** pressed down while selecting you will automatically select the group of which this object is a member if the ***Standard Selection*** is active. If the ***Group Selection*** is active a context menu appears when identifying an object with the pressed ***Shift*** key. In this menu the following can be selected:

- **only the objects in the group and in all subgroups (*Elements of:)***
- **the group and all subgroups**
- **the next higher group(s) in the structure and all subgroups**

When you select any groups, the colour used to indicate the objects in a particular group will be the one that has been specified in the ***Settings, Options*** as the ***Selection colour for groups***.

This selection can also be carried out using the ***Model Explorer***.

3. If the selection list already includes objects, they will be deleted from the list by using one of the methods described above.

After you have selected a group, the only way that you can remove it from the list is by identifying an object in it and pressing down the ***Alt key*** at the same time.

Select/*deselect* using the cursor

When you want to add single objects or objects partly inside a rectangular window to a selection list or to remove them, use the following method:

1.  Click the two icons shown here one after the other or select the ***Process, Select/Deselect object*** menu command.

2. Provided that the 2D drawing window is active, a dialog box will appear so that you can specify which of the partial drawings are to be considered during the selection/deselection procedure.

3. Next, select one of the options described below to add objects to the selection list:

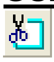
Individual

Identify each individual object using the cursor. Double elements at the same or a very near position are listed in a text box when identified. The correct element can be choosen within the text box. By moving the mouse pointer in the box the related elements are marked in the drawing in the selection colour. The list view can be deactivated in ***Settings, Object Filter*** in the option ***Ask for objects if selection is ambiguous***.

Using window

Using the cursor, determine a corner of the window.
Keep the left mouse button pressed and drag the window so that it contains all the required objects.
Confirm the size of the window by releasing the left mouse button.

Using window with clip (2D objects)

 Activate the Selection mode: Clip in the main toolbar.

With this form of selection, you may select parts of 2D line objects for the commands ***Extend, Move, Rotate, Mirror, Scale*** and ***Erase object/selection***.
Deselection is not available.
Define the window in the same manner as described above.

When reloading the command, the selection mode will be reactivated in its previous state.

Please note:

> With ***Settings, Object Filter***, you may, in addition, determine the level of performance for certain types of 2D objects (e.g. circles, standard parts, text).

All the objects in a group

You can select/select all the objects in a group and its subgroups by keeping the ***Shift key*** pressed down while identifying. This action will not include the group itself if the ***Standard Selection*** is active.

If the ***Group Selection*** is active a context menu appears when identifying an object with the pressed ***Shift*** key. In this menu the following can be selected:

- **only the objects in the group and in all subgroups (*Elements of:)***
- **the group and all subgroups**

- **the next higher group(s) in the structure and all subgroups**

<u>Group with objects</u>

When you keep the ***Alt key*** pressed down while selecting you will automatically select the group of which this object is a member if the ***Standard Selection*** is active. If the ***Group Selection*** is active a context menu appears when identifying an object with the pressed ***Shift*** key. In this menu the following can be selected:

- **only the objects in the group and in all subgroups (*Elements of:)***
- **the group and all subgroups**
- **the next higher group(s) in the structure and all subgroups**

This selection/deselection can also be carried out using the ***Model Explorer.***

4. If the selection list already includes objects, they will be deleted from the list by using one of the methods described above.

After you have selected a group, the only way that you can remove it from the list is by identifying an object in it and pressing down the ***Alt key*** at the same time.

Selecting objects with the cursor

This command corresponds to the ***Select/Deselect with the cursor*** command. The difference is that, objects are only taken over into the selection list.

Deselect objects with the cursor

This command corresponds to the ***Select/Deselect with the cursor.*** The difference is that, objects are only removed from the selection list.

Select/Deselect Objects by Polyline

To select/deselect objects by a closed polyline, please follow this stepps:

1. Select the desired Selection mode.

Please note:

Contrary to the selection in a window the objects are not clipped using the selection mode *Clip*. In this selection mode all the objects are selected which are completely or partially outside of the polyline.

2. , Click the displayed icons ***Select***, ***Select by polyline*** to insert objects from the current partial drawing into the selection list.

   , Click the displayed icons ***Select***, ***Deselect by Polyline***, to delete objects from the selection list in the current partial drawing.

3. Create a polyline consisting of two elements minimum by point definition or by entering the values in the status dialogue window. The start point of the first segment is joined with the end point of the last segment as a rubber bend. In every case a closed polyline is created.

4. After defining the last segment quit the command by clicking the right mouse button.

When selecting objects, the relevant objects are marked in the colour defined in ***Settings***, ***Options*** in the tab ***General Options*** as ***selection colour for 2D/3D objects***.

Please note:

To delete all objects from the selection list, also the command ***Reselect Objects*** can be used.

To detect whether a symbol is inside the polyline or not, the box is used, displayed when placing the symbol.

Select/deselect all objects

By clicking on the displayed icons in the Symbol toolbar the commands ***Select***, ***Select All*** or ***Deselect All*** are started.
Also the commands in the menu ***Process***, ***Select All*** or ***Deselect All*** can be used.

2D Window
The current setting to access partial drawing is being considered using the commands ***Select All*** and ***Deselect All***.

3D Window
All 2D objects in the current workplane are selected or deselected.

Please note:

In order to remove all objects from the selection list, the command ***Reselect Object*** can also be used.

Select/deselect 2D objects using layer

You can include 2D objects linked to specific layers in a selection list or remove them by using the following method:

1.  Click the two icons shown here one after the other, if you are going to select objects.

 Click the two icons shown here one after the other, if you are going to select objects.

2. In the dialog box for layer selection, tag the layer whose objects are to be selected/deselected.

3. Double click the marked layer or click on the ***Accept Selection*** command button in order to execute the action for the selected layer.
Repeat steps 3 and 4 for further layers.

Please note:

Keep the ***CTRL*** key pressed to execute the opposite command.

4. Confirm the selection of layers by pressing ***OK.***

Note:

To remove all objects from the selection list also the command ***Reselect Object*** can be used.

In the commands ***Select Using Layer*** and ***Deselect Layer*** the current access to partial drawings is being considered.

Select/Deselect by Colour

To insert 2D line objects, texts and solids with identical colours into the selection list, please follow these steps:

1.  Click the displayed icons after each other ***Select, Select by colour***, to select objects.

 Click the displayed icons after each other ***Select, Deselect by colour***, to deselect objects.

2. Identify a 2D line object or a text.

Please note:

To erase all objects from the selection list you can also use the command ***Reselect objects***.

In the commands ***Select by colour*** and ***Deselect by colour*** the current access to partial drawings will be considered.

Select/deselect groups with objects

You can not only copy or remove the objects in a group and the group itself you can also select and deselect groups.
Use the following method:

1. Click the two icons shown here one after the other, if you are going to select groups.

Click the two icons shown here one after the other, if you are going to remove groups from the selection list.

2. Identify an object that belongs to the group that you are going to select or deselect.
If the object belongs to a group that includes subgroups, these and the objects they contain will be selected or deselected as well.

After you have selected groups, their objects will be displayed in the ***Selection colour for groups***, which is defined via ***Settings, Options***.

Please note:

If the ***Standard Selection*** is activated (setting in the main toolbar) the group is selected which contains the identified object.

If the ***Group Selection*** is activated also the groups in higher structures are listed.

To remove all objects from the selection list also the command ***Reselect Object*** can be used.

Selection/deselection of 2D objects using attribute

Using the command ***Select, Select Using Attribute*** or ***Deselect Using Attribute***, 2D objects which have been allocated attribute values can be taken over into the selection list or removed from it.

The description of the procedure can be found in the chapter *2D* ***Commands, Use attributes***.

Please note:

To remove all objects from the selection list also the command ***Reselect Object*** can be used.

Select/Deselect Objects by Face

Using this command objects can be selected/deselected which are placed inside or outside of a 2D Face. Please follow this steps:

1. Select the desired Selection mode.

Please note:

In the selection mode Clip objects are selected which are completely and partially inside or outside the face.

2. ,  Click the displayed icons ***Select***, ***Select by Face*** to insert objects in the current partial drawing into the selection list.

,  Click the displayed icons ***Select***, ***Deselect by Face***, to delete objects from the selection list in the current partial drawing.

3. Identify a 2D face.

When selecting objects, the relevant objects are marked in the colour defined in ***Settings***, ***Options*** in the tab ***General Options*** as ***selection colour for 2D/3D objects***.

Please note:

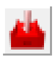
 To delete all objects from the selection list, also the command ***Reselect Objects*** can be used.

To detect whether a symbol is inside the polyline or not, the box is used, displayed when placing the symbol.

Add selection list

 By clicking on the displayed icon in the icon toolbar, the command ***Select, Add Selection List*** is loaded and the current selection is saved for later use. Clicking again sets up further selection lists to be used later.

Remove selection list

 By clicking on the displayed icon in the icon toolbar, the command ***Select, Remove Selection List*** is loaded.
This rejects the selection list last set up.

Change between selection lists

 By clicking on the displayed icon in the icon toolbar, the command ***Select, Last Selection List or Select, Next Selection List*** is loaded. The command makes it possible to change between the saved selection lists.

Remove set up selection lists

 By clicking on the displayed icon in the icon toolbar, the command Select, Remove Set Up Selection Lists is loaded. The command makes it possible to reject all selection lists that have been set up.

Accept objects into action list

When you are going to insert objects into an action list so that they can be used when a command is executed, use the following method:

1. Load the command that you want to execute, and then specify any parameters that may be required.
2. Provided that the 2D drawing window is active, define using access to partial drawing in the 2D toolbar which of the partial drawings are to be considered when selecting / deselecting.
3. Finally, select one of the options described below to insert the objects in the action list:

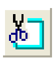
   Individual

   Identify each individual object using the cursor. Double elements at the same or a very near position are listed in a text box when identified. The correct element can be choosen within the text box. By moving the mouse pointer in the box the related elements are marked in the drawing in the selection colour. The list view can be deactivated in ***Settings, Object Filter*** in the option ***Ask for objects if selection is ambiguous***.

   Using window

   Using the cursor, determine a corner of the window.
   Keep the left mouse button pressed and drag the window so that it contains all the required objects.
   Confirm the size of the window by releasing the left mouse button.

   Using window with clip (2D objects)

    Activate the ***Selection mode:Clip*** in the main toolbar.

With this form of selection, you may select parts of 2D line objects for the commands ***Extend, Move, Rotate, Mirror, Scale*** and ***Erase object/selection***.
Deselection is not available.
Define the window in the same manner as described above.

When reloading the command, the selection mode will be reactivated in its previous state.

Please note:
With *Settings, Object Filter,* you may, in addition, determine the level of performance for certain types of 2D objects (e.g. circles, standard parts, text).

<u>All the objects in a group</u>

You can select all the objects in a group and its subgroups by keeping the ***Shift key*** pressed down while identifying if the ***Standard Selection*** is active. The group itself is not selected. If the ***Group Selection*** is active a context menu appears when identifying an object with the pressed ***Shift*** key. In this menu the following can be selected:

- **only the objects in the group and in all subgroups (*Elements of:)***
- **the group and all subgroups**
- **the next higher group(s) in the structure and all subgroups**

<u>Group with objects</u>

When you are using commands for copying or for removing objects, you can identify the group itself by pressing the ***Alt key*** while identifying an object.

<u>Selection by collecting</u>

The two procedures described immediately above can be combine to collect objects:

1. Press the ***Ctrl key*** and keep it pressed.
2. Select the objects either individually or by using a window.
3. Release the ***Ctrl key***.
4. Confirm the objects which have been selected/deselected by pressing the left mouse button or lose them by pressing the right mouse button.
   If you identify an object now that you have not identified up to now, the object itself will also be selected.

3.9 Redo and undo commands

If, when executing commands, the wrong objects have been processed or incorrect values have been accidentally entered, these commands can be undone step-by-step to a predetermined number of steps or redone to display the results required. The number of commands that can be undone can be determined using the menu command ***Settings, Options*** in the option card ***Save and Recover***.

The undoing of commands is not valid for modifications of settings or saving files.

Determine settings for undo/redo

Select the menu command ***Settings, Options***.
In the text box ***Save and Recover*** the number of processing states can be determined which can be undone step-by-step and redone if necessary.

Please note that a high number is linked with a large amount of memory.

Undo commands

 This command can be loaded using this icon in the main toolbar.

In order to undo the last command, select the menu command ***Process, Undo***.

Repeat the menu command if previous steps are also to be undone.

The command is not valid for loading or saving files.

Please note:

With the menu command ***Settings, Options*** in the text box ***Save and Recover*** you are able to define how many steps may be restored.

Redo commands

 This command can be loaded using this icon in the main toolbar.

In order to redo the last command, select the menu command ***Process, Redo***.

If a number of commands have previously been undone, repeat this menu command if these undo steps are to be redone again.

Please note:

With menu command ***Settings, Options*** in the text box ***Save and Recover*** you are able to define how many steps may be restored.

Erase undo list

The processing state of the model is protocolled in an "undo list". The model can be redone using the same list.

In order to reduce the memory required when processing extensive models, the list can be erased using the menu command ***Process, Erase Undo List*** and be made smaller using the menu command *Settings, Options, Save and Recover* and reducing the value in the *save actions for undo* text box.

3.10 Erase objects

The following described commands are available for erasing 3D and 2D objects.

Please note:

Objects are erased without any prompt appearing.
If objects have been erased accidentally, they can be retrieved again using the menu command ***Process, Undo***.

This icon  in the main top toolbar can also be used to retrieve the erased object.

Erase object/selection list (Del)

 Alternatively, the command can be loaded using the menu command *Process* or in the CAD Menu using the *Object/Selection* command under the *Erase* heading.

The following possibilities are available for erasing objects:

Erase with selection - action
If the selection list contains objects, these are then erased once the command is loaded. This command remains active afterwards so that you can delete further objects. Quit the command with the right mouse button.

Erase with action - selection
If the selection list is empty, identify the individual objects or accept the respective objects in the action list.

Please note:

Toolbar button “undo“ . This can be used to undo the commands carried out last.

Erase window

To partially delete only those parts from 2D line objects which are inside or outside a defined rectangle the command ***Erase Object/Selection*** can be used.

1. The following definitions can be made:

Selection mode: All
All 2D line objects touched or inside a selection by window are completely selected.

Selection mode: Inside
Only those 2D line objects are selected which are completely inside the selection window.

Selection mode: Outside
Only those 2D line objects are selected which are completely outside of the selection window.
Important:
Using this definition all the elements outside of the window are selected and maybe deleted if you choose so, even those elements which maybe are outside of the visible area of your model.

Selection mode: Clip
2D line objects are clipped at the frame of the selection window. The Clip frame is shown in a green rectangle.
The following objects cannot be clipped:

Dimensions, Hatches, Symbols, 2D standard parts, finish characteristics, Form- and Position tolerances, welding symbols
Using this command a hatch can only be deleted by mouse click not by selection window.

2. Define one – or with the pressed Ctrl-key – multiple windows, which should be erased:
Define one endpoint of the window with the left mouse button and keep it pressed to define the other endpoint.

Please note:

With ***Settings, Object filter*** you may in addition determine the level of performance for certain types of 2D objects (e.g. standard parts, text).

Delete Double Objects

Using the menu command **Process**, **Delete Double Objects**

- Identical curves, on the same layer with identical viewing properties and exactly on top of each other
- double symbols placed on each other

are detected and can be erased from the current drawing / workplane.

Please note:

Associative objects, e.g. for dimensions, hatches and so on, can also be deleted using this command.

Erase group with objects

 Alternatively you can use the menu command ***Process, Erase object/selection, Remove object/selection*** or in the CAD using ***Object / Selection*** beneath the heading ***Erase***.

When you are going to erase a group, then the subgroups and the objects in it, you can use one of the following methods:

Erase with Selection - Action
If the selection list contains groups, they, their subgroups and the objects they contain will be erased as soon as you load this command.

Erase with Action - Selection

If the selection list is empty, press the ***Alt key*** and identify an object in the group at the same time. If the ***Group Selection*** is active a context menu appears when identifying an object with the pressed ***Shift*** key. In this menu the following can be selected to be deleted:

- **only the objects in the group and in all subgroups (*Elements of:)***
- **the group and all subgroups**

- **the next higher group(s) in the structure and all subgroups**

Please note:

The 'Undo' icon 

This icon in the main symbol bar can be clicked to undo the results of the most recent commands.

Erase active 2D section

In the 2D drawing area, an active 2D section is the active partial drawing, in the 3D view window, it is the currently selected workplane.

In order to erase all 2D objects and the group structure contained in the active 2D section, select the menu command ***Process, Erase Active 2D Section*** or, in the CAD Menu, the ***Active 2D Section*** command under the ***Delete*** heading. This command remains active afterwards so that you can delete further objects. Quit the command with the right mouse button.

Please note:

Toolbar button “undo“
This can be used to undo the commands carried out last.

3D objects are not affected by this command since they are independent of the workplane.

Erase entire model

If all objects and groups are to be erased from the model, select the menu command ***Process, Delete All 2D/3D Objects***.

Please note:

Toolbar button “undo“
This can be used to undo the commands carried out last.

Erase MOD file

A MOD file cannot be erased using the ***File, Open Model*** dialog box, since the drawing preview of the program accesses the file.

Erase MOD files you no longer require using an external program such as Windows Explorer.

For the case that you have activated automatic file backup using ***Settings, Options***, you should occasionally also erase these from time to time using an external program or save accordingly since these are not automatically erase by *BeckerCAD*.

3.11 Determine program settings

When firstly loading *BeckerCAD*, the program is started with the standard settings.

Since these surely only correspond to some of your requirements, they can be modified and saved at any time so that you have your own predefined settings available each time you start the program.

You should note the two different types of program settings:

- General settings
- Settings relevant to the models

Determine general settings

The general settings are those listed immediately below:

- The current position and size of the ***Dialog boxes*** that are used in the main program.
- The folders that can be defined using ***Settings, Folder*** for the saving of data. Information on this can be found in one of the following chapters.
- The units determined using the menu command ***Settings, Units***, used for the coordinate and length values. Information concerning this can be found in the chapters ***2D Commands, Model Units*** and ***3D Commands, Model Units***.
- The settings for the user interface defined using the menu command ***Settings, Options, General Options***, such as cursor highlight. Information concerning this can be found in one of the following chapters.
- The settings defined using the menu command ***Settings, Options, Settings Grafic Windows*** such as background colour or for the automatic move of an enlarged window over the current drawing window.
  The descriptions concerning this can be found in the chapter ***Determine program settings, Determine general settings, Determine settings for autopan/overview window***.
- The settings definable for the automatic saving of entry steps, models, backup copies and preview settings using the menu command ***Settings, Options, Save and Recover***. The descriptions concerning this can be found in the chapter

Determine program settings, Determine general settings, Determine save settings.

- The settings definable for the appearance of toolbars and menus as well as the format and appearance of the **Status dialog box** using the menu command ***Settings, Options, User Interface.*** The descriptions concerning this can be found in the chapter ***Determine program settings, Determine general settings, Determine settings for the user interface***.
- The toolbar definable by the user using the menu command ***View, Toolbar***. Information concerning this can be found in the previous chapter.
- The settings definable for the display of solids as well as lighting using the menu command ***Settings, 3D Display***. Information concerning this can be found in the chapter ***3D commands, Control solid display***.
- The presettings determined using the commands ***Drawing Additions***, ***Define Finish Characteristics*** and ***Define Form/Positional Tolerance***. Information concerning this can be found in the chapter ***2D Commands, Drawing additions***.

Use the following method to specify the settings and then to save them:

1. Start the program.

2. Load the menu command that opens the dialog box, the CAD menu or the Camera toolbar. Place the windows in a suitable position, specify their size and then close the windows again.

3. Next specify the other settings detailed above by means of the menu commands for them.

4. Exit the program.
Depending on which program has been started, the settings will now be saved in one of the file UIBECKER.SYS.

The file is always saved in the folder that is used as the working directory of the program. By default, this is the directory ..\USER\, which is located below the main directory of BeckerCAD.

Please note:

You may decide to start the program with **different settings** for the user interface. This can be done in the following ways:

- Before you start the program complete an additional link on the desktop and define a separate work directory for it. Then carry out the steps described above.

- Before starting the program change the link by adding a parameter in the ***Target*** text box to the command for loading the program, e.g.:
  `..\BECKERCAD.EXE –SYS:TEST`
  Then complete the steps described above. When you exit the program a file will be saved in the work directory using `TESTUIBECKER.SYS` as its name.

Determine general options

Using the menu command ***Settings, Options***, the type of display and identifier of specific objects can be determined as well as the number of possible undo steps.

When the program is quit, the current settings are saved in the file UIBECKER.SYS. This is also valid for the user defined colours which are determined in the colour selection dialog box.

selection colour for 2D/3D objects

This can be used to specify the colour in which the object in the selection list or action list is to be displayed.

Specify the colour using the same method as was used for specifying the background colour.

selection colour for inactive partial drawings

This can be used to specify the colour for displaying the selected 2D objects that the active partial drawing does not contain.

Specify the colour using the same method as was used for specifying the background colour.

selection colour for groups

This can be used to specify the colour for displaying objects belonging to a selected group.

Lowlightcolour

Allows the definition of the colour, in which 2D objects are displayed which, for example. are not in the active part / group. Also non active work planes in the 3D window and dimensions on model views, which have no link to the 3D object and are therefore not associative any more, are displayed in this colour.

Color for objects marker

Allows the color definition for object markers, e.g. for Geopoints shown on 2D curves or the snap point when Auto Snap is activated.

Cursor highlight

Allows for the support of 2D and 3D object identification by colour highlighting the objects over which the cursor is moved.

1. Activate the option for ***3D objects*** if the cursorhighlight should also work for 3D objects. Instead for 3D objects the ***face highlight*** can be uesed also.

In case of complex geometries the cursor starts to jerk, you can use the following parameters to modify the performance of the cursor highlight.

2. **Max. activation distance**
   The highlighting of objects is only activated if the distance travelled with the cursor in the model is shorter than the value defined here (in millimetres).
   Default: 30
3. **Activation speed**
   The highlighting of objects is only activated if the mouse cursor speed is lower than the value defined here. The smaller the value, the fewer objects are highlighted at normal cursor speed.
   Default: 0.2
4. Activate the settings using the control box at the start of the line.

Please note:

If access is activated (*access to defined partial drawings* or *access to all partial drawings*) on non-active partial drawings (status: read only) the objects are also highlighted.

3D-face highlight

This can be used to support the identification of solids, by colour highlighting the solid faces over which the cursor is moved.

1. Specify the colour using the same method as was used for specifying the background colour.
2. Specify how the solid faces are to be denoted either as ***Complete*** or by ***Edges***: select the option from the list.
3. Activate the settings using the check box at the start of the line.

Please note:

These settings have no effect on solids that are displayed using the option ***Without shading***.

2D Point marker

This can be used to select an option for denoting the points by means of a marker.
The display that is chosen will also be adhered to during printing. If the option is ***Without***, a point will be displayed and printed in the same pixel size.

geopoint marker

This can be used to specify for the ***Transform, Show Geo Points*** command in which the size of all the geopoints are to be displayed.

draw 2D lines with priority

This option can only be used when using an Open GL graphics card.

If active, the 3D polygons and 2D line objects which are drawn in the 3D window, will be displayed with priority.
This means, that if a workplane lies in a flat solid face, 2D lines which have been drawn in this workplane will not be covered by these solid faces.

Please note:

Errors could occur with the display.

display thick lines

If this check box is active, 2D line objects which have been allocated a line width, will be displayed in this line width - in relation to the set 2D drawing dimensions. It is independent of the selected scale.

If this check box is not active, all lines are displayed in the width of a pixel.

Independent of this display, for print output, lines are always displayed which are allocated themselves or using layers.

Please note:

 If you often switch between both of this display modes, you can controll it by a separate icon in a user defined toolbar, in a user defined menu or in user defined hotkeys.
The command ***Display thick lines*** is placed in the folder ***View*** (dialogue for user defined toolbars).
In the case you want to control this view mode using hotkeys the command name is: **`"view.toggleLineWidth"`**

Dimensions without a Reference in Model Views

Dimensions in model views, which have lost their reference due to chnages in the 3D model are faded out automatically. This system

setting assures, that after a change in the 3D model there are no wrong dimensions in a model view.
To display the dimensions with a lost reference in grey, this option must be activated.
The ***Lowlight Colour*** for dimensions without a reference can also be defined is this tab.

2D-face display

tagged
Clipped 2D-faces are displayed as transparent and are made visible by a point grid.
This setting can also be used to make all objects of type 2D-face visible with a point grid.

not tagged
Objects lying "behind" clipped 2D-faces are covered by the areas of these faces, therefore not displayed.

Scale line pattern with factor (Model specific setting)

This factor influences the pattern length of the Windows standard line types in the program: the larger the value, the longer the lengths of the lines as well as the distances between them.

This value is saved separately as model specific for each active drawing or for the 3D window. Preset values for the respective areas can be saved in your template

Erase 2D objects

If this option is active, the area in which an object or window has been previously erased, is redrawn after the defined time. The time area can be defined from a minimum 200 ms to a maximum 10000 ms.

Message, when activating invisible Groups, …

Using the Model Explorer partial drawings, work planes, groups and layers can be set to invisible. If one of this component is activated and new objects are created, they will not be displayed in the graphic window.
BeckerCAD supports you, displaying a note if a Partial Drawing, a Workplane, a Group or a Layer Geometry is activated, which has the status invisible.
If this message shouldn´t show up every time, please activate the option ***Don't show this message again***.
The other way round will display this message again, if you activate the option ***Message, when activating invisible Groups…*** in the menu command ***Settings, Options*** in the Tab ***General Options*** in the ***Miscellaneous*** area.

Determine Settings Grafic Windows

background colour

This can be used to specify the background colour of all 2D and 3D windows.

1. Click on the button next to this entry.
2. In the colour selection dialog box, determine the required background colour and confirm with ***OK***.

Depending on the background colour, the defined character colours are modified automatically so that the drawn objects are always visible.

Please note: You can swap between the background colours of black and white using the menu command ***Window, Black Background*** or ***White Background***.

Activate colour gradient in 3D windows: With this option the colour gradient for 3 windows can be activated or deactivated.

Gradient: With this value the transition between the choosen colours can be defined.

Colours: The gradient will be shown between the two defined colours.

Effect: Starting from the first colour the direction can be defined here.

Accept for all 3D windows: When the option ***Activate colour gradient in 3D windows*** is active the actual settings can be adjusted to all 3D windows.

Define colour gradient for each grafic area: Activate this option if you wish to take over the settings of colour gradient from one grafic area to another grafic window or parts of it.

Get parameters from current grafic area: When choosing this option the colour gradient is taken over from the current 3D window.

Apply parameters to current grafic area: When choosing this option the parameters are passed only to the current grafic window.
In divided 3D windows the parameters are only passed to the current area.

Apply parameters to current grafic window: When choosing this option the parameters are passed only to the current grafic window.

In divided 3D windows the parameters are passed to all the grafic areas.

Note:
Please be aware that the grafic performance can be lower with activated colour gradients.

Overview Window
For the display list area and for the zoom area presettings can be made for the colour and the frame width.
If the current zoom in the display area should be displayed in a filled mode activate the option ***Display current zoom area filled***.

Activate autopan
If this option is active, Autopanning can be used with the settings determined on this option card.

Horizontal/vertical frame width
These values can be used to determine the width of the scroll area (top and bottom/right and left), in which the cursor can be positioned in order for the Autopanning to start. This is valid for all windows.

Activation time
If the cursor is moved into a scroll area of the window, Autopanning is not started straightaway, but only once the cursor has been in the scroll area for the activation time entered here without being moved.
This value should be selected to a large enough time so that no unrequired move is carried out, e.g. if the cursor is simply being moved from the drawing into the toolbars.

The following settings can influence the speed of the move.

Repetition time
The move of the window is repeated, as long as the cursor is in the scroll area and it is not moved. This occurs after the repetition time is reached.
With a larger number of drawing elements, this value should not be selected too small.

If the cursor is moved, another move only occurs once the above defined activation time is reached.

Scroll width
With each move step, a move takes place by the value entered here. It is defined in a thousandth of the maximum scroll area.

Scroll factor

If the Ctrl key is kept pressed whilst moving the window, the scroll width is multiplied by the factor entered here.
In this way for example, you can move the window section quickly from the left border of the window to the right.

dynamic graphics (Time interval for dynamic redraw)

Determines, for all cases in which graphics must be permanently redrawn, when the drawing procedure should be cancelled and reinitialised. For dynamic operations such as moving or copying as well as inserting symbols or automatic moves (autopanning), this is valid for complete drawing sections.

Reverse the zoom direction of the mouse wheel

Changes the direction of the mouse wheel for dynamic zooming in and out of the view in the graphics windows.

Working Planes

Settings for Workingplanes

Determine save and recover settings

Save actions for undo/redo

This text box can be used to define the maximum number of work steps which can be undone or redone.

Please note that the work step protocol requires memory, i.e. the higher the number entered here, the greater the memory requirement.

Backup automatically

If this setting is active, the current status of the model or template is automatically set up in a backup file after a determined number of ***actions*** or ***minutes***.

The backup file has the name of the currently processed model or template which has the character # proceeding.
E.g. a file with the name #MODEL.MOD will be set up for the current model MODEL.MOD. If the number is >1 the save files will be named according the key: **#<model name>-#nn#.mod**.
The folder for the backup files (#*.MOD) can be specified by using the ***Settings, Folder*** menu command.

Generate drawing preview with automatic backup

If this setting is active, preview data is generated for each drawing contained in the model within the automatic file backup. If this option is deactivated, this will lead to speed advantages.

Create backup (*.bak) when overwriting models

If this setting is active, during a save procedure the previously saved file is backed up with the same file name in the same directory with the extension *.bak.
This option affects models, templates and backup copied which have been saved at least twice. If this option is active, they will only be created if the *number of auto save files* = 1. The creation of *.bak files in the work directory is not changed by this setting.
A *.bak file cannot be loaded from the Explorer using Drag&Drop. However, you can select the file types *.bak using the ***File, Open Model*** command. Then execute the ***File, Save As*** command, in order to reallocate the extension *.MOD to the model.

Number of Auto Save Files

With the setting *number of auto save files* it can be defined, how many auto save copies (#*.MOD) should be stored to a model in the current folder for the *auto save files*. If the number is >1 the save files will be named according the key: **#<model name>-#nn#.mod**.
The file with the lowest number is the oldest one, the file with the highest number the newest one. In the range of the numbers for auto save files, the files will be updated according to the settings „backup after ?? actions“ or „backup after ?? minutes“.

Save temporary file

The options available in this area can be used for optimum saving of files within the commands **Save** and **Save As**. In particularly when saving in networks, the use of a temporary save can lead to extensive speed advantages.

General options for file save

If the setting **delete existing references to symbol libraries when loading model** is active, the references to symbol libraries currently found in memory are erased. After the required model is loaded, only those libraries set up in the model are directly available for selection.

When saving models, if the option **generate drawing preview for separate *CADdy⁺⁺ Viewer*** is used, separate drawing preview data can be generated when using the commands **Save** and **Save As**. This data can then be read only with a special, separately attained ***CADdy⁺⁺ Viewer***. The Viewer itself is used to view drawings within a model file, without having to have a *BeckerCAD program* version installed. Therefore, e.g. as Viewer within the EDM systems.

Models including 2D data can be saved in a compressed format. This affects the file size on the hard disk. A high compression rate affects in small file sizes, but the time for saving is longer. The default value for the compression is 6. A lower value affects in a faster saving but bigger file size. Deactivate the option ***Save 2D data in a copressed format*** if the models should not be saved in a compressed format.

Background colour for drawing preview

Using the colour command button, you can define the background colour for the preview window in the ***File, Open Model*** dialog box. This setting is saved in the UIBECKER.SYS file after you quit the program.

Number of Recent Files menu entries

The list of the recent files is shown in the menu command ***File***. Click on an entry to load the model.

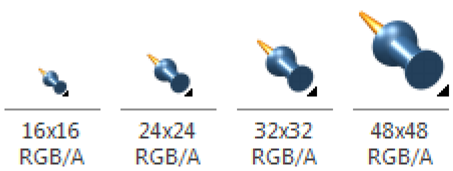
Determine settings for the user interface

On this option card, define the size of your icon in the icon bar, the settings for the Status dialog box and the Model Explorer, for Zoom window, Animated view on working plane / 3D face and for the context menus in the grafic area.

Pull-down menu and icon menu

Activate/deactivate the display of the icons in the pull-down menus. Modifications made here are only effective after the program is restarted.

The size of the icons in the toolbars and icon menus can be determined separately.
Setting options for the size of the
icons in the pull-down menu: 16 – 24 (pixels)
icons in the toolbar: 16 – 48 (pixels)
standard sizes: 16, 24, 32, 48



Status dialog box

Using the value in the ***Dynamic display exactness*** text box, you can define the decimal places to be displayed in the Status dialog box after you have activated a command and moved dynamically with the cursor into the drawing interface.

The value in the ***Static display exactness*** text box defined the decimal places to be displayed in the Status dialog box, if you position the cursor in a text box of the Status dialog box in order to enter a value.

Using the colour command buttons, select a colour which defines the background colour in the ***fields to be edited*** of the Status dialog box.

Activate/deactivate the ***transparent display*** of the Status dialog box in order to be able to recognised the objects lying behind the Status dialog box. The objects which lie behind the Status dialog box cannot however be picked.

The percentage ***degree of transparency*** is determined using the slider field.

Activate/deactivate whether the Status dialog box is to be displayed ***when entering a transparent value***.

Zoom Window

If you activate the ***Drag Window*** option, you can use the ***Zoom Window*** command to generate enlarged windows, i.e. you define the window you wish to enlarge, by dragging a rectangle around the respective area keeping the mouse buttons pressed. If the option is not active, the window to be enlarged is defined using 2 points which need to be determined in the drawing by mouse click.

Animated view on working plane / 3D face

Determines for the ***View, View On Face/WP*** command, whether the modification of the view is to be animated step by step.

The first value determines the number of steps for the animation, the second defined the time interval between the individual steps.

Model Explorer

The various colour command buttons can be used to set the colour defaults for the identifiers of the active group, partial drawing, 2D drawing, workplane, the active 2D/3D context, the active 3D workplane and the active 3D space.

Use model explorer to display mesage

If this option is activated, error messages or notes for diverse commands are signalled in the news area of the Model Explorer

and not displayed in the dialog boxes on the interface.
Example:
Activating a snap point using the keyboard (e.g. E for endpoint), without a command previously being started.

Focus objects

The following settings can be used to control the focussing of objects using the Model Explorer (context menu in the list view):

highlight colour for focussed objects
Select a colour using the colour command button to display objects marked in the model explorer.

activate colour highlight
If this option is activated, focussed objects will be highlighted in the drawing / 3D model in the colour set in the colour command button.

allocate with line thickness
If the ***colour highlight*** option is activated, the objects to be focussed can be identified by defining a special line width in the line thickness text box.

centered and enlarged display
If this option is activated, the object(s) to be focussed are displayed over the entire screen. The view is therefore enlarged or reduced.

Context menu

The setting made here only affects the allocation of the mouse buttons within the drawing area of *BeckerCAD*.
If the option is not active, the context menu is on the middle mouse button. The right mouse button is used to abort a command.
If the option is active, the middle and right mouse buttons are swapped. In this case, the context menu lies on the right mouse button and a command can be aborted using the middle mouse button.

Determine folder settings

In the menu command ***Settings, Folder*** it can be defined, in which folder the *BeckerCAD* files should be stored by default.

1. For every file type the list contains two fields:
   one with the file type name and one with the default folder.
   To define a new folder, double click the folder field or open the context menu with the right mouse button and choose the option ***Select a new folder (Ctrl+N)***.
   How to change several folders in one step is described below.

2. In the following dialogue window select a new folder for the file type and confirm with ***OK***.
3. To store the folder list, the desired folders must be activated. **Only those folders are saved, which are active ☑ .** To activate all folders use the keys ***Ctrl+A*** (or context menu: ***Select all***) and then the ***space bar***.
4. Select the button ***Save directory definition file*** , to store the settings in the current file or select the button ***Save directory definition file as...*** , to store the settings with a different name. Define the name and the directory for the parameter file and click the button ***Save***.
5. If changes were made, they can be confirmed with the button ***OK*** or ***Apply***. Click the ***Cancel*** button, if changes should not be used.

Edit the Folder List

Besides the above described possibility to create new folders, the entries in the list can be edited as follows. Please note, that changes can only be made, if the path and folder already exists.

- A directory can directly be changed in the column folder, by marking the field and select ***Edit (F2)*** from the context menu.
- To exchange a list of folders activate ☑ the file types to be exchanged first. Then select ***Replace marked items (Strg+F4)*** in the context menu. Enter the old partial folder name in the field ***Search*** und the new partial folder name in the field ***Replace*** always starting with the volume name . Confirm the ***Replace*** button for every folder or select ***Replace all*** to exchange all the marked folders in one step.
- If all folders should be exchanged by a new name first activate all file types. Then start the option ***Select all (CTRL+A)*** from the context menu and press the space bar. Select the option ***Replace (F4)*** and enter the old directory name in the field ***Search*** and the new directory name in the field ***Replace*** always starting with the volume name **.** Confirm the ***Replace*** button for every folder or select ***Replace all*** to exchange all the marked folders in one step.

Load a folder list

1. Click the ***Open directory definition MPS-file*** button .
2. Select the desired MPS file.
3. Click the ***Open*** button.
4. Confirm the new folders by clicking the ***OK*** or ***Apply*** button.

Please note:

When leaving the program the current settings for folders are stored in the file UIBECKER.SYS.

File types

#*.MOD	Save copies of models
*.DAT	CADdy-fonts
*.DXF/DWG	DXF/DWG- files
*.FNT	BeckerCAD fonts
*.ICO	Icons
*.INF	CADdy-Info files
*.LAY	CADdy-Layer files
*.MOD	Models
*.MPS	Parameter –files
*.PIC	CADdy-Picture files
*.PRN	Print / Plot files
*.SAT	ACIS SAT-files
*.SHX	SHX- fonts
*.SYB	CADdy symbols
*.SYL	Symbol libraries
*.TLW	User defined icon bar
*.TMP	Temporary files
*.TPL	Templates
PIX	Images
SSH	Library for BeckerCAD standard sheet symbols
TEX	Textures

Determine model specific settings

The model-related settings are those described immediately below. They will be saved in template files (*.TPL) and whenever you start on a new model you can select one of these template files. The file then transfers the settings it contains to the model.

The settings are:

- The 2D drawings set up with the ***Model Explorer*** , the dimensions and partial drawings that go with these drawings that you can specify with ***Settings, 2D Drawing***.
  Further information is to be found in the chapter ***2D commands, Drawings and partial drawings***.
- The current 2D or 3D window as specified in ***Window,*** including any divisions.
  Further information is to be found in the chapter ***Program control, General commands***.
- The layers specified by using the ***Layer structure tree*** inside the ***Model Explorer***, including the colours, line types and line widths for the 2D objects.
  Further information is to be found in the chapter ***2D commands, Control display of 2D objects***.
- The groups specified using ***Model Explorer*** for the 2D and 3D objects.
  Further information is to be found in the chapter ***2D commands, Using groups***.
- The attributes specified using ***Define, Attributes*** that are linked as additional information to the 2D objects.
  Further information is to be found in the chapter ***2D commands, Use attributes***.
- The symbol libraries loaded using ***Define, Symbol***.
  Further information is to be found in the chapter ***2D commands, Use symbols***.
- The line types specified using ***Define, User defined line types***.
  Further information is to be found in the chapter ***2D commands, Control display of 2D objects***.
- The specifications laid down in the ***Settings*** menu for
  Units
  3D-Display
  3D/2D Commands
  Gesture Recognition
  3D-Selection Mode
  Drawing
  Text
  Dimensioning
  Table Dimension
  Fit dimension table
  2D area calculations

Hatching
Multiple line
2D Standard Parts
Symbols
Pixel drawings
Drawing grid
Object filter
Point definition
Further information is to be found in the relevant sections of the chapter ***2D commands*** and ***3D commands***.

Use the following method to specify and then save the settings:

1. Start the program.

2. Select the ***File, New Template*** menu command if you are going to design a new template file on the basis of the current settings. Alternatively you can use ***File, Open template***, when you are going to modify the settings for an existing template file.

3. Specify the settings you will need in accordance with the list given above.

4. Select ***File, Save*** or ***File, Save As*** to save the new settings in a template file.
If the template file is given STANDARD.TPL as its name, the settings in it will be activated as soon as you start the program.

Please note:

STANDARD.TPL
This file determines the settings when you start *BeckerCAD*, when you close the start-up assistant.

Save/load parameter sets

Model specific settings can be modified at any time whilst processing a model or template, by loading one of the respective commands from the **Define** or **Settings** menu and executing the required modification.

Since this procedure can be very extensive when modifying a number of settings, e.g. for text, dimensioning, the menu command **Save/Load Parameter Sets** should be used in order to increase the speed of your work. This enables the current settings in files to be saved with the extension MPS (model parameter sets) and modify the current settings by loading such a file.

Save parameter sets

Using the menu command **File, Save/Load Parameter Sets** the current settings for the following topics can be saved:

Parameter:	**Settings from menu:**
3D Import/Export	File, Import and Export, Settings.
CADdy Converter Parameters.	File, Import, *CADdy* PIC-File
Camera animation parameter	View, Presentation Mode
Dimension Parameters	Settings, Dimension
Directory parameters	Settings, Folder
Edge symbol parameter	2D Toolbar, Drawing Additions
Font Parameters	Settings, Text, Font Selection
Form / Pos. parameter	2D Toolbar, Drawing Additions
Grid Parameters	Settings, Drawing Grid
Hatch Parameters	Settings, Hatch
Layer Parameters	View, Model-Explorer, Layer Explorer
Lighting parameter	Settings, 3D Display, Tab: Lighting
Multi Line Parameters	Settings, Multi line
Plot parameters stored in modell	File, Print
Plot parameter stored in Ui	File, Print
Point Definition Parameters	Settings, Point Definition
Referenced Text / Position Number Style Parameters	Settings, Text, Referenced Texts / Position Numbers
Referenced Text Parameters	Settings, Text, Referenced Texts / Position Numbers
Sectional View Parameters	Settings, 3D-/2D-Commands, Section View
Surface Parameter	2D Toolbar, Drawing Additions
Symbol Parameters	Settings, Symbol
Table Dimension Parameters	Settings, Table Dimensioning
Text Style Parameters	Settings, Text

1. Tag the settings that are to be saved, by clicking the control box at the start of the respective lines.
With the commands, known from the Windows-Explorer® to tag several elements, e.g. by defining a window or collecting of multiple elements with the pressed Ctrl key or elements between with the

pressed shift key multiple lines can be tagged and be activated / deactivated by pressing the space bar.

To activate / deactivate all parameters the commands in the context menu on the right mouse button or key combinations can be used:

Activate all = Ctrl+M

Deactivate all = Shift+Ctrl+A

2. In the dialog box, click on the icon **Save**, in order to save the file under the name that is displayed in the neighbouring text box, or Click on the icon **Save As**, and in the dialog box which then opens , enter the file names under which the settings are to be saved.

You define the directory for MPS files under ***Settings***, ***Folder***.

Load and accept parameter sets

Using the menu command **File, Read/Write Parameters** the settings saved in a MPS file can be loaded and all or parts of these settings can be taken over as the current parameters.

1. In the dialog box, click on the icon **Open**.
2. In the selection dialog box which then opens, tag the required MPS file and click on the **Open** command button.
   Parameters loaded from an MPS file are marked by the symbol.
3. If necessary, erase the tags on the settings that are not to be taken over. With the commands, known from the Windows-Explorer® to tag several elements, e.g. by defining a window or collecting of multiple elements with the pressed Ctrl key or elements between with the pressed shift key multiple lines can be tagged and be activated / deactivated by pressing the space bar.

To activate / deactivate all parameters the commands in the context menu on the right mouse button or key combinations can be used:

Activate all = Ctrl+M

Deactivate all = Shift+Ctrl+ M

4. Click on the button ***Accept***, if the dialog box is to remain open or

   Click on the **OK** command button if the dialog box is to be closed after the command is executed.

Please note:
With layers and drawing grids, only the settings of existing

structure elements are modified, i.e. no layers or grids are removed or added.
This means that the descriptions of the layers and drawing grid in the current file must be the same as those in the MPS file.

3.12 Work with models and templates

After you have started the program, an empty model is available in which solids and/or 2D objects can be generated, processed and saved.

The template file that contains the settings made using the method described in the previous chapter serves as the basis for your model.

Further information on the relationship between templates and models and a description of the associated menu commands is to be found in the following sections.

Please note:

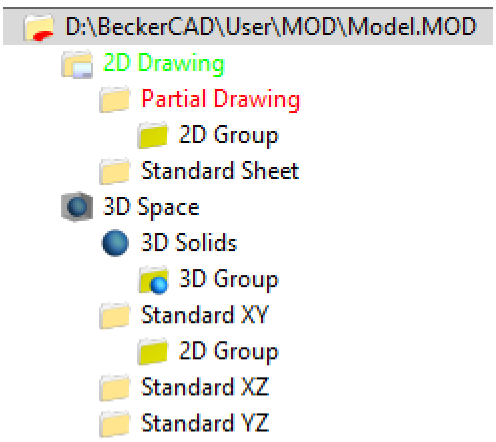
BeckerCAD models and templates are protected against simultaneous access from different workstations. An unintentional parallel processing of the files is therefore not possible.

When opening a Model or a Template in *BeckerCAD*, an owner file with the same name is created in the same directory. If another computer tries to open this file, an error message appears, in which the current user is displayed.

Model structure

All data that has been generated using *BeckerCAD* are managed in a model.

After you have started the program with the templat file that is supplied with the software <STANDARD.TPL>, the model contains nothing, i. e. it contains no 2D or 3D data.
It does, however, contain the following structure components for generating your 2D or 3D objects:



The super-ordinate sections are **3D model** and a **2D drawing** with the name *Drawing*.

The 3D space contains the necessary **workplanes** (WP) for the generation of 2D objects and solids. After starting the WPs *Standard_XY, Standard_YZ* and *Standard_XZ* are available.
Whilst the 2D objects in 3D space are linked to the WP on which they have been generated, the solids and the 3D polygons are managed completely independently in the **3DGroupStructure** section that can also contain groups for these 3D objects.

The **2D drawing** contains two **partial drawings** with the names *Partial Drawing1* and *Standard sheet area*. These two are separated work sections inside a 2D drawing that can each be assigned a separate scale and separate access authorisation with reference to the 2D objects.
The partial drawings and the *Standard sheet area* is reserved for the use of standard sheets (text fields). The scale applicable to them is always 1:1.
Partial Drawing1 contains a **group** with the name *Group*. The advantage of groups is that you can use them to organise the 2D objects in a partial drawing in a logical manner. You can also use them when completing component lists of materials. In addition, you can select/deselect all the objects in a group in a single step.

In addition to the structures already mentioned all the 2D and 3D objects as well as all the model-related settings (e.g. layers, fonts and font sizes, point filters) are managed in the model itself and moved to the HD when you save your data.

When you execute a **Save** procedure, all the data relating to a model will be saved in a file using **MOD** as its extension.

<u>Change model structure:</u>

<u>3D model</u>

When you are designing solids in **3D model** the position of each is defined by means of **WPs**. For this reason the normal practice is to use more than just the standard WPs.
You can, for example, use the ***Define WP*** command all the time or design them temporarily. These will be inserted beneath the 3D space in the model structure.

If the 3D objects (solids, 3D polygons) are to be managed in groups, you can use the ***Model Explorer*** to set up a 3D group structure underneath the **3D workplane**.

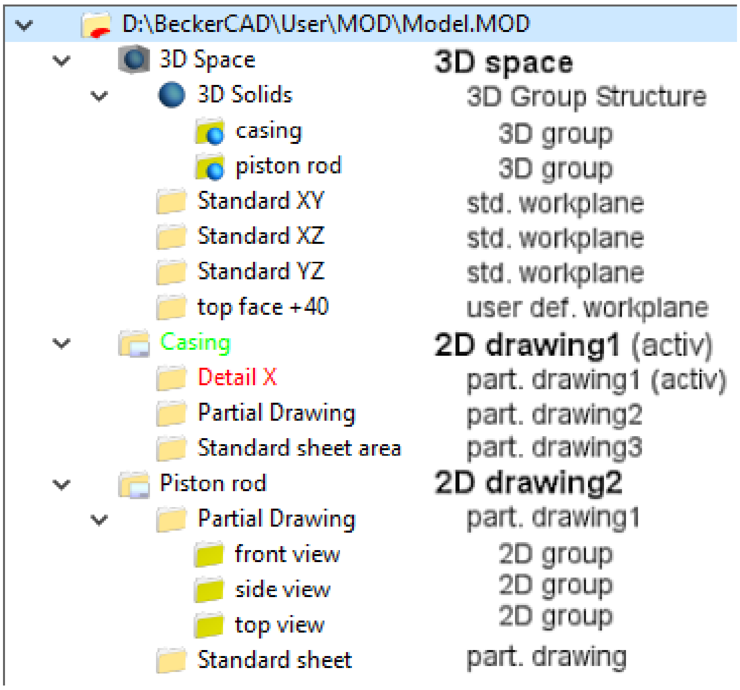
2D drawings

If you want to work with a number of **2D drawings** within a model, you can set this up using the ***Model Explorer*** and allocate another description.

If you wish to set up further partial drawings within the 2D drawing in addition to those already set, select the desired 2D drawing and load the menu command ***Settings, 2D- Drawing***.

Below the current partial drawing or the current workplane in 3D space, you can set up a **group structure** for 2D objects using the ***Model Explorer***.

After carrying out the modifications, the following structure e.g. could result:



Insert 2D and 3D data from another model

At present you can only work on one model during a work session. But you can integrate 2D or 3D data from another model into the current one by using the following options:

2D objects

2D objects that are required very frequently in several models can be saved first as **symbols** in a symbol library. After you have loaded the library, you can insert all the symbols it contains in the new model. (For further information please read chapter ***2D commands, Use symbols***).

You can also insert entire drawings in a new model. This is done using the method for loading an additional drawing. (For further information please read the chapter ***2D commands, Drawings and partial drawings***).

3D objects

3D objects can be inserted in another model. They must first be saved using the ***File, Export, SAT file*** menu commands. Then use ***Insert, SAT file*** to insert whatever is required in the new model. (For further information please read the chapter ***3D commands, Insert and export fixed solids***).

Template

A template (extension TPL) is a file, in which defaults can be saved as presettings for the generation and processing of models (see chapter ***Determine program settings, Determine model specific settings***).

I.e. templates take work away from you - after starting the program or after setting up a new model, all settings are determine in the way required for generating and processing objects.

When you start *BeckerCAD* a template is loaded with the file name STANDARD.TPL, after you have quit the start-up assistant. The settings within this file are then available.

If a model is to be generated and processed using settings from a different specific template, do not select the empty model which is offered after starting the program on the basis of one of these templates .
Instead, generate a new model using the menu command ***File, New Model***. A template must be entered which "informs" your model of the settings required.

When setting up and then saving a model, it takes over the information from the two first areas which has been "passed" to it from the template.
These remain in the model and are not influenced by later modifications to the template.

Further information concerning the use of templates - particularly in connection with presettings for the drawing and printing the drawing - can be found in the chapter ***2D Commands, Drawings and Partial Drawings***.

Please note:

> Templates can be used to determine your individual work environment.
> However, in order to ensure continuity of work results when a

number of workstations are working with *BeckerCAD* all workstations should use identical templates.

The template allocated to a model cannot be replaced by another.

Set up new template

If a new template is to be set up, use the menu command ***File, New Template***.

1. After you have made any alterations since last saving the current file, a message will appear asking whether or not the current version of the file is to be saved under the name stated at the very top of the screen.
If the *Yes* command button is pressed, the data is saved, if ***No*** is pressed, the data is lost.
Cancel is for exiting the procedure for designing a model.

2. The new template is automatically given the name TEMPLATE (system internal).
This name is displayed in the top of the main screen.

This system internally determined name is only used until another name is defined for the template, by using the menu command ***File, Save*** or ***Save As***.

Open template

If an existing template is to be processed, this can be opened using the menu command ***File, Open Template***.

1. In the ***Drive*** and ***Folder*** list boxes, the backup location for templates is displayed which was determined using the menu command ***Settings, Folder***. The predetermined ***file type*** *.TPL cannot be modified.
If necessary, determine a different drive and folder.

2. In the ***file name*** text box, enter the template name which is to be opened.
If the templates are displayed in the ***Found*** dialog box, highlight the one you wish to open.

3. The template is then opened by selecting *OK*.

Set up new model

 This command can also be activated using this icon.

Use the method described here to set up a new model:

1. Select the ***File, New model*** menu command.

2. If you have made any alterations since last saving the current file, a message will appear asking whether or not the current version of the file is to be saved under the name stated at the very top of the screen.
If the ***Yes*** command button is pressed, the data is saved, if ***No*** is pressed, the data is lost.
Cancel is for exiting the procedure for designing a model.

3. The dialog box for loading a template is then displayed.

Using the menu command ***File, Open Template****,* determine the ***drive***, ***folder*** and ***file*** name of the template and confirm with *Open*.

4. The new model is automatically given the name MODEL.MOD (system internal).
This name is displayed in the top of the main screen.

This system internally determined name is only used until another name is defined for the template, by using the menu command ***File, Save*** or ***Save As***.

Please note:

The data forwarded from the template to the model is taken over when saving the model.
If settings are modified whilst processing the model, their current values are taken over in the model and saved with them.

Open model

 This command can also be activated using this icon.

When you need information on how to open a model that has been saved, read the contents of this chapter.

When you want to insert 2D drawings already in another model without erasing any of the data relating to the current model, read the chapter ***2D commands, Drawings and partial drawings, Add drawing.***

1. Select the ***File, Open model*** menu command.

2. You will see the name of the folder containing models in the ***Find in*** text box of the dialog that has just opened. The name is that specified using the ***Settings, Folder*** menu.
You may perhaps have to select a different folder.

3. Tag the file you want in the list of MOD files, or type the name of the file in the ***file name*** text box.
4. Click the ***Open*** command button.
5. If you have made any alterations since last saving the current file, a message will appear asking whether or not the current version of the file is to be saved under the name stated at the very top of the screen.
   If the ***Yes*** command button is pressed, the data is saved, if ***No*** is pressed, the data is lost.
   Cancel is for exiting the procedure for opening a new model.

Please note:

You can also open a model using drag & drop.

Save model or template (F2)

When saving models and templates, one of two menu commands can be used. You can also automatically set up a backup file of the processing status of a model or template.

Save

The menu command ***File, Save*** can be used to save the current model or template under the name which is displayed in the header at the top of the screen.

This does not apply, when the temporary model name MODELL.MOD or the temporary WP name VORLAGE.TPL is displayed. In this case the dialog box for the ***File, Save As*** command will appear.

 The command ***File, Save*** can also be loaded using the displayed icon the main toolbar.

Save As

The menu command ***File, Save As*** can be used to save the current model or template under a name of your own choice.

1. Depending on the file type currently being processed, *.MOD for a model or *.TPL for a template will be displayed in the ***file type*** text box in the dialog box.
   If necessary, another file type can be selected, i.e. a model, for example, is to be saved as a template.
2. In the ***drive*** and ***folder*** list boxes, the presettings for the backup location of the file is displayed which was determined

using the menu command ***Settings, Folder***.
If necessary, determine another drive and folder.

3. In the ***file name*** text box, enter the name under which the model or template is to be saved (special characters, spaces and umlauts are not permitted).
Using the cursor, a name of an existing file can be selected from the file list and then written over.

4. Saving is started with ***Save***.
If a file name has been entered which already exists, the ***Yes*** and ***No*** command buttons must then be used to specify whether the old file should be replaced or not.

Automatic save

If you wish to automatically set up a backup file of your results of the processing status of a model or template, proceed as follows:

1. Select the menu command ***Settings, Options***.
2. Define on the option card ***Save and Recover***, after which number of *actions* the automatic save is to be carried out.
3. Activate the setting ***backup automatically***.
After the entered interval, the model or the template is saved.
The backup file has the name of the currently processed file which has the character # preceding it. With the setting number of auto save files it can be defined, how many auto save copies (#*.MOD) should be stored to a model in the current folder for the *auto save files*. If the number is >1 the save files will be named according the key: **#<model name>-#nn#.mod**.
The file with the lowest number is the oldest one, the file with the highest number the newest one. In the range of the numbers for auto save files, the files will be updated according to the settings „backup after ?? actions“ or „backup after ?? minutes“.This file will be saved in the folder that you have specified for backup files (#*.MOD) using ***Settings, Folder***.

3.13 The Model Explorer

Open the Model Explorer using the menu ***View***, ***Model Explorer***.

The Model Explorer offers three option cards:

Model Explorer – Shows all objects in the 2D and 3D area in a structured tree and the content of selected areas in a detailed view.

It allows the management and structuring of drawings and the 3D model.

Layer Explorer – Manages the display properties of 2D objects. Further informations can be found in the chapter ***2D commands***, ***Control display of 2D objects***.

Change to the option card ***Model***:

The Model Explorer displays the hierarchy in your model as well as all structure elements within the 2D/3D area. In addition, you can manage and structure your drawings and 3D designs extremely efficiently using the Model Explorer.

Open the Model Explorer using the ***View, Model Explorer*** menu.

The entire structure of the opened model is displayed in the ***Tree structure*** on the left side of the Model Explorer. In the ***List view*** on the right side, the objects and/or groups of the selected folder are displayed in the tree structure. In the bottom half of the Model Explorer, you will find the ***News area*** in which error messages or notes concerning design problems can be displayed (example: after activating a snap point using the keyboard without previously having activated a command). You can activate or deactivate the display of news using the ***Settings, Options*** menu on the ***User Interface*** option card under ***use model explorer to display message***.

The individual areas of the Model Explorer can be adapted individually to the required size by clicking on the grey strip that separates the areas and keep the mouse button pressed to drag the area to the required size.

Overview of the icons in the Model Explorer

The Model Explorer displays different icons related the the current option card. Please find summary of the icons with a short explanation of the commands:

Option card: Model Explorer

,	Scan groups on / off
,	New group will (not) be current group
,	(Do not) activate selected component
	Localise object in model tree
,	Highlight active groups

System internal tipps and warnings are deleted

All texts are deleted

Option card: Layer Explorer

, Scan layer on / off

, New layer (un)marked

Localise object

Current layer for Geometry

Current layer for Text

Current layer for Construction_Aids

Current layer for Hatches

Current layer for Dimensions

Current layer for Centre_Lines

Current layer for Hidden_Lines

Current layer for Hidden_Edges

Current layer for Winding_Lines

Current layer for tangent edges

, List view / Tree view active

System internal tipps and warnings are deleted

All texts are deleted

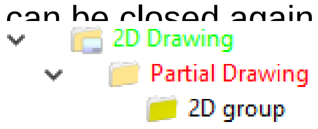
Working with the Model Explorer

The ***Model Explorer*** can be *docked* onto the program window. Once you have opened the dialog box, move the ***Model Explorer*** to the border of the program window until it visibly modifies its shape and then let it fall. The docked on ***Model Explorer*** can be released again by clicking on the strip and keeping the mouse button pressed, drag the dialog box into the centre of the program window.
In docked on status, minimise the ***Model Explorer*** using the icon and close it using the icon. Once, in the docked on state, you have minimised the ***Model Explorer***, you can open it again by double-clicking next to the strip in the program window frame or by dragging it to the strip by keeping the mosue button pressed.
If you wish to move the ***Model Explorer*** to the border of the program window without it being docked on, keep the **Ctrl key** pressed until you have positioned it on the required location.

Scan groups on / Scan groups off

If scanning is switched on, you can scan through the entire model structure with a mouse click or by using the arrow keys on the keyboard.
Click on a structure branch in the left half of the Model Explorer and then confirm the arrow key on the keyboard.
Closed node points 

can be opened whilst scanning by pressing the right arrow key on the keyboard, in order to allow you to scan in the subordinate structures. Opened nodes can be closed again by pressing the left arrow key.



Determine new group as current group / Do not determine new group as current group

If this command is active, a new group is always generated one level under the active group or partial drawing and can be directly renamed.

If the command is not active the previously active group or partial drawing remains active. Groups that are later generated in the same procedure are generated on the same hierarchical level as the first new group.

Localise object in model tree

Once you have started the command using the displayed icon, identify an object in the active graphic window. The structure tree (left) will be opened up to the folder in which this object is found. The selected object will be suppressed in grey in the list view (right) of the Model Explorer.

If the identified object is a model view, you can choose whether you want to locate the model view or the corresponding solid. To switch to the 3D window at the same time when locating the body, activate the option *Activate tagged components* described below if necessary.

Non active (Part) Group lowlighten on / Non active (Part) Group lowlighten off

If the command is ***on,*** in this case only the objects in the active group are displayed according to the standard settings. All other objects are lowlighted. When changing the active group, e.g. in the Model Explorer, the display of the objects in active and non active groups will change accordingly.

If the command is ***off,*** all objects in active and non active groups are displayed according to the standard settings.

Activate tagged components / Do not activate tagged components

If this command is active, each folder you have selected by mouse click in the tree structure of the Model Explorer, is defined as the active working area.

If you identify the *3D Model* folder or the folder of a *2D Drawing,* the previously active workplane or the previously active partial drawing becomes the active working area.

If, however you identify a workplane or a partial drawing/group directly, this folder becomes the active working area.

Display

The colour display of the respectively active areas can be determined using the ***Settings, Options*** menu on the ***Model Explorer*** option card.

In the view of the Model Explorer the folders can be differentiated between via colours:

- (yellow) = 2D group
- (yellow) = 3D group

Notes are stored in the Model Explorer

The message box in the ***Model Explorer*** is not only used to output error messages, it can be used to store own notes with the model.

Using the command ***Clear message box*** system internal tipps and warnings, are deleted. User notes will no be deleted.

To delete all the texts in the message box, he new command ***Erase All*** is used. Single texts can also be handled with standard text procedures, e.g. Mark text and delete it using the delete key.

If texts where deleted by accident, an undo is available with the key combinations ***Ctrl+z*** or ***Ctrl+Shift+z*** , while the cursor is in the message box.

Overview of Model Explorer commands

Independent of the structure levels on which you find yourself, the following actions can be carried out using the Model Explorer. In the tree structure (left in the Model Explorer) or the list view (right in the

Model Explorer) in the required folder or object, open the context menu using the right mouse button and carry out one of the commands described under the heading ***Context menu in Model Explorer***.
The following gives you an overview of a number of functions that you can carry out using the Model Explorer.

- **Display**
  Model/model description including path
  3D space
  3D groups
  3D objects (solid, axes, etc.)
  3D workplanes
  2D drawings
  Partial drawings
  Groups
  Geometry objects
- **Set up**
  2D drawings
  Partial drawings
  2D groups in partial drawings
  2D groups in workplanes
  3D groups in 3D work area
- **Erase**
  2D drawings
  Partial drawings
  Workplanes
  Groups in partial drawings
  Groups on workplanes
  3D groups in 3D work area
  2D objects
  3D objects
- **Disassemble 2D and 3D groups**
- **Structure**
  Groups in partial drawings
  Groups in workplanes
  3D groups
  2D and 3D objects

  Available methods:
  Single and multiple selection
  Drag&Drop
  Cut&Copy&Paste

- **Rename**
  3D space
  3D workplanes
  3D groups
  2D drawings
  Partial drawings
  2D-/3D groups
  Find and replace any names through the entire tree or partial trees

- **Selection...**
  ... of components including all subordinate content (groups and elements)
  ... of geometry objects which lie directly within a component
  ... of all geometry objects within a component including substructures
  ... of the tagged components (geometry object, group, partial drawing, etc.)

- **Direct access to**
  Workplane dialog box
  Partial drawing dialog box
  Drawing manager
  Object editor (by double click on geometry object)

- **Activate...**
  ... *the current group*
  ... *the current work area* (partial drawing/WP) whereby the active group of this area is also set as this. This means that the active group is recognised per work area and set as this again once activated.
  ... *the target context* (2D drawing or 3D space)

- **Process groups**
  Status (activated/visible) including optical feedback through ICON and multiple selection (process a number of groups in one go)

- **Process attributes on**
  Groups
  Symbols, Standard Sheets
  Individual objects

- **Scan**
  not only through individual groups or partial drawings but also through the entire model structure

- **Work Planes**
  hide and unhide
  rename
  delete
  change to clip planes
  create standard clip plane

Context menu in Model Explorer

Open the ***Model Explorer*** using the menu ***View*** and switch to the option card ***Model Explorer***. The following describes the individual options of the context menu. The context menu can be opened using the right mouse button in the Model Explorer. Depending on whether you are currently in the tree structure or list view, different options are offered to you in the context menu.

Current group (F12)

The area, in which solids are generated (3D solids), as well as workplanes and groups in the 3D area or partial drawings and groups in 2D drawings, can be activated as the current target context for new objects, at the same time, as the current work area. The following objects are actually created in this active area:

3D Solids
Workplanes
2D / 3D Groups
2D Partial drawings

In the tree structure in the left of the Model Explorer or in the list view in the right of the model Explorer, identify the *3D Solid* node point, a workplane, a partial drawing or a group and select the ***Current Group*** option from the context menu. Then press the ***F12*** key to define the respective area as the active context.

Current 2D area (Shift+F12)

Workplanes (WP) or partial drawings (PD) can be activated as the current work area using this option. Opposite of the ***Current Group (F12)*** option, the group previously activated as the target context within this workplane/partial drawing, remains active. Only the work area is changed.

Current 2D/3D context (Ctrl+F12)

Alternatively to the key combination ***Ctrl+TAB*** or the ***Window*** menu, you can use this command to swap between opened 2D drawings and the 3D window using the Model Explorer. Within the structure of the selected drawing or the 3D window, the previous work area (3D, workplane, partial drawing or group) remains active.

Please note:

A new view of closed windows can be gained by opening the context menu on the closed drawing / the closed 3D window and selecting the ***New, New View*** option.

Selection

Using the ***Selection*** option, another menu is opened with which you can select numerous selection possibilities. You will need to differentiate between the objects (lines, circles, text, etc.) in a group and the groups (including the content of the group).

Select (Ctrl+S) /Deselect (Ctrl+D) geometry objects

This command can be used to select or deselect the objects in the selected structure folder (3D solid, workplane, partial drawing, group) in the *tree structure* or the objects in Groups selected in the *list view*.
Subgroups or objects contained within them, are not taken into consideration with this command.
If you keep the Ctrl key pressed when identifying folders in the tree structure by mouse click, the *Select/Deselect geometry object* is also carried out.

Select (Shift+S)/Deselect (Shift+D) all geometry objects

This command selects/deselects all objects within the selected folder (3D solid, workplane, partial drawing, Groups) including all objects in the subgroups.
If you keep the Shift key pressed when identifying folders in the tree structure by mouse click, the ***Select/Deselect geometry object*** is also carried out.

Select object (Alt+S) / Deselect object (Alt+D)

In the tree structure, select or deselect a group and its subgroups in the objects selected in the list view and /or groups and their subgroups.
Proceed as follows:
In the structure tree:
...the selection is equivalent to the group selection.

- Keep the ***Alt key*** pressed and click on the groups(s) that you wish to select.

or

- Tag the group you wish to select/deselect. Then use the key combination ***Alt+S*** to select or ***Alt+D*** to deselect.

or

- Tag the group you wish to select/deselect. Then select the ***Selection / Select Object*** or ***Deselect Object*** command from the context menu.

Repeat the procedure for other groups you wish to select.

<u>In the list view:</u>
In the list view the selection contains groups if groups are selected and objects if objects are selected. The difference is displayed using different colours.

- Tag the objects and groups you wish to select/deselect. Then use the key combination ***Alt+S*** to select or ***Alt+D*** to deselect.

or

- Tag the required objects and groups you wish to select/deselect. Then select the ***Selection / Select Object*** or ***Deselect Object*** command from the context menu.

Select all contents (Ctrl+Alt+S) / Deselect all contents (Ctrl+Alt+D)
This option can be used to select the objects and/or groups contained in the list view of the *tree structure*. Objects tagged in the *list view* are selected as objects, in tagged groups their objects and subgroups are selected.
<u>In the structure tree:</u>

- Keep the ***Ctrl*** and ***Alt*** keys pressed and click on the required structure folder whose content is to be selected. All folders below the 3D space and the drawings in the structure can be selected.

or

- Tag the structure folder *3D Solid*, workplane(s), partial drawing(s) or group(s) whose content you wish to select/deselect. Then choose the key combination ***Ctrl+Alt+S*** to select or the key combination ***Ctrl+Alt+D*** to deselect.

or

- Tag the structure folder *3D Solid*, workplane(s), partial drawing(s) or group(s) whose content you wish to select/deselect. Then choose the ***Selection / Select All Content*** command or ***Deselect All Content*** command from the context menu.

<u>**In the list view:**</u>

- Tag the groups whose content you wish to select/deselect. Then choose the key combination ***Ctrl+Alt+S*** to select or the key combination ***Ctrl+Alt+D*** to deselect.

or

- Tag the groups whose content you wish to select/deselect. Then choose the ***Selection / Select All Content*** command or ***Deselect All Content*** command from the context menu.

Paste selected objects (Shift+Alt+S)

This command can be used to move the objects previously selected from the structure folder in the tree structure or the groups selected in the list view.
The command correspond to the command from the ***Process*** menu, ***Accept Selection In Group***

Please note:
Objects and groups can be moved between the structured using Drag&Drop:

- Firstly tag the objects or groups you wish to move in the structure.
- Click again on the objects and, whilst keeping the mouse button pressed, drag them to the required structure folder in which you wish to insert the objects/folder.

Insert copies of selected objects (Shift+Alt+C)

This command can be used to copy objects previously selected in the drawing / in the 3D window into the selected structure folder of the tree structure or the selected group in the list view. The absolute position of the copies is taken over from the originals.

Please note:

Objects and groups can also be copied between the structures using Drag&Drop:

- Firstly tag the objects or groups that are to be copied into the structure.

- Click on the object again und keep the mouse button and the Ctrl key pressed to drag it to the required structure folder into which the object/folder is to be inserted. The copy procedure is recognised by the plus sign displayed on the mouse pointer.

Visibility/Pickabilitiy

- **Visible / Invisible (O)**
  This command can be used to suppress a workplane or a partial drawing with all the elements contained and the objects of a group and their subgroups. WP, Partial Drawings and Groups that cannot be seen are automatically made inactive and are identified by the icon.
  Please note:
  If you wish to make a previously suppressed Component visible and active again, the easiest way is to use the option ***Active / Inactive*** so that the Component is active and visible at the same time.
- **Active / Inactive (P)**
  Objects on workplanes, partial drawings or in groups that you have deactivated with this command, remain visible but cannot be selected or processed. Inactive workplanes, partial drawings and groups are tagged with the icon.
- **Toggle visability incl. child groups (Ctrl+Alt+O)**
  This command can be used to suppress or hide the objects of workplanes and partial drawings, as well as groups and their subgroups. In this case, in opposition to the ***visible / hidden*** option, all groups and subgroups also receive the status visible / hidden.
- **Toggle pickability incl. child groups (Ctrl+Alt+P)**
  Objects on workplanes, partial drawings and in groups which you deactivate using this option, remain visible but cannot be worked with or picked. In this case, in opposition to the ***active / inactive*** option, all groups and subgroups also receive the status active / inactive.
- **Branch visible (Ctrl+O)**
  All workplanes, partial drawings and groups and subgroups below the node get the status visible.
- **Branch invisible (Ctrl+Shift+O)**
  All workplanes, partial drawings and groups and subgroups below the node get the status invisible.

- **Branch active (Ctrl+P)**
  All workplanes, partial drawings and groups and subgroups below the node get the status active.
- **Branch inactive (Ctrl+Shift+P)**
  All workplanes, partial drawings and groups and subgroups below the node get the status inactive.

Use Workplanes as Cutting Planes

Existing workplanes can be switched to cutting planes. If the cutting plane interferes a solid this is cut represented.

In addition three standard cutting planes can be called each of them in the centre of all solids with different directions. A maximum of 6 cutting planes can be active at the same time, additional cutting planes will be displayed as non active cutting planes.
The display of the sections can be set with the option ***clipped colour properties*** in the dialogue window ***3D Display*** (menu ***Settings***, ***3D Display*** in the tab ***Solid Display***).

Cutting planes can be modified and transformed with the same commands as the workplanes, e.g. move along Z direction to “walk” through the model with the command:  ***Move WP along Z*** or switch the cutting direction with the command :  ***Invert WP***.

A standard cutting plane can be activated as follows:

1. Open the ***Model Explorer*** using the menu command ***View*** and select the tab ***Model Explorer***.
2. Start the context menu and select from the option ***Cutting Plane*** the standard cutting plane.

To switch an existing workplane to a cutting plane do the following:

1. Open the ***Model Explorer*** using the menu command ***View*** and select the tab ***Model Explorer***.
2. Place the cursor on the workplane and open the context menu. Select the option ***Cutting Plane***, ***Add and Activate***.***(Shift+Q)***.

To deactivate a cutting plane, so that the solid is not displayed in a sectioned manner any more, open the context menu on the cutting plane and select the option ***Cutting Plane***, ***Activate/Deactivate***.

To switch from a cutting plane to a workplane the command ***Cutting Plane***, ***Remove*** is used.

Icons in the Model Explorer:

 Active Cutting Plane

 Inactive Cutting Plane

Properties (Shift+F2)

Only in the context menu in the list view (right)

Double click on the required object or tag it in the list view and then from the context menu, select the ***Properties*** option. Depending on the selected object, the ***2D Object editor*** dialog box is opened or a respective other dialog box (text editor, 2D standard part, etc.).
This command corresponds to the ***Information / Edit Object*** command from the toolbar.

Focus objects (Shift+F3)

Only in the context menu in the list view (right)

By selecting this command in the context menu, objects tagged in the list view or the content of tagged folders will be highlighted in colour and focused in the respective drawing or the 3D model according to the menu ***Settings, Options*** on the ***User Interface, Model Explorer*** option card.

The command can also be started using the following key combinations:

Ctrl key

If the ***Ctrl key*** is pressed while starting the command in the context menu, the current setting in the dialogue window ***Options*** is inverted.

Shift key – only for 3D objects

If the ***Ctrl key*** is pressed while starting the command in the context menu, in addition all other solids in the current display are faded out.

 or  To fade in all solids, select the icon ***Hide Solids*** or ***Hide all Others*** on the top of the 3D window.

The two keys can also be used in combination.

Please note:

The additional keys are only valid when starting the command using the context menu and not when using the key combination ***Shift+F3***.

New

Dependent on the selected folder or object, use this command to define a new ***group*** (2D/3D area), a new ***partial drawing*** (only 2D area) or a new ***2D drawing*** (only 2D area).

Set up new group (Ctrl+N)
If in the tree structure in the 3D area you tag a workplane, or in a 2D drawing a partial drawing or a group, a new group will be set up under this folder once the key combination ***Ctrl+N*** has been selected or ***New / Group*** is selected from the context menu.

Depending on the setting you have made in the Model Explorer using the icon  **Determine new group as current group /**  **Do not determine new group as current group**, further new groups are stuctured or set up on the same structure level.

New partial drawing (Ctrl+N)
In the structure tree, tag a 2D drawing and select the key combination ***Ctrl+N*** or select ***New / Partial Drawing*** from the context menu.
In order to be able to edit the settings of a partial drawing, tag a partial drawing in the tree structure and select the ***New / Drawing Settings*** option from the context menu.

2D drawing (Ctrl+N)
In the tree structure, tag the topmost node point (model path) and select the key combination ***Ctrl+N*** or the option ***New / 2D Drawing*** from the context menu.
The Drawing Manager can be opened using the Model Explorer by tagging a 2D drawing in the tree structure and selecting the ***New / Drawing Manager*** option from the context menu.

New view (Shift+Ctrl+N)

If you wish to open a new 2D or 3D view window, for a new 2D view window, tag a drawing in the tree structure and then select the ***New, New View*** option in the context menu.
The same command can be used to set up a new 3D view window if the context menu is opened on the *3D space* structure folder.

Edit attributes (Shift+A)

If a group has been allocated attributes, this command can be used to display or edit the attributes. Open the context menu by pressing the right mouse button whilst on the respective group in the tree structure or list view and select the command ***Edit Attributes (Shift+A)***.
In order to be able to display or edit the attributes of an object (such as

the a standard sheet or another symbol), firstly tag the object in the list view and then select the required command from the context menu or using the keyboard commands

Rename (F2)

Tag the folder you wish to rename in the tree structure or the list view. Press the ***F2*** key or select the ***Rename*** command from the context menu. Enter the new name and confirm the entry with the Enter key. Objects such as lines, circles and text cannot be renamed.

Find (F3)

Enter the word to be found entirely or part of it in the dialog box that appears by selecting the ***F3*** key or using the ***Find*** command in the context menu. Start the search using the ***Find next*** command button. After every further click on this command button, the next search results are displayed.

Match whole text only
There must be no further letters in front of and behind the search text.

Match case
The search is case-sensitive.

Match regular expression
With regular expressions, any complex search queries are possible. With special characters, search terms can be put together so that not only a single term is found, but also precisely defined combinations of terms.
Examples for the search with regular expressions can be found at the end of this chapter.

Wrap around
The search is continued at the beginning of the list up to the selected element.

Siblings only
The search is restricted to the current structure level within the selected structure tree.

<u>**Examples for regular expressions**</u>

For regular expression, the search text is expanded using simple controls or control elements. Here are some examples, which can only give a small overview of the entire spectrum of the search with regular expressions.

Character	Meanings	Example
.	Represents exactly one character	.ower = tower, lower
\w	An alphanumeric character	aAbBcC …
\d	A digit	0123456789
\d+	A digit sequence	
*	No or several repititions	The precending search expression can not occur once, or occur once or several times
+	At least one repitition	The precending search expression can occur once or several times
[ ]	Character class Matches all characters within the brackets. Areas can be specified with the hyphen	[abc] finds "a", "b" and "c". [a-z] finds a lower character. [0-9A-Z] finds a digit and an upper character
( )	Describes a sequence of characters	Grouping of characters and search operators

Replace (F4)

In order to replace descriptions of folders with new descriptions, select the ***F4*** key or select the ***Replace*** command from the context menu. In the ***Search for****:* text box, enter the text to be found. In the ***Replace by*** text box, enter the new text. Click on ***Search***, ***Replace*** or ***Replace All***.
As opposed to folders, the object names of lines, circles, text, etc, cannot be replaced by other descriptions.
The search is limited to the word start. Wildcards are not permitted.

Refresh (F5)

To update the view in the Model Explorer, press the ***F5*** key or select the ***Refresh*** option in the context menu.

Tag all (Ctrl+a) (Ctrl+A)

In order to tag all geometry objects and folders in the list view of the Model Explorer, press the key combination ***Ctrl+a*** or ***Ctrl+A***.
This command is active in the tree representation (left) as well as in the list view (right).

Select geometry objects (Shift+Ctrl+a) (Shift+Ctrl+A)

In order to only tag the geometry objects in the list view of the Model Explorer, press the key combination ***Shift+Ctrl+a*** or ***Shift+Ctrl+A***.
This command is active in the tree representation (left) as well as in the list view (right).

Cut (Ctrl+X) / Paste (Ctrl+V)

This command can be used to move groups or objects in the structure and between partial drawings and workplanes.

1. In the Model Explorer, tag the group(s) or object(s) you wish to move.
2. Select the key combination ***Ctrl+X*** or select the ***Cut*** option in the context menu.
3. In the tree structure, tag the node point under the group(s) or object(s) is to be positioned.
4. Press the key combination ***Ctrl+V*** or use the ***Paste*** option in the context menu.

Please note:

Groups and/or objects can also be moved in the structure and between different partial drawings and/or workplanes using ***Drag&Drop***.
Collect the required elements by keeping the Ctrl key pressed. Whilst keeping the mouse button pressed, drag the object and/or groups to the required position within the tree structure.

Copy (Ctrl+C)

2D and 3D objects, groups can be copied using this command.
Tag the desired element and choose the command ***Copy*** or press ***Ctrl+C*** keys. Select the destination folder and choose ***Paste*** in the context menu or press ***Ctrl+V*** keys. The copied 2D/3D objects will keep their absolute positon and will maybe on top of each other.

2D and 3D objects and groups can also be copied in the structure tree of the Model Explorer by Drag&Drop with the pressed ***Ctrl-key***.

Delete (Del.)

The following objects can be removed from the model using the Model Explorer:

- 2D- and 3D-Groups

- 3D workplanes

- 2D objects

- 3D objects

- Partial drawings

In the tree structure, tag the respective node point or, in the list view the respective objects and/or groups to be erased. Then press the ***Del.*** Key or select the ***Erase*** command from the context menu. The selected objects including the objects contained within them are removed from the model.

Please note:

The last partial drawing and the partial drawing for standard sheet area within a drawing, cannot be erased.

Disassemble (Ctrl+Del.)

The ***Disassemble*** command can be used to remove groups from the structure. The content of the group (objects and subgroups) remain and, within the structure, move up by one hierarchical level.

- Tag a group in the structure tree or collect them with the pressed ***Ctrl-key*** in the list view.
- Press the key combination ***Ctrl+Del.*** or select the ***Disassemble*** command from the context menu.

Edit display format (ShiftF6)

Instead of the internal object names for fixed solids as well as for the new 3D graphic objects free names can be assigned. For a text object instead of the object name the text itself can be displayed and edited in the model explorer. For symbols the symbol name will be displayed, optionally also with the folder structure from the symbol library. The names of 3D graphic objects, fixed solids as well as groups folders can also be assigned to attributes. The selected display format is always valid for the specific object type in the model.

Further information can be found here.

Format, Sort

Using the ***Format, Sort*** option opens another menu, which contains many different sort criteria. Different sort criterias can be defined for the structure tree and for the list view.

- **Sort on/off (F6)**
  This option can be used to define whether one of the following sorts are to be used or whether the objects and groups are to be listed in the order in which they were generated in the Model Explorer.

- **Toggle sort direction (F7)**
  The elements in the structure tree or the list view are sorted ascending or descending.
- **Sort case sensitive (F8)**
  The sorting is case sensitive.
  Condition: The sorting is active (F6)
- **Sort logical (Ctrl+F8)**
  The sorting follows logic criteria. An example for the logic sorting are numbers. Without a logic criteria when sorting 2 digit numbers a 0 msut be in front of all 1 digits. The logic criteria will sort first 1 digits and then 2 digits.
- **Sort groups separatly (F9)** (only in list view)
  The sorting differs between groups and objects. If this option is not active both arelisted on the same level.
  Condition: The sorting is active (F6)

Format, Open / close branch (F11)

This command can be used to open or close the selected branch in the tree structure including all the subordinate structures.

Format, Open all / close all (Shift+F11)

This command can be used to open or close all branches of the entire model area right up to the lowest structure.

Format, View side by side (Ctrl+L)

With this command the positon of the list view can be changed to be below or beside the structure tree.

Format, View among each other (Shift+Ctrl+L)

With this command the positon of the message window can be changed to be below or beside the structure tree.

Display Formats in the Model Explorer

Instead of the common object names, displayed in the Model Explorer for ***Fixed Solids*** as well as for the new ***3D Graphic Objects,*** a free name can be defined for those objects. For ***Texts*** it can be defined if the text should be displayed instead of the object name to edit it directly in the Model Explorer if needed. For **Symbols** the symbol name, even with the folder structure from the symbol library can be displayed.

The names of ***3D Graphic Objects***, ***Fixed Solids*** as well as for ***group*** folders can alternatively also be controlled by a number of attributes. The chosen display format is always valid for the object type in the model.

Rename Object Names

To change the name of specific ***3D Graphic Objects***, ***Fixed Solids***, mark the object name in the Model Explorer and click the ***F2*** key (context menu: ***Rename***) Enter the desired name and click the ***RETURN*** key.

Please note:

The internal object name is displayed again, when deleting the name.

Display and Edit Texts

If the text should be displayed and edited instead of the internal object name, click the right mouse button on the text and select ***Edit Display Format (Shift+F6)*** from the context menu. Activate the check box Show Text and click the ***Apply*** or ***OK*** button.
Then the text is displayed and can be edited in the Model Explorer. Mark the text and click the ***F2*** key (context menu: ***Rename***).

Please note:

When double clicking a text in the Model Explorer the dialogue window Process Text is offered, where besides the text further parameters for the text can be changed.

Display format by attributes

For **3D-Graphic Objects**, **Fixed Solids**, **Group folders** attributes can be displayed as names in the Model Explorer.

1. Click the right mouse button on the object and select ***Edit Display Format (Shift+F6)*** from the context menu.
2. To switch fast between different display formats define a specific ***display format*** by clicking the ***New*** button. Enter a name for the new display format and click the ***Apply*** button.
3. Activate the option ***Display Attribute***.
4. Define the display format by using the ***Append***, ***Insert*** and ***Remove*** buttons when selecting previously defined attributes from the list.
   The setting defined here, affects all objects of the same kind in the current model / template.
5. Confirm the settings by clicking the ***Apply*** or ***OK*** button.

Please note:

The display format by attributes must be defined in a template, if this display format should be active in all new models.

If the option ***Display Attribute*** is activated in the ***Display Format*** dialogue window, the objects are displayed with their internal names or their user defined names.
The ***Object name*** in the list of the current ***attributes*** represents the internal or user defined names for the display format by attributes.

When choosing a display format by attributes with only one attribute, this name and also the attribute value can be edited in the Model Explorer directly.
Please be aware: If the display format by attributes contains more than one attribute, the name cannot be edited in the Model Explorer.

4 2D Commands

4.1 Model units

The menu command ***Settings, Units*** can be used to determine how entered coordinates and length values are to be interpreted for 2D and 3D objects.
In the dialog box which opens, select one of the required units ***inch, mm, cm, m*** and confirm with ***OK***.

From now on, the required unit will be used in all dialog box fields followed by the description of the (length) unit, as well as in the Status dialog box in which coordinates and length values are displayed and entered for the 2D and 3D objects.

The selected unit is saved in the current model or in the current template file.

4.2 Coordinate systems

The position of each of the points in the 2D drawing window or in the current workplane (WP) of the 3D-view window can be determined by entering the coordinate values.

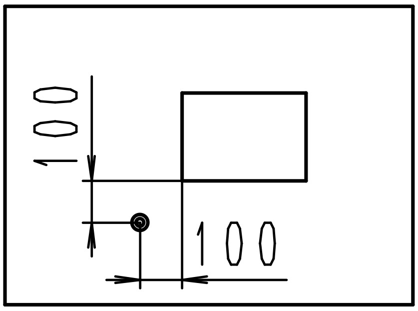
These can refer to the global or to a local coordinate system, they can be entered as Cartesian coordinates or as polar coordinates.

2D global coordinate system (GCS)

BeckerCAD uses a global coordinate system for the drawing area of each 2D drawing window or for the workplanes (WP) of the 3D window. The **origin** (zero point) of the GCS, to which the coordinate entries refer, always lies in the intersection of the coordinate aces.

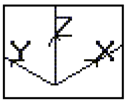
- In the 2D drawing window, the origin usually lies in the left bottom corner.
  From this point, the X-axis points to the right, the Y-axis upwards.

  If the origin of the GCS does not lie in the left bottom corner of the 2D drawing window, using the menu command ***Settings, 2D-Drawing*** in the ***origin***, values for the movement of the drawing can be entered, e.g.:



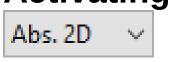
In this case, a drawing with the dimensions of an A4 sheet (297 x 210) is moved starting from the origin by 100 units in the direction of both the X and Y axes. This means that the left bottom corner of the drawing now has the coordinates (100, 100) in the GCS, the top right corner has the coordinates (397, 310).

- If working in the 3D-view window, in **2D mode**, the origin of the GCS lies in the origin of the current workplane, which is identified by a coordinate cross.
  The direction of the axes is more clearly shown in this illustration.

  Just as for the 2D drawing, a workplane can also be moved from the origin of the GCS.
  This is done with the help of the commands in the 3D icon bar: ***Edit Workplane Properties***.

Activating the GCS



In the list box in the main top toolbar, activate **Absolute 2D**.

The position of the origin is displayed by a marker when generating and transforming objects.

2D local coordinate system (LCS)

As well as the global coordinate system, a local coordinate system can also be defined in each 2D drawing window or in each workplane of a local coordinate system (LCS).

This icon in the main toolbar can be used to determine or modify the position of the LCS.
The axes of the LCS have the same directions as the GCS.

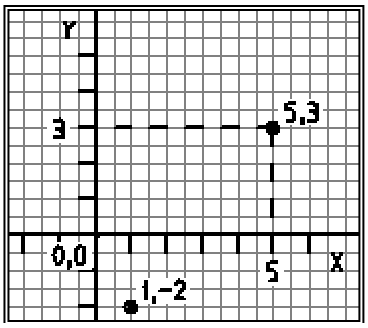
Activating the LCS

In the list box in the main top toolbar, activate **Relative 2D**.

If no origin has been defined for the LCS, it is identical to that of the GCS.

The position of the origin is displayed by a marker when generating and transforming objects.

Cartesian coordinates



The Cartesian coordinate system is formed from axes running at right angles to one another.

Each point in the plane is described by a coordinate pair, in which the distance of the point are given from the origin of the selected coordinate system in the direction of the X-axis and in the direction of the Y-axis.

Therefore, the point (5,3) is 5 units away from the origin in the direction of the X-axis and 3 units away from the origin in the direction of the Y-axis.

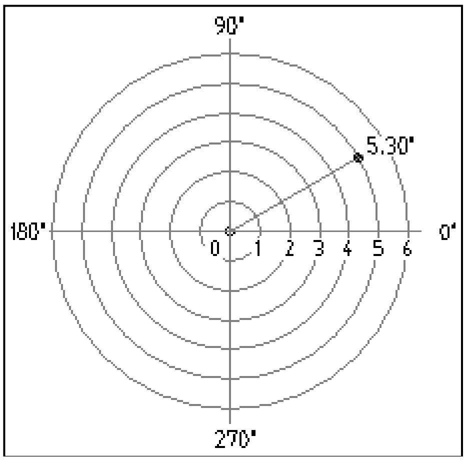
Activate Cartesian coordinates:
Load the menu command ***Settings, Point Definition*** and change to the ***Coordinates*** option card in order to activate the ***cartesian*** radio button.

If this is active, the first two text boxes in the Status dialog box are annotated with *X* and *Y* or *DX* and *DY*.

Please note:

When designing relative to a point (Relative To Point, Relative To Last Point) with the shortcut ***Strg+p (Strg+P)*** you can switch between cartesian and polar coordinates.

Polar coordinates



When using polar coordinates, each point in the plane is defined by a distance and an angle from the origin of the selected coordinate system. The angle is taken from an angle of 0° to the "right" or "east" of the origin in an anticlockwise direction.

Therefore, the point (5, 30°) lies 5 units distance away from the origin and is under an angle of 30°.

Activating polar coordinates:
Load the menu command ***Settings, Point Definition*** and change to the ***Coordinates*** option card in order to activate the ***polar*** radio button. This is not available in the 3D-view window.

If this is active, the first two text boxes in the Status dialog box are annotated with *Pol. dist.* and *Pol. angle*.

Please note:

> When designing relative to a point (Relative To Point, Relative To Last Point) with the shortcut ***Strg+p (Strg+P)*** you can switch between cartesian and polar coordinates.

Display coordinates

If points need to be defined when drawing or modifying the position of 2D objects, a cross-hair cursor is displayed.
Whilst moving the cross-hair cursor in the drawing area of the 2D window or in a workplane in the 3D window, the coordinate values of the current position (intersection of cross-hair) are displayed in the first two text boxes of the Status dialog box.
The displayed values refer to the position of the origin of the active coordinate system. This is identified by a marker.

The format of the Satus dialog box display for input and output can be determined using ***Settings***, ***Options***, ***User Interface*** in the ***Status dialog box***.

If Cartesian co-ordinates have been selected, the text boxes are annotated with ***X, Y, DX, DY***, for polar coordinates with ***Pol. dist., Pol. angle***.

4.3 Define points

The design and modification of objects as well as a number of further *BeckerCAD* commands require the determination of points and that additional values are entered using the keyboard.

For example, when drawing a line - the starting point, when drawing a circle - the centre, when moving objects - the starting point of the move vector.

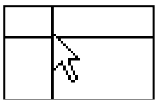
Further entries which are required can be entered by either defining a point or making entries in the Status dialog box, e.g. length and angle of a line, radius of a circle, angle and length of the move vector.

The following options are available for the definition of points:

- Define any point with the cursor
- Define points with the support of a drawing grid
- Snap points on objects with the Snap Menu commands
- Snap points on objects with activated point filter
- Define points by entering their coordinates in the Status dialog box

Define points using the cursor

If points are to be determined anywhere in the drawing area or in the selected workplane (WP), use the simple cross-hair cursor:



Move the cross-hair cursor to the required position whose coordinates are displayed in the first two text boxes of the Status dialog box and confirm by pressing the left mouse button.
If a cross-hair cursor other than that shown above is displayed, the cursor is in snap mode. If an object is within the cursor snapbox, the program tries to determine a specific point on this object, e.g. an endpoint.

If this is not required, the snap mode can be deactivated permanently. Select the menu command ***Settings, Point Definition***. Activate the ***none active*** check box.

If the snap mode is only to be deactivated whilst determining a point, load the ***Snap Menu*** and select ***Cursor***. This can also be activated after you have typed either ***c*** *or* ***C***.

Further information concerning the snap mode can be found under the settings for snapping object points.

Move cursor horizontally/vertically
After pressing the ***H, h*** key on the keyboard, the cursor can only be moved horizontally - based on the last point defined, i.e. parallel to the X-axis of the drawing area or to the first axis of the selected WP.
After pressing ***V, v*** the cursor can only be moved vertically, i.e. parallel to the Y-axis of the drawing area or the second axis of the WP.

Pressing the respective key again then deactivates the active mode.

Define points with active drawing grid

Using the menu command ***Settings, Drawing Grid*** one or more point grids can be defined.

This limits the movement options of the cross-hair cursor to just the grid points or it can simply be displayed as an orientation grid on the screen.

This is valid for the 2D window as well as for the current workplane of the 3D window.

Determine settings for drawing grid

Select the menu command ***Settings, Drawing Grid*** in order to define the settings for one or a number of drawing grids.

In the dialog box which then appears, determine the required options and values:

Grid name
If a grid is to be defined and saved in a grid list with the ***Generate Grid*** command button , enter the name of the grid, e.g. "5 x 5".

If a grid already exists with this name, this is generated with a new name after a message has been displayed. This corresponds to the old name plus a point and a number.

Current grid
A grid which has been saved in the grid list can be selected and is taken over as the current grid.

Active grid
Activates the selected grid, i.e. when defining points, the cursor can only move on the grid points.
However, if the cursor is to be freely movable again for the definition of the next point, the grid can be temporarily deactivated by keeping the Shift key pressed.

For the case that one of the following options is selected from the Snap Menu, the drawing grid deactivates itself automatically for the time that the selected option is active.

Autosnap
Endpoint
Midpoint
Centre
Quadrant Point
Intersection Point
Intersection 2 Lines
Relative to Foot Point
Point
Text Reference
Symbol Reference
Snap Point
Face/Centre Of Gravity
Tangential To
Perpendicular To

Draw grid
If this check box is active, the grid is displayed according to the defined settings within the boundaries of the 2D drawing area or the current workplane (WP) of the 3D-view window.
If the grid is not ***active***, it can only be used as an orientation grid.

A displayed grid is not printed.

Start point X/Y
Determines the start point of the grid, i.e. a point which corresponds with a point on the grid.
The coordinates of the starting point always refers to the origin of the global coordinate system of the 2D drawing area or the current WP.

 The start point can either be set via absolute coordinates or via the displayed icon using the point definition menu.

dX/dY
Defines the increment of the grid in the X and Y direction.

Display of the grid

Grid color
Defines the color of the grid points.

Point grid
The points of the grid are displayed in the size of one pixel.

Line grid
Continuous lines are drawn at grid spacing. The representation corresponds to that of a checkered writing pad.

Crossed grid
The grid is represented by crosses. You can define their line length in the ***grid cross width*** and ***height*** fields. The lengths of the lines are displayed depending on the scale of the active drawing.

Grid colour
Determines the colour of the grid points.

General settings

The following settings are global and not related to the individual grid. They are saved in the UI file when you exit the program.

Adaptive grid
When you zoom in or out of the view using the zoom functions, the grid spacing is adjusted according to the view. For example, if you zoom in extremely, the density of the displayed grid is automatically reduced.

Restrict display to drawing boundaries
The display of the grid in the 2D window is limited to the drawing borders.

 Generate grid

Saves all current settings and the respective name as a newly defined grid in the grid list.
A name must be entered for the grid in the ***grid name*** text box.

If this field contains the name of a grid which has already been saved in the grid list, a warning is given and the grid is saved with this name plus an additional continuous number.

 Remove Current grid

Erases the currently selected grid in the ***current grid*** text box from the grid list.

Accept
Saves all modifications to the settings of the currently selected grid and, if necessary, updates the display in the drawing area.

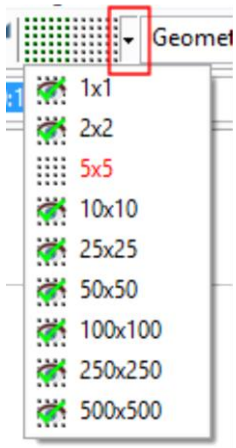
Please note:

> If drawing grids are generated in a template, in each model which has been generated based on this template, all grids and their settings which have been defined in the template can also be used in the respective models.

Select drawing grid

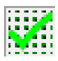
If a drawing grid has been saved in the grid list using the menu command ***Settings, Drawing Grid*** and the command button ***Generate Grid***, it can be selected in the ***current grid*** text box.

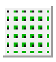
Alternatively you can select a grid using the ***Grid selection menu*** in the main toolbar.



Activate/deactivate drawing grid

A drawing grid can be activated using the menu command ***Settings, Drawing Grid***, by selecting the ***active grid*** check box and activating the ***Update grid*** command button.
To deactivate the grid, deactivate the ***active grid*** check box. Then press the ***Accept*** command button.

A drawing grid can be activated and deactivated using this icon in the toolbar. If the symbol is displayed , the grid is active:

 Grid **off**

 Grid **on**

Deactivate drawing grid temporarily
If a drawing grid is active, but is not to be used to position an individual

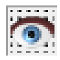
point, press the Shift key and keep it pressed until the position of the point has been confirmed.

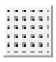
This happens automatically for the following Snap Menu options:

Autosnap
Endpoint
Midpoint
Centre
Quadrant Point
Intersection Point
Intersection 2 Lines
Relative to Foot Point
Point
Text Reference
Symbol Reference
Snap Point
Face/Centre Of Gravity
Tangential To
Perpendicular To

Display/do not display drawing grid

A drawing grid can be displayed by selecting the menu command ***Settings, Drawing Grid***, by activating the ***draw grid*** check box and pressing the ***Update grid*** command button.
If the grid is not to be displayed, deactivate the ***draw grid*** check box. Then press the ***Accept*** command button.

The display of a drawing grid can be activated or deactivated using this icon in the toolbar. If the symbol is displayed , the grid is active:

 Draw grid **No**

 Draw grid **Yes**

A displayed grid is not printed.

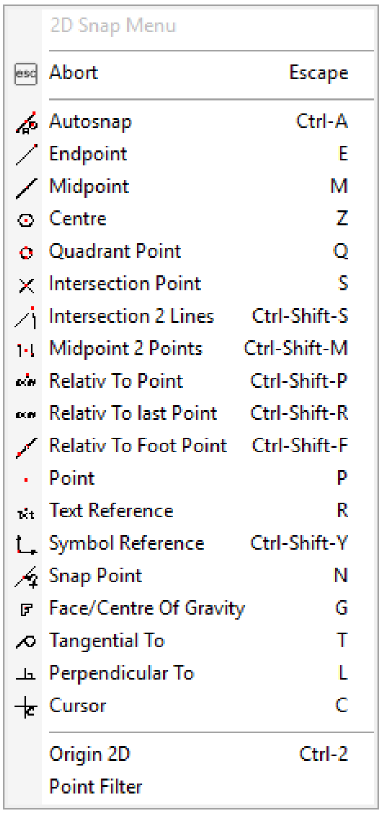
Define points with the Snap Menu

The 2D Snap Menu commands can be used to determine the next point to be defined in reference to existing 2D or 3D objects or to other points.

In this way, for example, the endpoint of an existing line or the centre of an existing circle or ellipse can be snapped. A line can also be drawn tangent to a circle or ellipse.

Loading the Snap Menu:

Three button mouse	middle mouse button
Two button mouse	Ctrl + right mouse button



Some of the commands in the Snap Menu can be activated directly without calling up the menu: this is done by typing a letter.

The letter to be used is included the snap menu and in the heading of the command name.

In order to snap a point on an object, the cursor must be positioned so that the object projects into the snapbox. Should several objects cut into the snapbox, the one found will be that closest to the cursor cross-hair.

Please note:

If none of the objects within the snap box fulfil the preselected snap point options or no object is active within the snapbox (also refer to ***Selection Filter***, ***Layers***, ***Partial Drawings***, ***Groups***), no point is set. An appropriate message is displayed in the prompt line of the main window and you are requested to re-identify.

A snap point activated using the Snap Menu or the keyboard remains active in the selected command until another snap point option is activated or the point filter is activated via the Snap Menu. This relation can be deactivated using ***Settings, Snap Menu***, ***“Offer last Snap Menu option again“***.

Exceptions with cursor display is the ***Cursor*** command where the cursor is displayed as a cross-hair cursor without the additional symbol and both the ***Tangent To*** and ***Perpendicular To*** conditions which are given another special symbol.

Closing the Snap Menu:

If the Snap Menu has been loaded whilst using a command, but it is now to be closed without defining a point, simply select ***Abort***, or press the ***Esc*** key on the keyboard.

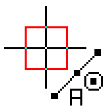
Please note:

- Points on objects can only be snapped if these objects are linked
  - to an active layer using the ***Layer Explorer*** in the ***Model Explorer***,
  - to one of the partial drawings that have been set to active or read only using the ***Settings***, **2D-*Drawing*** menu point,
  - to one of the groups that have been set to active in the ***Model Explorer***
  - to one of the selection options set to active using the ***Settings***, ***Object Filter*** menu point.
- 

 If working in the 3D-view window, by activating the 3D mode, points on solids can be snapped with some of the commands described in the following, e.g. the endpoint of a solid edge.
  The point defined in this way is not the determined spatial point but its projection on the current workplane (WP).

When using the 2D Snap Menu, the following snap commands and design conditions can be used:

Automatic snap of endpoints, midpoints, centres, quadrant points and intersection points (A, a)



 Snaps the nearest endpoint, centre, midpoint or quadrant point of lines, circles and arcs, ellipse and elliptical arcs and open splines or the nearest intersection point of such 2D line objects.

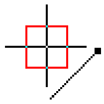
The Autosnap also snaps the following points on 2D line objects inside Symbols, without disassembling them before: End point, Mid Point, Centre Point, Quadrant Point.

1. In the Snap Menu, select the ***Autosnap*** command.
2. Move the cursor to an object. The nearest snap point (endpoint, midpoint, centre, quadrant point or intersection point) is identified by a marker.
   If you press the left mouse button, the currently marked point will be snapped.

Please note:

This mode can also be activated by entering the letter ***a, A*** or using the menu command ***Settings, Point Definition***. However, the latter does not only apply for the definition of a point but permanently.
By entering the letter ***f, F*** or using the mentioned menu command, the 'snap with absolute coordinates' can be activated again.

Snap endpoint of a line (E, e)

 Snaps the nearest endpoint of lines, arcs, elliptical arcs and open splines.

If the 3D mode is active, endpoints of solid edges or 3D axes can also be snapped in the 3D window.

1. In the Snap Menu, select the ***Endpoint*** command.

2. Identify one of the named 2D objects, the solid edge or a 3D axes near the required endpoint.

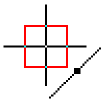
Please note:

- Full circles, full ellipses and closed splines do not have endpoints. This is also valid for solid edges with these geometric forms. Using the ***Process 2D, Divide At Point*** or ***Divide On Line*** such 2D objects can be given an "artificial" endpoint.
- Construction aids infinite lines and bisectors do not have an endpoint.
- For centre lines drawn as lines or generated from a "normal" line, for centre crosses generated to circles or 2 lines as well as solid axes - the visible endpoint is not snapped but the designed endpoint.

  Using ***Settings***, ***Snap Menu***, ***"Offer last Snap Menu option again"***, you can determine whether the snap point option ***Endpoint*** remains active after an endpoint is snapped.
  It is deactivated if

  - another snap point option is selected
  - the mode (snap, tangential, perpendicular mode) is changed
  - the current command is aborted

Snap midpoint of a line (M, m)

 Snaps a point halfway along the length of lines, arc, elliptical arcs.

If the 3D mode is active, the midpoint of solid edges and 3D axes can also be snapped in the 3D window.

1. In the Snap Menu, select the ***Midpoint*** command.

2. Identify one of the named 2D objects, the solid edge or a 3D axes near the required point.

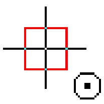
Please note:

- Full circles, full ellipses and closed splines do not have a midpoint. This is also valid for solid edges with geometric forms.
  With the commands ***Process 2D, Divide At Point*** or ***Divide On Line****,* the named 2D objects can later by given an "artificial" endpoint and then in this way also be given a midpoint.
- Construction aids infinite lines and bisectors do not have a midpoint.

  Using ***Settings***, ***Snap Menu***, ***"Offer last Snap Menu option again"***, you can determine whether the snap point option ***Midpoint*** remains active after an endpoint is snapped.
  It is deactivated if

  - another snap point option is selected
  - the mode (snap, tangential, perpendicular mode) is changed
  - the current command is aborted

Snap centre of a circle/ellipse (Z, z)

 Snaps the middle point (centre) of circles, ellipses and respective arcs.

If the 3D mode is active, in the 3D window, the centre of solid surfaces with the named geometric forms can be snapped. The requirement for this is that these faces are not curved in 3D space.

1. In the Snap Menu, select the ***Centre*** command.

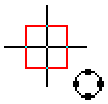
2. Identify the circumference of the circle or ellipse or the solid edge.

Please note:

Using ***Settings***, ***Snap Menu***, ***"Offer last Snap Menu option again"***, you can determine whether the snap point option ***Centre*** remains active after an endpoint is snapped.
It is deactivated if

- another snap point option is selected
- the mode (snap, tangential, perpendicular mode) is changed
- the current command is aborted

Snap quadrant point of a circle/ellipse (Q, q)

 Snaps the nearest of the points which lie on **0°, 90°, 180°, 270°** on a full circle or a full ellipse.

If 3D mode is active, in the 3D window, quadrant points of solid surfaces can also be snapped, which are bordered by a full circle or full ellipse.

1. In the Snap Menu, select the ***Quadrant Point*** command.

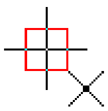
2. Identify the circumference of the circle or ellipse or the solid edge near the required quadrant point.

Please note:

Using ***Settings***, ***Snap Menu***, ***"Offer last Snap Menu option again"***, you can determine whether the snap point option ***Quadrant Point*** remains active after an endpoint is snapped.
It is deactivated if

- another snap point option is selected
- the mode (snap, tangential, perpendicular mode) is changed
- the current command is aborted

Snap intersection point (S, s)

 Snaps "real" intersection points of 2D line objects (line, infinite line, polyline, multi-line, circle, ellipse, spline, centre line). Both objects that form an intersection, must be within the red snap box.

1. In the Snap Menu, select the ***Intersection Pt*** command.

2. Identify the intersection point of two named objects.
   Position the cursor so that both of the objects project into the snapbox.

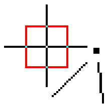
Please note:

Using ***Settings***, ***Snap Menu***, ***"Offer last Snap Menu option again"***, you can determine whether the snap point option ***Intersection Pt*** remains active after an endpoint is snapped.
It is deactivated if

- another snap point option is selected
- the mode (snap, tangential, perpendicular mode) is changed
- the current command is aborted

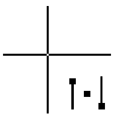
If the lines only have a virtual, therefore imaginary intersection, use the ***Intersect 2 Lines*** command.

Snap intersection of two lines

 Snaps "true" and "imaginary" intersection points of two 2D line objects (line, infinite line, polyline, multi-line, circle, ellipse, spline), lines in ***model views*** and in ***symbols***.

1. In the Snap Menu, select the ***Intersect 2 Lines*** command.
2. Identify the first object.
3. Identify the second object.
4. If one of the objects identified first is a circle or ellipse, at least two intersection points are possible.
   In this case, the possible intersection points are identified by markers.
   Identify one of the objects near the required intersection point.

Determine midpoint of two points

 Determines the point which lies halfway between the imaginary connecting line between two points.

1. In the Snap Menu, select the ***Midpoint 2 Points*** command.
2. Determine the first point.
   The commands ***Intersect 2 Lines, Perpendicular To*** and ***Tangent To*** cannot be used.

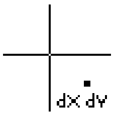
3. Determine the second point.
   The commands ***Intersect 2 Lines, Perpendicular To*** and ***Tangent To*** cannot be used.

The midpoint between both points is identified as the defined point by a marker.

Please note:

If the 3D mode is active, the midpoint between spatial points is also determined.

Define points relative to a point



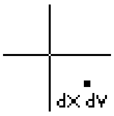
Defines a point using coordinates which do not refer to the origin but to any other point to be defined.

1. In the Snap Menu, select the ***Relative To Point*** command.
2. Determine the relative coordinate origin using the Snap menu or by making keyboard entries.
   Identify the reference point by pressing the ***Shift key*** if you wish to define the new point using polar coordinates.
   The commands ***Intersect 2 Lines***, ***Perpendicular To*** and ***Tangent To*** in the Snap Menu are not permitted in this case.
3. If needed, you can switch between cartesian and polar coordinates using the key combination ***Ctrl+p (Ctrl+P).*** This setting is only valid for active 2D windows and is active in the current model until it is changed.
   In the Status dialog box, enter the coordinates for the drawing area or the current WP.
   For Cartesian coordinates, the new point is determined by the axis distance to the reference point, for polar coordinates by the angle and distance.

Please note:

If, in the 3D-view window, commands are selected with which "real" spatial points can be defined, all three coordinate values can be entered in the Status dialog box. For this, the 3D coordinate reference must be active.

Define point relative to last point


Defines a point using coordinates which do not refer to the origin but to the point defined last.

1. In the Snap Menu, select the ***Relative To Last Point*** command.

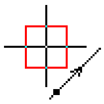
2. Define a point. With the shortcut ***Strg+p (Strg+P)*** you can switch between cartesian and polar coordinates.

3. In the Status dialog box, enter the coordinates for the drawing area or the current WP.
For Cartesian coordinates, the new point is determined by the axis distance to the reference point, for polar coordinates by the angle and distance.

Please note:

If, in the 3D-view window, commands are selected with which "real" spatial points can be defined, all three coordinate values can be entered in the Status dialog box. For this, the 3D coordinate reference must be active.
Using the menu command ***Settings, Snap Menu*** you can activate the ***Snap With Relative Coordinates*** mode for general point snapping. In this setting, the dX and dY coordinates in the Status dialog box always refer to the point defined last in the drawing.

Relative to footpoint


Relative to footpoint can be used to define a new point on a point on a circle / arc, ellipse / elliptical arc.

Relative to footpoint on lines

1. In the Snap Menu, select the command ***Relative to footpoint***.

2. Identify the line near the required footpoint (the nearest endpoint or the centre of the line is automatically snapped). The new point can then be dynamically moved to a temporary auxiliary line.

3. If necessary, in the ***Distance 2*** text box, define a parallel distance for the temporary auxiliary line to identify the object. The prefix for the entry defines the direction of the parallels.

4. If necessary, define an ***Angle 2*** by which the identified auxiliary line is to rotate about the determined point.

5. Define either
   - a ***Distance 1*** or
   - a percentual ***Position*** (in reference to the length of the line)
   to the selected footpoint.

Relative to footpoint on circles and arcs, ellipses and elliptical arcs

For non linear objects, there are two methods for defining the footpoint:

- on an (parallel) orbit or
- with the **Shift key** pressed, on a tangent auxiliary line.

Proceed as follows:

Orbit

1. In the Snap Menu, select the command ***Relative to footpoint***.
2. Identify a non linear object near the required footpoint (the nearest endpoint, quadrant point or centre of the circle, arc, ellipse or elliptical arc is automatically snapped). The new point can then be dynamically moved to an orbit.
3. If necessary, in the ***Distance 2*** text box, define a parallel distance for a parallel orbit to identify the object. The prefix for the entry defines the direction of the parallels.
4. Define either
   - a ***Distance 1*** or
   - a percentual ***Position*** (in reference to the length of the object) or
   - an ***Angle 2*** on the orbit
   to the selected footpoint.

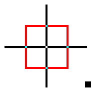
Tangent

1. In the Snap Menu, select the command ***Relative To Footpoint***.
2. Keep the ***Shift key*** pressed whilst identifying a non linear object near the desired footpoint (the nearest endpoint, quadrant point or centre of the circle, arc, ellipse or elliptical arc is automatically snapped). The new point can then be dynamically moved to a tangent.
3. If necessary, in the ***Distance 2*** text box, enter a value by which tangents are to be placed in reference to the footpoint. The prefix for the entry defines the direction.

4. If necessary, define
- an ***Angle*** in order to rotate the tangents to the selected point;
- an ***Angle 2*** in order to rotate the tangents to the selected object.

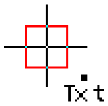
5. Define either
- a ***Distance 1*** or
- a percentual ***Position*** (in reference to the length of the line) to the selected footpoint.

Snap point

 Snaps a point which has been drawn in the drawing area using the command ***Draw 2D, Point***.

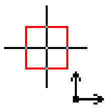
1. In the Snap Menu, select the ***Snap Point*** command.
2. Identify the point.

Snap text reference (R, r)

 Snaps the reference point of a text.

1. In the Snap Menu, select the ***Text Reference*** command.
2. Identify the required text.

Snap symbol reference

 Snaps the base point of a symbol.

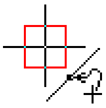
1. In the Snap Menu, select the ***Symbol reference*** command.
2. Identify an object of the required symbol reference point that is identified by a coordinate system.

Please note:

Using ***Settings***, ***Snap Menu, "Offer last Snap Menu option again"*** you determine whether the snap option ***Symbol Reference*** is to remain active after you have snapped a symbol reference.
It is not active if

- another snap point option is selected
- the mode (snap, tangent, perpendicular) is changed
- the current command is cancelled

Snap point / snap next point on line (N, n)

 Snaps the point on a 2D line object (line, circle, ellipse, spline) which lies nearest to the cursor position. For points, these themselves are snapped, for text the reference point.

If the 3D mode is active, the point is also determined on solid edges and 3D axes.

1. In the Snap Menu, select the ***Snap Point*** command.

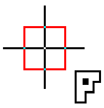
2. Identify the object.

Please note:

Using ***Settings***, ***Snap Menu***, ***"Offer last Snap Menu option again"***, you can determine whether the snap point option ***Snap Point*** remains active after an endpoint is snapped.
It is deactivated if

- another snap point option is selected
- the mode (snap, tangential, perpendicular mode) is changed
- the current command is aborted

Snap centre of gravity of face / model view (G, g)

 Determines the face centre of gravity of a 2D object of type 2D-face and the centre of gravity in a model view.

If the 3D mode is active in the 3D-view window, the centre of gravity is also determined on the flat solid surface.

1. In the Snap Menu, select the **Face/*Centre Of Gravity*** command.

2. Identify the 2D face / a planar solid face / a model view.

If the centre of gravity of a flat solid surface is to be determined, identify this face if the solid is to be displayed as shaded.
If only the edges of the solid are to be displayed, identify two of the bordering edges.

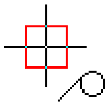
Please note:

Using ***Settings***, ***Snap Menu***, ***"Offer last Snap Menu option again"***, you can determine whether the snap point option ***Face/Centre Of Gravity*** remains active after a centre of gravity of an area is snapped

It is deactivated if

- another snap point option is selected
- the mode (snap, tangential, perpendicular mode) is changed
- the current command is aborted

Tangent to

 This command defines no fixed point, but determines a design condition for lines, circles and arcs.

It can be used to define a line, vector or axis tangent to a circle or ellipse. A circle (arc) can be defined tangent to an existing line or another circle.

After activating the condition, the cursor is displayed with an additional symbol - as above.

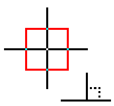
1. In the Snap Menu, select the ***Tangent To*** command.
2. Identify a 2D line object near the required tangent point.
3. Define further parameters for the line, circle or arc.

Please note:

This mode can also be activated by the key combination ***Ctrl-t, Ctrl-T*** or using the menu command ***Settings, Point Definition***. This however, is not only valid for the definition of a point but is permanent.
By pressing the ***f, F*** key or using the menu command ***Settings, Point Definition*** the "snap" mode can be activated again.

If the determination of the tangent point is to be made possible using the endpoint of the first identified line or arc, whilst defining the further points, keep the Shift key pressed.

Perpendicular to

 This command defines no fixed point but determines a geometric design condition for lines.

It makes it possible to define a line, vector, or axis in the right angle to a 2D line object (line, circle, ellipse).

After activating the condition, the cursor is displayed with an additional symbol - as above.

1. In the Snap Menu, select the ***Perpendicular To*** command.

2. Identify a 2D line object.

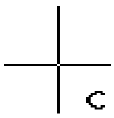
3. Define the endpoint dynamically. The perpendicular can be moved to the identified 2D line object.
If this is carried out dynamically with the cursor, the design or movement using endpoints of lines or arcs is possible, if the Shift key is kept pressed whilst doing so.

Please note:

This mode can also be activated by the key combination ***Ctrl-t, Ctrl-T*** or using the menu command ***Settings, Point Definition***. This however, is not only valid for the definition of a point but is permanent.
By pressing the ***f, F*** key or using the menu command ***Settings, Point Definition*** the "snap" mode can be activated again.

If the determination of the perpendicular point is to be made possible using the endpoint of the first identified line or arc, whilst defining the further points, keep the Shift key pressed.

Define point using cursor (C, c)


This can be used to define a point freely with the cursor. I.e. without taking any of the settings determined in the ***Point filter*** block using the menu command ***Settings, Point Definition***. Therefore, the point can, e.g. be defined near a line without the endpoint being snapped.

After activating the command, the cursor is displayed as a simple cross-hair cursor.

1. In the Snap Menu, select the ***Cursor*** command.

2. Place the cursor on the required position.
Confirm the position by pressing the left mouse button.

Please note:

Using ***Settings***, ***Snap Menu***, ***"Offer last Snap Menu option again"***, you can determine whether the snap point option ***Cursor*** remains active after an endpoint is snapped.
It is deactivated if

- another snap point option is selected
- the mode (snap, tangential, perpendicular mode) is changed

- the current command is aborted

Origin 2D

Allows to snap the current origin (absolute or relative) in the 2D area or on the active workplane (3D).

Snap a point using point filter

This can be used to suppress any commands which may have been selected in the Snap Menu previously and reactivate all snap commands of the point filter.

These are set using the menu command ***Settings, Point Definition***.

Define a point by entering coordinates

When you are drawing with the cursor, you can enter the coordinates for the point in the boxes *X, Y* in the Status dialog box.
If you first activate a polar coordinates system by using ***Settings, Point Definition***, you can specify the values in the ***Pol. dist.*** and ***Pol. angle*** boxes.

These values are in relation to the origin of the global coordinates system, provided that it is activated in the main symbols bar:

Please note:

This method of snapping the points can be used when you are working with a points filter or you have activated a command in the Snap Menu.
It cannot be used, though, with the four commands ***Intersect 2 Lines***, ***Centre of Gravity***, ***Perpendicular to***, ***Tangent To***.

Determine settings for snapping object points

As well as the Snap Menu (using middle mouse button or Ctrl + right mouse button) and the hotkeys (refer to ***keyboard commands***) you also have the option of activating general snap points. These are always active and the snap point which lies nearest to the activated snap options is snapped. However, if you select a snap option using the Snap Menu or the keyboard, in the selected command, this holds a higher priority than the generally set snap points. The general snap points become active again as soon as a new command is started or the ***Point Filter*** is selected using the Snap Menu.

General snap points are activated as follows:

1. Start the menu command *Settings, Point Definition*.

2. Select the option card ***Snap mode***
Settings for the definition of points using the cursor can be determined. These are permanently active, i.e. until modified again.

Independent of the settings determined here, individual points can be determined using the Snap Menu.

The following options are available:

<u>Snap with absolute coordinates, F, f</u>

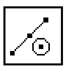
<u>Snap with relative coordinates, Ctrl+F, Ctrl+f</u>

The snap mode is the standard mode for the definition of points with the cross-hair cursor as well as for snapping points on objects.

Depending on the setting you have selected, with ***Snap With Absolute Coordinates*** for the definition of coordinate points, the values X and Y are displayed in the Status dialog box in reference to the absolute origin. If you have selected the ***Snap With Relative Coordinates*** setting, the coordinates in the dX and dY fields refer to the point last defined in the drawing. This setting is highly recommended for the definition of 2D polygons using the Status dialog box.

This mode can be activated in the dialog box and also whilst the cursor is in the drawing area or the current WP by pressing the ***F, f*** key.
The mode remains active until one of the other two modes are selected.

If the ***snap*** mode is active, the right side of the option card can be used to determine whether the points are to be defined with the simple cross-hair cursor, or whether a **point filter** is to be activated. This enables automatic snapping of points on objects:

<u>Point filter</u> 

If the standard mode ***Snap*** is selected, a point filter can be determined, i.e. which types of points on objects are to be snapped automatically.

If a point filter is active, the cursor is displayed with an additional symbol as illustrated.

Activating the point filter
There are two options for activating the point filter:

- Activate the check box ***all active****.*
  This activates all the settings available in the point filter block for snapping points on objects.

A different setting for individual points is not modified by this, i.e. it is still available after deactivating.

- Deactivate both check boxes ***all active*** and ***none active***.
  This enables a differing setting to be determined for which points should be snapped.

When identifying, an object is determined which lies within the snapbox of the cursor.
If the point filter contains a setting for a point on this object type, this point is snapped, e.g. the nearest endpoint of a line or the nearest quadrant point to a full circle.

If an object lies within the face of the snapbox which does not meet any setting on the point filter, a message is displayed stating that the point could not be found. This is also valid if no object lies within the snapbox.

Deactivating the point filter
There are two options for deactivating the point filter:

- Activate the check box ***none active***.
- Deactivate both check boxes ***all active*** and ***none active***.
  Then deactivate all settings for the individual types of points.

If the point filter is to be deactivated for one point only, select the command ***Cursor*** in the Snap Menu, or activate directly by typing in ***c, C***.

Please note:

If a number of settings are activated for one object type in the point filter, e.g. ***Centre*** and ***Quadrant point*** for full circles, the determination is carried out in a specific order.
For the named combinations, the nearest quadrant point is snapped.

If this is not required, modify the point filter as necessary or, in such a case, use the Snap Menu command ***Centre***.

Auto Snap, A, a

This mode is for snapping endpoints, centres, midpoints, quadrant points and intersections on 2D linear objects.
If there are 2D line objects in the area covered by the snapbox while the cursor is being moved, the point that is determined will be tagged.
After you have acknowledged by clicking the left mouse button, the point will be snapped.

This mode can be activated in two ways. You can use this dialog box as described already. In addition, provided that the cursor is in the drawing area or in the current WP, you can type ***A,*** or ***a***.
The mode remains active until you active one of the other modes.

Tangent, T, t 

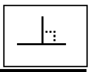
If this mode is active, when defining lines and circles as well as vectors and axes, the tangent point is determined which nearest to the cursor on a 2D line object (circle, arc, ellipse, elliptical arc).

This mode can be activated in the dialog box and also whilst the cursor is in the drawing area or the current WP by pressing the key combination ***Ctrl+T, Ctrl+t***.
The mode remains active until one of the other two modes are selected.

If this mode is active, the cursor is displayed with an additional symbol as illustrated.

Please note:

- When you are using this mode, a design condition is defined.
  It cannot be used for the definition of individual points since a direction is required for this.
- For arcs and elliptical arcs, a tangent point is usually determined on the arc line. If this determination is also to be carried out in the imaginary extension of this object, keep the Shift key pressed whilst moving the cursor.
- If this mode is not to be used permanently, the Snap Menu command ***Tangent To*** can be used.

Perpendicular, L, l 

If this mode is active, when defining lines and circles as well as vectors and axes, the perpendicular point is determined which is nearest to the cursor on a 2D line object (circle, arc, ellipse, elliptical arc).
This mode can be activated in the dialog box and also whilst the cursor is in the drawing area or the current WP by pressing the key combination ***Ctrl+L, Ctrl+l***.
The mode remains active until one of the other two modes is selected.

If this mode is active, the cursor is displayed with an additional symbol as illustrated.

Please note:

- When you are using this mode, a design condition is defined.
  It cannot be used for the definition of individual points since a direction is required for this.
- The perpendicular point is usually determined on the objects. If this determination is also to be carried out in the imaginary extension of this object, keep the Shift key pressed whilst moving the cursor.
- If this mode is not to be used permanently, the Snap Menu command ***Perpendicular To*** can be used.

Offer last Snap Menu option again

Using this switch, define whether snap points which have been activated using the Snap Menu or the keyboard are to be used for further snap options in the active command.

If you have deactivated the control field, once a point is snapped, the previous status is reset. If no general snap points are activated, the cursor mode is then active again and another snap point must be reactivated using the Snap Menu or the keyboard.

Determine settings for point definition using coordinates

The Status dialog box is not only for display purposes, but can also be used for the entry of numeric values, also for the entry of coordinate values.
The coordinates values which can be displayed and entered in the Status dialog box depend on the settings which are determined using the menu command ***Settings, Point Definition*** on the ***Coordinates*** option card.

System
The radio buttons under this heading can be used to determine whether the coordinate values to be displayed or entered are to be ***cartesian*** coordinates or ***polar*** coordinates.
Activate the required radio button.

Zero point
An angle value can be determined here which determines the 0° direction when using polar coordinates.
This value can be entered directly in the respective text box, or can be determined after activating the ***Determine*** command button.
To determine the angle, define a point in the drawing area or in the current WP of the 3D window. The angle from the current origin to the

defined point is determined and then entered in the text box in the dialog box.

Please note:

The coordinate values to be displayed or to be entered refer to the defined origin of the currently selected GCS or LCS.

After you have selected ***Relative to point*** or ***Relative To Last Point*** in the Snap Menu, the defined reference point is naturally valid as the origin for coordinate entry.

Depending on the selected setting, the following values are displayed or can be entered:

X, Y, dX, dY
X and Y coordinates (= Cartesian coordinates) of the current cursor position or of the point to be defined, in reference to the currently selected origin or the previously defined reference point.

Pol. dist., Pol. angle
Polar coordinates of the current cursor position or of the point to be defined, in reference to the origin of the currently selected coordinate system or the previously defined reference point.

Change circle into ellipses when scaling
If this setting is active using the control box, circles are changed into ellipses if their x / y factors are not equal.

4.4 Draw 2D

 Alternatively, the commands for drawing can be activated by clicking on this icon in the toolbar. These commands will generate 2D objects (line, full circle, full ellipse, spline, point).

The objects so generated will be integrated into the structure of a model and drawn:

Drawing, Partial Drawing, Workplane
The objects will be saved in the active partial drawing of the drawing currently selected or in the workplane currently selected (WP) of the 3D-view window.

If the 2D objects are drawn in a WP of the current 3D window, they are linked to this WP. If the WP is processed or erased, all the 2D objects contained on this WP will also be erased.

If the 3D-view window is activated, points, which are defined for drawing objects or snapped on solid edges, are generally projected in the selected WP.

The generated 2D objects are included and drawn into the structure of a model in the following way:

Group
The generated objects are saved in the **current group**. This can be selected from the existing groups using the group selection of the main toolbar.
Group allocation can later be modified in the ***Model Explorer***.

Colour, line type, line width
The 2D objects drawn with the commands in this menu are linked with the **current drawing layer**. These can be selected using the layer selection from the main toolbar from the defined layers or using an object in the drawing.

Objects are allocated the display attributes colour, line type and line width which are currently set for the layer, unless other settings have been made using the top toolbar.
This is not valid, if the display attributes are defined as object specific using the main toolbar.

The display of line type and line width correspond to those settings determined using the menu command ***Settings, Options***.

The link with a layer as well as colour, line type and line width can later be modified using the menu command ***Process, Object Display***.

To ensure that whilst processing your drawing, only specific object types can be identified, deactivate the snap options for all other object types using the menu command ***Settings, Object Filter***.

Determine settings for draw commands

Using the menu commands ***Settings, 3D/2D Commands****,* the ***2D draw*** option card can be used to determine presettings for the *Draw 2D, Blend* or *Chamfer* and *Polygon* commands:

blend radius
Defines the preset value of the radius for the ***Draw 2D, Blend*** command.
Once the command has been started, this value is displayed in the Status dialog box, if this check box in the dialog box is active.
If this check box is active, once the command is started, the radius value is set to 0, i.e. the radius can be defined dynamically or by entering a numeric entry.

chamfer length and **chamfer angle**
Defines the preset values for the ***Draw 2D, Chamfer*** command.

Once the command has been started, these values are displayed in the Status dialog box, if the check box in the dialog box is active.
If this check box is active, once the command is started, the values are set to 0, i.e. the value can be defined dynamically or by entering a numeric entry.

Polygons with … corners
Defines the preset values for the number of corners of a polygon.

Once the command has been started, this value is displayed in the Status dialog box.

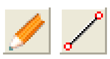
Angle grid for drawing elements
If this setting is active in the check box, you cannot generate lines and partial lines in polylines drawn with the cursor except at angles that are a multiple of the value that is specified.
This setting will be ignored, if the Snap Menu has been used to snap a point on a line that already exists or defined using the Status dialog box of another angle or by pressing the ***Shift key***, while the line object is on the cursor.

Angle grid for construction aids
If this setting is active in the check box, you cannot generate construction lines, partial lines on construction polylines and construction lines drawn with the cursor except at angles that are a multiple of the value that is specified.
This setting will be ignored, if the Snap Menu has been used to snap a point on an object that already exists or defined using the Status dialog box of another angle or by pressing the ***Shift key***, while the line object is on the cursor.

transform circles into ellipses in case of anisotrop scaling
If this option is active, circles are transformed to ellipses when the x /y scaling factors are different.

Circle definition by
Determines whether the circle definition for 2D objects and solids should be made by radius or diameter. This setting affects the construction of circles and cylinders, arcs and spheres as well as the circle definition when multiplying by circular spacing.

Draw line

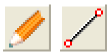
 This command can also be activated using this icon.

This command can be used for drawing 2D objects in the line category. If you are going to use the cursor to do this as a series of specific

angles, you must first select ***Settings, 3D/2D Commands***, and then use the option card ***2D Drawing*** to specify the ***Angle grid for drawing elements***.
When you are using the Snap Menu or by pressing the ***Shift key*** the angle grid will be ignored.

The following section presents some examples for drawing lines:

Horizontal, perpendicular line

 This command can also be activated using this icon.

Lines which run parallel to the first (horizontal) or second (vertical) axis.

Proceed as follows:

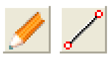
1. Define the start point.

2. Using the cursor, change to the drawing area or to the required WP.
Press the letter ***H, h*** (horizontal) on the keyboard, if a horizontal line is to be drawn, ***V, v*** (vertical), if a vertical line is to be drawn.

3. Determine the endpoint of the line.
This can be determined using the cursor, by entering the X coordinate for a horizontal line, the Y coordinate for a vertical line or by entering the required length.

Please note:
As an alternative you can activate the option card ***2D Drawing*** and specify an ***Angle grid for drawing elements*** at 90° before using ***Settings, 3D/2D Commands*** to start the command.

Line using absolute coordinates

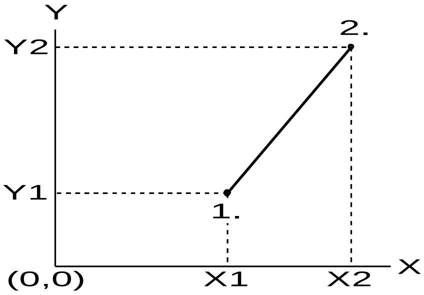
 This command can also be activated using this icon.

Line, whose start and endpoints are defined using absolute coordinates.

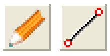
Proceed as follows:

1. In the Status dialog box, in the ***X*** and ***Y*** text boxes (***Pol. dist.***, ***Pol. angle*** with polar coordinates) enter the coordinates for the start point.
Confirm the entries with *Enter*.

2. In the Status dialog box, in the ***X*** and ***Y*** text boxes (***Pol. dist.***, ***Pol. angle*** with polar coordinates) enter the coordinates for the endpoint.



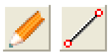
Line create by midpoint

 After clicking the displayed icons the command ***Draw 2D, Line*** is started.

To create a line by midpoint please follow these stepps:

1. Click the ***X*** key to create the line by midpoint.
2. Define the midpoint of the line by entering the coordinates in the status dialogue window or by point definition.
3. Define the endpoint of the line.

Line using point and relative coordinates

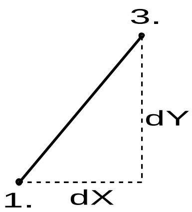
 This command can also be activated using this icon.

Line whose first point is defined in any way and whose second point is defined by coordinates which refer to the first point.

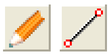
Proceed as follows:

1. Define the start point.
2. In the Snap Menu, select ***Rel To Last Point***.
   The previously defined point is identified by a marker.
3. In the Status dialog box, in the ***dX*** and ***dY*** text boxes (***Pol. dist***., ***Pol. angle*** with polar coordinates) enter the coordinate values which refer to the first point, i.e. X and Y distance (angle and distance) from the start point.

   If only one of these values is entered, the respective other value can be entered dynamically using the cursor.



Line using point, angle and length

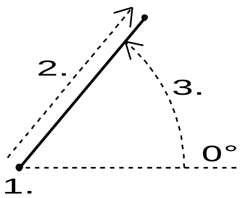
 This command can also be activated using this icon.

Line whose start point is defined in any way and whose endpoint is determined by entering an angle and length.

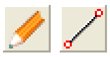
Proceed as follows:

1. Define the start point.
2. In the ***Length*** and ***Angle*** text boxes in the Status dialog box, enter the required values.

   If only one of the values is entered, the other value can be determined dynamically using the cursor.
   I.e. after entering the length, the line can be rotated using the cursor, after entering the angle, the length of the line can be modified in any way.



Line perpendicular to a line

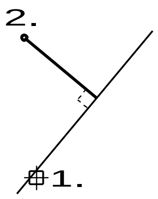
 This command can also be activated using this icon.

Proceed as follows:

Right angle at start point:

1. Load the Snap Menu and select ***Perpendicular To***.
   By the cross-hair cursor, the additional symbol is displayed for 'perpendicular'.
2. Identify the base line, i.e. the line to which the new line is to run perpendicular to.
   In the ***Distance 1*** text box, the distance of the start point to the identified endpoint of the base line can be determined.
3. Define the endpoint of the new line.

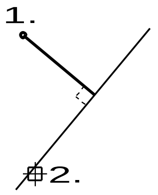
   If the ***Distance 1*** is not entered and the endpoint is determined dynamically with the cursor - whilst keeping the Shift key pressed - the line can be projected using the endpoint of the previously defined base line.



Right angle at endpoint:

1. Define the start point of the line.
2. Load the Snap Menu and select ***Perpendicular To***.
   By the cross-hair cursor, the additional symbol is displayed for 'perpendicular'.
3. Identify the target line, i.e. the line on which the new line is to end perpendicular to.

   If the bottom point of the perpendicular lies outside the target line, an error message is displayed.
   If the line is to be generated anyway, keep the Shift key pressed whilst identifying the target line.



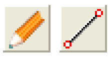
Please note:

If - instead of selecting the ***Perpendicular To*** command in the Snap Menu - the ***L*** key is pressed, the ***Perpendicular To*** condition is not only valid for the next point to be defined, but is permanent. By entering ***F*** (switch to "normal" snap mode) it can be deactivated again.

Centre perpendicular

Tod draw a Centre perpendicular to an existing line, use the following procedure:

Option 1:

1.  Select ***Draw 2D, Line***.
2. Select the command ***Perpendicular To*** in the Snap Menu and then identify the base line.
3. Press the right-hand mouse button once to break off the line definition.

4. Activate the Status dialog box and then accept the value shown against ***Angle*** by pressing Enter.
5. Select the command ***Midpoint*** in the Snap Menu and identify the base line.
6. Define the second point for the line.

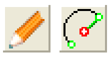
Option 2:

1. Select the command ***Perpendicular To*** in the Snap Menu and then identify the base line. The endpoint is marked.
2. Move the cursor with the perpendicular to the other endpoint of the line.
   In the Status dialog box in the field ***Distance*** the distance between the perpendicular and the marked endpoint is shown. This distance is equivalent to the length of the base line.
3. Activate the ***Distance*** field in the Status dialog box and presee the End key to move the cursor to the end of the value.
   Add the formula /2, e.g. 100.00/2, and press the Enter key.
4. Define the second point of the line.

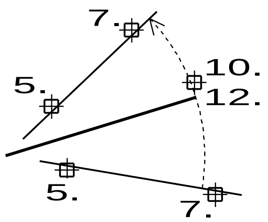
Bisector

Bisector of two existing lines.

Either use the ***Bisector*** command in the ***Const Aids*** menu as the basis, or proceed as follows:

1. Select the menu command ***Settings, Point Definition, Snap mode***.
2. Select the ***snap*** mode, deactivate the options ***all active*** and ***none active***, and in the point filter, activate the options ***Endpoint*** and ***Midpoint***.
3.  Select the command ***Draw 2D, Arc Dynamic***.
4. Select the command ***Intersect 2 Lines*** in the Snap Menu.
5. Identify the endpoints of the two lines close to their intersection.
6. Define the radius of the circle (any radius) by means of a point or by entering a value in the Status dialog box.
7. Identify each of the other endpoints of the two lines for the start and end angles of the arc.
   It is important to remember that the arc will be constructed anti-clockwise.

8. Select the command ***Draw 2D, Line***.
9. Select the command ***Midpoint*** in the Snap Menu.
10. Identify the arc close to its midpoint.
11. Select the command ***Centre*** in the Snap Menu and then identify the arc.
12. Erase the arc.



Line tangent to circle or ellipse

This command can also be activated using this icon.

Proceed as follows:

1. Load the Snap Menu and select ***Tangent To***.
   By the cross-hair cursor, the additional symbol is displayed for 'tangent'.
2. Identify the circle or ellipse near the required tangent point.
   If the tangents are to run under a specific angle, enter the required value in the ***Angle*** text box of the Status dialog box.
3. If the endpoint of the line is also to be the tangent point to a circle or ellipse, repeat both steps. Otherwise, define the second point of the line with one of the other options.

   If the endpoint is to be defined using the cursor, the line can be projected over the endpoints of the arc by keeping the Shift key pressed.

Line perpendicular to circle, ellipse (normal)

This command can also be activated using this icon.

Proceed as follows:

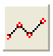
1. Load the Snap Menu and select ***Perpendicular To***.
   By the cross-hair cursor, the additional symbol is displayed for 'perpendicular'.

2. Identify the circle, ellipse to which the line is to run perpendicular to.
   Pick the circumference near the required bottom point of the perpendicular.
   In the ***Distance 1*** text box of the Status dialog box, the distance of the bottom point to the nearest quadrant point or endpoint (for arcs) can be predefined.
3. If the line endpoints are also to run perpendicular to a line, circle or ellipse, repeat both steps.

   If the ***Distance 1*** is not predefined and the endpoint is defined dynamically using the cursor, the line can be projected over the endpoints of the circle or arc by keeping the Shift key pressed.

   If, for circles and arcs, the cursor is moved out of the circle half picked, the start point of the line is moved to the "opposite" half of the circle.

Draw polyline

  This command can also be activated using this icon.

This command can be used to draw polylines made up of 2D objects of the type line and will be created in a group. This group is created below the current partial drawing / group and gets the command name as group name.
Using the menu command ***Settings***, ***3D / 2D Commands*** in the tab ***2D Draw*** you can globally activate if the elements of ***Polylines***, ***Rectangles*** and ***Polygons*** should be created in a group.

A polyline consists of at least 2 partial lines, whereby the endpoint of a partial line is also the start point of the following line.
Therefore, when defining the first partial line, the start and endpoints need to be defined, but only the endpoint of the following lines.

For the transition from one line to the next a rounding can be defined by entering a ***Radius*** in the status dialogue window. With this radius all transitions are rounded, as long as the radius can be defined due to the minimum length of a line.

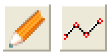
If you are going to use the cursor to draw in a series of angles, first select the menu command ***Settings, 3D/2D Commands***, and then use the option card ***2D Drawing*** to specify ***Angle grid for drawing elements***.

When you are using the Snap Menu or by pressing the ***Shift key*** the angle grid will be ignored.

Each partial line of a polyline can be processed individually.

The following section presents some examples for drawing polylines:

Polyline using absolute coordinates

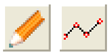
 This command can also be activated using this icon.

Start and endpoints of partial lines of a polyline are defined using absolute coordinates.

Proceed as follows:

1. In the Status dialog box, in the ***X*** and ***Y*** text boxes (***Pol. dist., Pol. angle*** with polar coordinates) enter the coordinates for the start point of the first partial line.
2. In the Status dialog box, in the ***X*** and ***Y*** text boxes (***Pol. dist., Pol. angle*** with polar coordinates) enter the coordinates for the endpoint of the first partial line.
3. In the Status dialog box, in the ***X*** and ***Y*** text boxes (***Pol. dist., Pol. angle*** with polar coordinates) enter the coordinates for the endpoint of the next partial line(s).

Polyline using relative coordinates

 This command can also be activated using this icon.

The endpoint of each partial line is defined using coordinates which refer to the start point of the partial line.

Proceed as follows:

1. Define the start point of the first partial line.
2. In the Snap Menu, select ***Rel To Last Point***.
   The origin, i.e. the point to which the coordinate values refer, is temporarily moved to the last defined point.
   The point is identified by a marker.
3. In the Status dialog box, in the ***dX*** and ***dY*** text boxes (***Pol. dist., Pol. angle*** with polar coordinates) enter the coordinate values for the endpoint of the first partial line, i.e. X and Y distance (angle and distance) from the previously defined point.

Repeat steps 2 and 3 for further partial lines.

Polyline using angle and length

This command can also be activated using this icon.

Each partial line is defined by an angle and length.

Proceed as follows:

1. Define the start point of the first partial line.
2. In the ***Length*** and ***Angle*** text boxes in the Status dialog box, enter the required values for the first partial line.
   The angle always refers to the direction of the X-axis of the global coordinate system.
3. In the ***Length*** and ***Angle*** text boxes in the Status dialog box, enter the required values for the next partial line.

Repeat steps 2 and 3 for further partial lines.

Draw rectangle

This command can also be activated using this icon.

This command can be used to draw rectangles made up of 2D objects of the type line and will be created in a group. This group is created below the current partial drawing / group and gets the command name as group name.
Using the menu command ***Settings***, ***3D / 2D Commands*** in the tab ***2D Draw*** you can globally activate if the elements of ***Polylines***, ***Rectangles*** and ***Polygons*** should be created in a group.

After starting the command a ***radius*** can be defined in the status block for the edges of the rectangle. These are created as long as the side length of the rectangle is at least double the value of the entered radius.

For the definition of a rectangle, two diagonally opposite corner points as well as the (rotation) angle, if necessary, should be defined. A rectangle can also be defined by 3 points, when first the 2 points in one axis are defined and then the third point in the second axis.

Each line of a rectangle can be processed individually.

The following presents some examples for drawing rectangles:

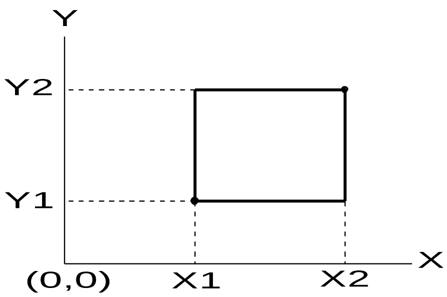
Rectangle using coordinates

This command can also be activated using this icon.

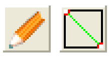
Rectangle corners are defined using coordinates.

Proceed as follows:

1. In the Status dialog box, in the ***X*** and ***Y*** text boxes (***Pol. dist., Pol. angle*** with polar coordinates) enter the coordinates for a corner.
2. If the rectangle sides are not to be parallel to the coordinate axes, enter the angle in the ***Angle*** text box under which the base line of the rectangle is to run.
   The angle always refers to the direction of the X-axis of the global coordinate system.
3. In the Status dialog box, in the ***X*** and ***Y*** text boxes (***Pol. dist., Pol. angle*** with polar coordinates) enter the coordinates for the diagonally opposite corner.



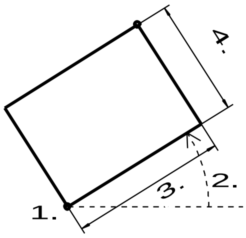
Rectangle using length and width

 This command can also be activated using this icon.

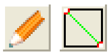
The rectangle is defined by the length and width, i.e. the length of the sides.

Proceed as follows:

1. Define a corner of the rectangle.
2. If the rectangle sides are not to be parallel to the coordinate axes, enter the angle in the ***Angle*** text box under which the base line of the rectangle is to run.
   The angle always refers to the direction of the X-axis of the global coordinate system.
3. In the ***Length X*** and ***Length Y*** text boxes, enter the values for the lengths of the sides.

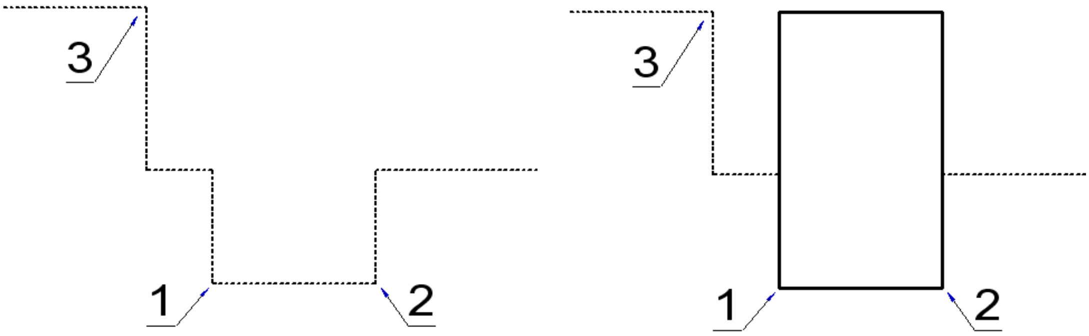


Define Rectangle by 3 Points

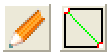
 After clicking the displayed icons the command ***Draw 2D***, ***Rectangle*** is started.

To define a rectangle by 3 points follow these steps:

1. Snap the first point. In the example below using ***Endpoint*** or ***Intersection Point***.
2. Snap the second point. In the example below again the ***Endpoint*** or ***Intersection Point***. If the second point is in line with the first point, so that the value for the x or y length of the rectangle would be 0, the second axis can be defined by a third point.
3. Snap the third point. gen Sie den dritten Punkt. In the example below again the ***Endpoint*** or ***Intersection Point***. Alternatively a length for the second axis can be entered in the status dialogue window.



Create Rectangle by Centre Point

 After clicking the displayed icons the command ***Draw 2D, Rectangle*** is started.

To create a rectangle starting from the centre point please follow these steps:

1. Click the ***X*** key to create the rectangle starting from the centre point.
2. Define the centre point of the rectangle by entering the coordinates in the status dialogue window or by point definition.
3. Define the end point of the rectangle by entering the coordinates in the status dialogue window or by point definition or enter the ***Angle*** and the ***Length X*** and ***Length Y*** in the status dialogue window.

Draw polygon

A regular polygon is defined by entering the number of corners, the position of the centre as well as the diameter of the in circle and circumcircle.

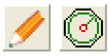
This command can be used to draw polygons made up of 2D objects of the type line and will be created in a group. This group is created below the current partial drawing / group and gets the command name as group name.
Using the menu command ***Settings***, ***3D / 2D Commands*** in the tab ***2D Draw*** you can globally activate if the elements of ***Polylines***, ***Rectangles*** and ***Polygons*** should be created in a group.

Each line of a polygon can be processed individually.

Using the menu command ***Settings, 3D/2D Commands*** the ***2D draw*** option card, the number of corners can be predefined.

This presetting can be taken over when executing the command or modified if necessary.

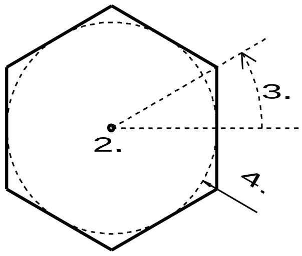
Polygon using incircle

 This command can also be activated using this icon.

With this command, the polygon is determined using the radius of the incircle (= half width of jaw) and the number of corners.

1. If the value suggested in the ***Number*** text box of the Status dialog box is not of the required value, modify this value.
2. Define the centre of the imaginary incircle.
3. Enter the rotation angle by which the corner is to be rotated in the ***Angle*** text box or determine dynamically.
   The angle always refers to the direction of the X-axis of the global coordinate system.

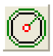
4. Enter the value for the ***Radius 1***, or determine the radius by defining a point on the circumference of the imaginary incircle.



Please note:

After defining the centre point with the hotkey ***X*** it can be defined to use the Incircle or the Circumcircle.

Polygon using circumcircle

  This command can also be activated using this icon.

With this command, the polygon is determined using the radius of the circumcircle (= half width across corners) and the number of corners.

1. If the value suggested in the ***Number*** text box of the Status dialog box is not of the required value, modify this value.
2. Define the centre of the imaginary circumcircle.
3. Enter the rotation angle by which the corner is to be rotated in the ***Angle*** text box or determine dynamically.
   The angle always refers to the direction of the X-axis of the global coordinate system.
4. Enter the value for the ***Radius 1***, or determine the radius by defining a point on the circumference of the imaginary circumcircle.

Please note:

After defining the centre point with the hotkey ***X*** it can be defined to use the Incircle or the Circumcircle.

Draw circle and arc

The commands defined in the following can be used to draw full circles as well as arcs.

With full circles, only the ***Centre*** and ***Quadrant point*** can be snapped, with arcs, ***Centre***, ***Quadrant point***, ***Midpoint*** and ***Endpoint***.

Draw circle (arc) dynamic

This command can also be activated using this icon.

This command draws full circles and arcs.

Full circles can be defined using the centre and a point or tangent condition as well as by entering a radius.
Arcs are defined by the additional entry of a start and opening angle.

The following presents some examples for drawing circles or arcs:

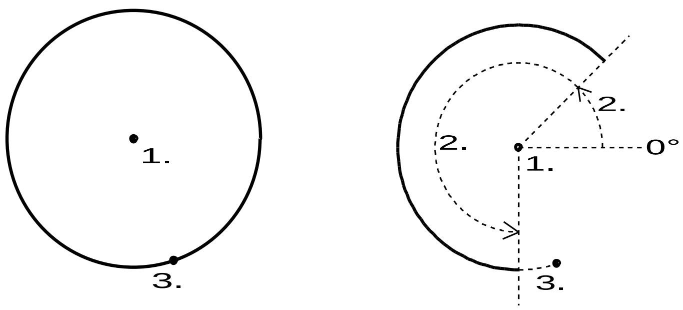
Circle (arc) using centre and point

This command can also be activated using this icon.

A circle (arc) is defined using the centre and a point on the circumference.

Proceed as follows:

1. Define the centre.
2. If a full circle is to be generated, enter the start angle of the arc in the ***Angle*** text box.
   The angle entered always refers to the direction of the X-axis of the global coordinate system.
   In the ***Open.angle*** text box of the Status dialog box, enter the opening angle of the arc.
3. Define a point on the circumference.



Circle (arc) using centre and diameter

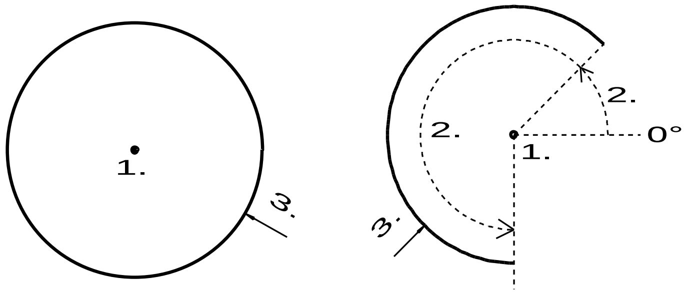
This command can also be activated using this icon.

A circle (arc) is defined using the centre and the radius.

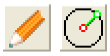
Proceed as follows:

1. Define the centre.

2. If an arc is to be generated, enter the start angle of the arc in the ***Angle*** text box.
   The angle entered always refers to the direction of the X-axis of the global coordinate system.
   In the ***Open.angle*** text box of the Status dialog box, enter the opening angle of the arc.
3. In the ***Diameter 1*** text box, enter the diameter.



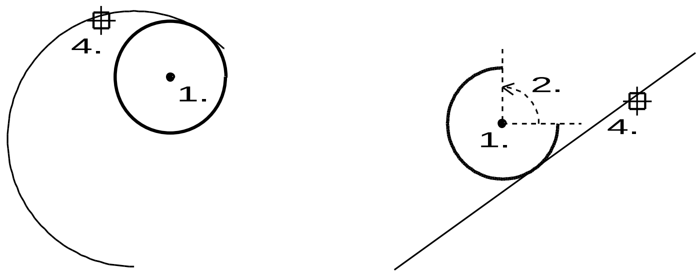
Circle (arc) using centre and tangent to a line

 This command can also be activated using this icon.

A circle (arc) is defined using the centre and by being tangent to an existing line (line, circle, arc).

Use one of the following options:

1. Define the centre.
2. In the Snap Menu, select ***Tangent To***.
   By the cross-hair cursor, the additional symbol is displayed for 'tangent'.
3. Identify the line or circle near the required tangent point.

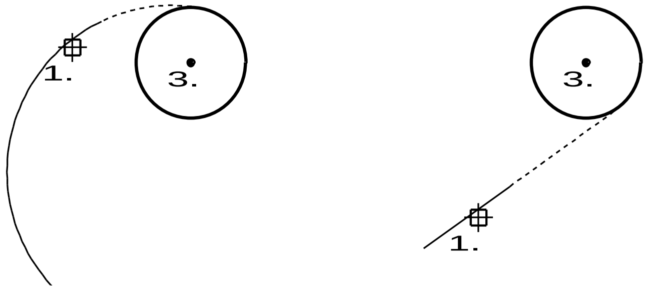


or

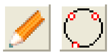
1. In the Snap Menu, select ***Tangent To***.
   By the cross-hair cursor, the additional symbol is displayed for 'tangent'.

2. Identify the line or circle near the required tangent point.
3. Define the centre of the new circle.

   If the centre of the circle is defined dynamically, the circle can be projected over the endpoints of the identified line/arc by keeping the Shift key pressed whilst moving.



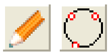
Full circle using 3 points

 This command can also be activated using this icon.

Full circles are clearly determined by three points which do not lie on an infinite line.

The following presents some examples for drawing circles:

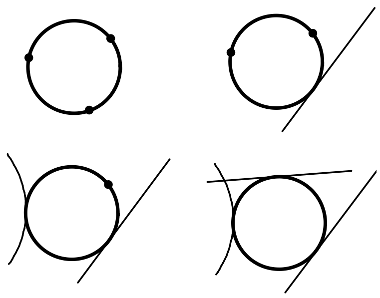
Full circle using points/tangent to lines

 This command can also be activated using this icon.

A circle is determined using three points. These can also be tangent points to existing lines, circles or arcs.

Therefore, this command enables circles to be defined

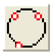
- using three points
- using two points and tangent to a line
- using one point and tangential to two lines
- tangent to three lines



Determine the points or tangent points on the lines, circles or arcs in the combination desired:

- A point can be defined dynamically, using the Snap Menu or by entering a value in the Status dialog box.
- If the circle is to run tangent to a line or circle (arc), load the Snap Menu and select ***Tangent To***.
  By the cross-hair cursor, the additional symbol is displayed for 'tangent'.
  Identify the line/circle (arc) near the required tangent point.
  If the position of the circle is not yet completely determined - whilst keeping the Shift key pressed - it can also be projected over the endpoints of a line/arc.

Full circle using radius, points/tangent to lines

  This command can also be activated using this icon.

A circle is defined using the radius, two points and the position in reference to the two points. The points can also be tangent points to existing lines, circles or arcs.

Therefore, this command enables circles to be defined

- using two points
- using a point and tangent to a line
- tangent to two lines

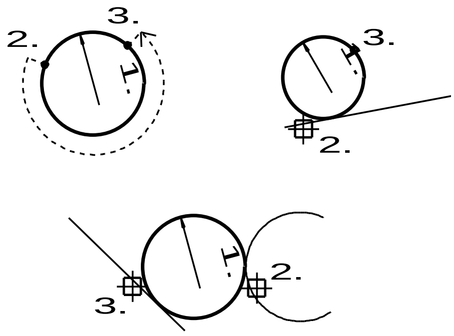
as well as by entering the radius and determining the position.

1. Determine the points or tangent points on the lines, circles or arcs in the combination desired:

- A point can be defined dynamically, using the Snap Menu or by entering a value in the Status dialog box.
- If the circle is to run tangent to a line or circle (arc), load the Snap Menu and select ***Tangent To***.
  By the cross-hair cursor, the additional symbol is displayed for 'tangent'.
  Identify the line/circle (arc) near the required tangent point.
  If the position of the circle is not yet completely determined - whilst keeping the Shift key pressed - it can also be projected over the endpoints of a line/arc.

2. Position the cursor - without pressing a mouse button - so that the circle has the required position.

3. Change to the Status dialog box and enter the radius of the circle in the ***Radius 1*** text box.



Arc dynamic

This command can also be activated using this icon.

Arcs can be defined using the centre and a point, by defining the radius as well as entering the start and end angle.

The following presents some examples for drawing circles:

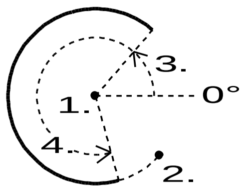
Arc using centre and point

This command can also be activated using this icon.

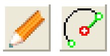
An arc is defined using the centre and a point on the circumference as well as the start and end angle.

Proceed as follows:

1. Define the centre.
2. Define a point on the circumference.
3. Determine the start angle of the arc.
   This can be determined by defining a point through which an imaginary line runs from the centre whose direction determines the angle.
   The value for the start angle can also be determined in the ***Angle*** text box. The angle entered always refers to the direction of the X-axis of the global coordinate system.
4. Determine the opening angle of the arc. This is anticlockwise - from the start angle; by holding the Shift key and for negative values for the opening angle clockwise.
   This can also be determined by defining a point or by entering a value in the ***Open.angle*** text box.



Arc using centre and diameter

 This command can also be activated using this icon.

An arc is defined using the centre and the diameter as well as the start and opening angle.

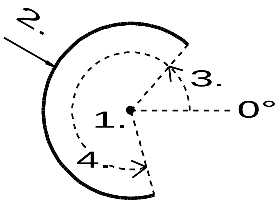
Proceed as follows:

1. Define the centre.

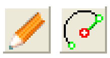
2. In the ***Diameter 1*** text box, enter the diameter of the circle.

3. Determine the start angle of the arc.
This can be determined by defining a point through which an imaginary line runs from the centre whose direction determines the angle.
The value for the start angle can also be determined in the ***Angle*** text box. The angle entered always refers to the direction of the X-axis of the global coordinate system.

4. Determine the opening angle of the arc. This is anticlockwise - from the start angle; by holding the Shift key and for negative values for the opening angle clockwise.
This can also be determined by defining a point or by entering a value in the ***Open.angle*** text box.



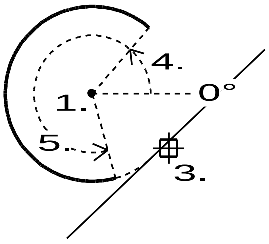
Arc using centre and tangent to a line

 This command can also be activated using this icon.

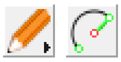
An arc is defined using the centre, the condition that it is to be tangent to a line (line, circle, arc) as well as the start and opening angle.

Proceed as follows:

1. Define the centre.
2. In the Snap Menu, select ***Tangent To***.
   By the cross-hair cursor, the additional symbol is displayed for 'tangent'.
3. Identify the line or circle (arc)near the required tangent point.
4. Determine the start angle of the arc.
   This can be determined by defining a point through which an imaginary line runs from the centre whose direction determines the angle.
   The value for the start angle can also be determined in the ***Angle*** text box. The angle entered always refers to the direction of the X-axis of the global coordinate system.
5. Determine the opening angle of the arc. This is anticlockwise - from the start angle; by holding the Shift key and for negative values for the opening angle clockwise.
   This can also be determined by defining a point or by entering a value in the ***Open.angle*** text box.



Arc using centre and arc length

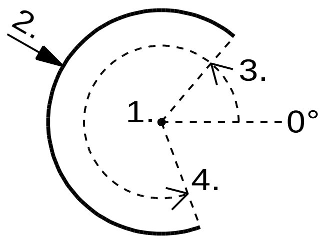
 After clicking the displayed icons the command ***Draw 2D***, ***Dynamic Arc*** is started.

To define an arc using its center point, diameter, starting angle and arc length, proceed as follows:

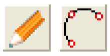
1. Define the center point.
2. In the field ***Diameter 1*** enter a value for the diameter of the circle.
3. Define the start angle of the arc.
   This can be done by defining a point through which an imaginary line runs from the center point, the direction of which determines the angle.
   You can also enter the value for the initial angle in the ***Angle*** field.

The angle entered in this field always refers to the direction of the X axis of the global coordinate system.

4. In the ***Arc length*** field, specify the value for the arc. This is drawn counterclockwise from the starting angle by holding down the ***Shift key*** and clockwise if the value is negative.



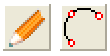
Arc using 3 points

 This command can also be activated using this icon.

Arcs are clearly determined by three points which do not lie on an infinite line.

The following presents some examples for drawing arcs:

Arc using points/tangent to lines

 This command can also be activated using this icon.

An arc is determined using three points. These can also be tangent points to existing lines, circles or arcs.

Therefore, this command enables arcs to be defined

- using three points
- using two points and tangent to a line
- using one point and tangential to two lines
- tangent to three lines

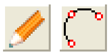
General procedure:

1. Define the endpoint of the arc.
2. Define another endpoint of the arc.
3. Define the final position and size by determining the third point of the arc.

Determine the points or tangent points on the lines, circles or arcs in the combination desired:

- A point can be defined dynamically, using the Snap Menu or by entering a value in the Status dialog box.
- If the circle is to run tangent to a line or circle (arc), load the Snap Menu and select ***Tangent To***.
  By the cross-hair cursor, the additional symbol is displayed for 'tangent'.
  Identify the line/circle (arc) near the required tangent point.
  If the position of the arc is not yet completely determined - whilst keeping the Shift key pressed - it can also be projected over the endpoints of a line/arc.

Arc using diameter, points/tangent to lines

 This command can also be activated using this icon.

An arc is defined using the diameter, two points and the position in reference to the two points. The points can also be tangent points to existing lines, circles or arcs.

Therefore, this command enables arcs to be defined

- using two points
- using a point and tangent to a line
- tangent to two lines

as well as by entering the diameter and determining the position.

General procedure:

1. Define the endpoint of the arc.

2. In the Status dialog box, enter the value for the ***Diameter 1***.
Then activate the drawing area again by pressing the spacebar.

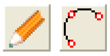
3. Position the cursor so that the arc has the required position.
Whilst keeping the Ctrl key pressed, the complementary arcs are drawn.
Confirm the position and length of the arc by pressing the left mouse button.

Determine the points or tangent points on the objects in the combination desired:

- A point can be defined dynamically, using the Snap Menu or by entering a value in the Status dialog box.
- If the circle is to run tangent to a line or circle (arc), load the Snap Menu and select ***Tangent To***.
  By the cross-hair cursor, the additional symbol is displayed for 'tangent'.

Identify the line/circle (arc) near the required tangent point.
If the position of the arc is not yet completely determined - whilst keeping the Shift key pressed - it can also be projected over the endpoints of a line/arc.

Tangent arc

 This command can also be activated using this icon.

A tangent arc is an arc which is tangent to the endpoint of a line or arc.

When drawing such a tangent arc, the following options are available:

Without diameter entry:

1. In the Snap Menu, select ***Tangent To***.

2. Identify the line/arc to which the new arc is to run tangent to.

3. Define the endpoint of the arc.

4. Position the cursor so that the arc touches the endpoint of the identified object and has the required position and size. Confirm by pressing the left mouse button.
Depending on the position of the cursor within the quadrant of the imaginary full circle, the program determines the "small" or "large" arc.

With diameter entry:

1. In the Snap Menu, select ***Tangent To***.

2. Identify the line/arc to which the new arc is to run tangent to.

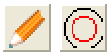
3. In the ***diameter 1*** text box of the Status dialog box, enter the radius of the arc.

4. If the length of the arc is also to be defined, enter the required value in the ***Length*** text box.
By entering a positive value, an arc is defined which runs anticlockwise, a negative value determines an arc which runs clockwise.

5. Position the cursor so that the arc touches the endpoint of the identified object and has the required position and size.
Depending on the position of the cursor within the quadrant of the imaginary full circle, the program determines the arc direction to the "right" or "left".
If the arc lies on the correct size, the "small" and "large" arc can be changed between by keeping the *Ctrl* key pressed and moving the cursor slightly.

Confirm the arc by pressing the left mouse button.
Please note that the cursor position must lie on the new arc. Otherwise the design is not possible.

Draw concentric circle/arc

 Alternatively, this command can be started using this icon.

After clicking on the displayed icons, the ***Draw 2D, Concentric Circle/Arc*** command is started.

A concentric circle/arc is defined from an existing circle/arc. The centre of the existing circle/arc is taken over, the radius is defined using a snap point or values entered in the Status dialog box.

Existing circle/arc

1. Identify an existing circle/arc.
2. The diameter of the existing circle/arc is displayed in the Status dialog box. Enter the diameter required for the concentric circle/arc and confirm the entry with the Enter key.
3. Enter another diameter in order to define another circle/arc concentric to the previous one or identify more circles/arcs. To these, concentric circles/arcs are created with the diameter previously defined in the Status dialog box.

As an alternative the command can be used in the following way:

Snap point

1. Define the required diameter fro it or the concentric circle by entering values in the Status dialog box.
2. Select the required snap point by using the Snap menu or by using keyboard commands.
3. Position the circle. If necessary, repeat steps 2 and 3 or more circles.

Coordinates

1. Define the required diameter for it or the concentric circle by entering values in the Status dialog box.
2. From the Snap Menu, select the option ***Cursor, Relative To Point*** or ***Relative To Last Point***.
3. Define the required position by entering values in the Status dialog box.

Draw point

Alternatively, this command can be started using this icon.

A point generated with one of these commands can be snapped as a design point using the ***Point*** command in the Snap Menu or by using the same option in the point filter.

The coordinates of a point can be defined dynamically using the cursor and the Snap Menu commands.

The coordinates can also be entered in the ***X*** and ***Y*** text boxes in the Status dialog box (***Pol.dist.***, ***Pol.angle*** for polar coordinates).

The size for displaying points on your monitor screen and for printing can be controlled by using the menu command ***Settings, Options*** and by selecting an option for ***Point marker***.

Draw ellipse

The commands defined in the following can be used to draw full ellipses as well as elliptical arcs.

Please note, that for full ellipses only the ***Centre*** and ***Quadrant points*** can be snapped as drawn points, for elliptical arcs ***Centre***, ***Quadrant point***, ***Endpoint*** and ***Midpoint*** .

Ellipse (elliptical arc) dynamic

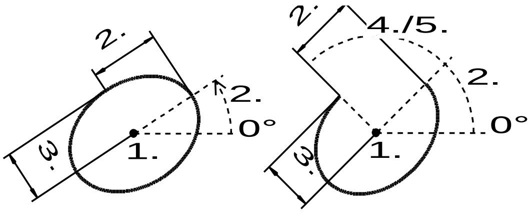
This command can also be activated using this icon.

An ellipse is defined using the centre, rotation angle as well as main and secondary radius.
For elliptical arcs an additional start and opening angle should be entered.

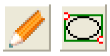
1. Define the centre.
2. Define the endpoint of the major axis in the same way or, in the ***Radius*** text box, enter half the length of the major axis and in the ***Angle*** text box, the rotation angle of the major axis.
3. Define the endpoint of the minor axis in the same way or, in the ***Radius*** text box, enter half the length of the minor axis.
   If a full ellipse is to be generated, press the right mouse button to quit the command.
4. Determine the start angle of the elliptical arc by defining a point. The line from the centre to this point, determines the angle.

The start angle can be defined in the same way, by entering the value in the ***Angle*** text box of the Status dialog box. This angle always refers to the direction of the X-axis of the global coordinate system.

5. Determine the opening angle by defining another point or by entering a value in the ***Open.angle*** text box of the Status dialog box. This angle refers to the direction of the ellipse major axis.
If the Shift key is pressed in dynamic mode and negative values for the opening angle, the ellipse arc is drawn clockwise.



Ellipse using rectangle

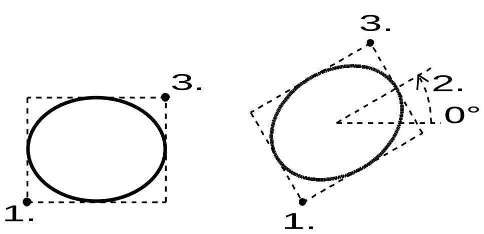
 This command can also be activated using this icon.

An ellipse will be drawn in a surrounding (imaginary) rectangle. First the corner points and, possibly, the rotation angle for the major angle of the ellipse will have to be specified.

1. Specify the first point of the surrounding rectangle.
Press the ***X*** key to create the rectangle starting from the centre point.

2. If the major axis of the ellipse is not to run parallel to the X-axis, enter the requires rotation angle in the ***Angle*** text box in the Status dialog box.

3. Specify the total length of the major and minor axes by specifying the second corner point of the rectangle.
These values can also be determined by entering the respective values in the ***Length X*** and ***Length Y*** text boxes in the Status dialog box.



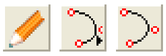
Draw spline

The following commands can be used to draw open and closed splines.

Please note, that with open splines, only the ***Endpoints*** and ***Midpoint*** can be snapped, with closed splines, no drawn points but only the ***Intersections*** with other lines.

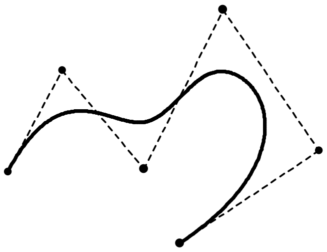
The control points of a spline can be moved using the command ***Transform, Show Geo Points***.

Open spline using control points

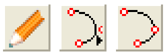
 This command can also be activated using this icon.

This command can be used to draw open splines which are determined between the corner points (control points) of a supporting point polyline.

As with the command ***Draw 2D, Polyline***, draw a polyline with at least two partial lines, whereby the endpoint of a partial line is also the start point of the following line.
The start and endpoints of the partial lines form the corner points between which the spline is generated as a rounded curve.

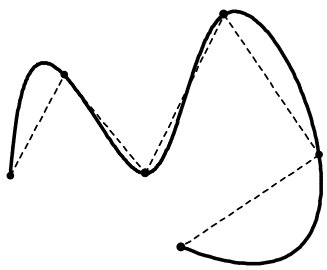


Open spline using interpolation

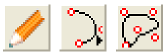
 This command can also be activated using this icon.

This command draws open splines which actually run through the corner points of a supporting point polyline.

As with the command ***Draw 2D, Polyline***, draw a polyline with at least two partial lines, whereby the endpoint of a partial line is also the start point of the following line.
The start and endpoints of the partial lines form the corner points through which the spline runs.

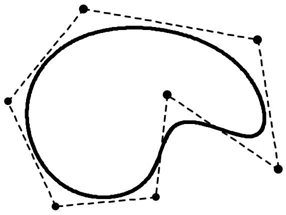


Closed spline using control points

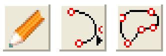
 This command can also be activated using this icon.

This command can be used to draw closed splines which are determined between the corner points (control points) of a supporting point polyline.

As with the command ***Draw 2D, Polyline***, draw a polyline with at least two partial lines, whereby the endpoint of a partial line is also the start point of the following line.
The start and endpoints of the partial lines form the corner points between which the spline is generated as a rounded closed curve.

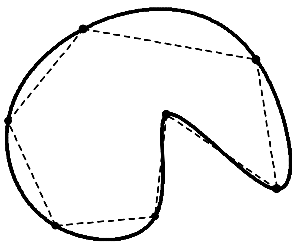


Closed spline using interpolation

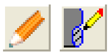
 This command can also be activated using this icon.

This command draws closed splines which actually run through the corner points of a supporting point polyline.

As with the command ***Draw 2D, Polyline***, draw a polyline with at least two partial lines, whereby the endpoint of a partial line is also the start point of the following line.
The start and endpoints of the partial lines form the corner points through which the closed spline runs.



Draw freehand line

 The command **Draw, Freehand Line** can be started using this icon.

The command draws an open spline which runs through the corners of a freehand line. This line is drawn according to the cursor movement. The exactness can be influenced by entering the distance between each two points.

1. Start the command and, in the ***distance*** text box, enter the maximum distance between each two points of the freehand line.

2. Using the cursor, position the staring point of the freehand line, enter the coordinates in the Status dialog box or use one of the Snap Menu commands.

3. Move the cursor appropriately to determine the run of the line.

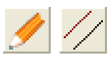
4. Define the endpoint of the freehand line on the current cursor position by confirming with the left mouse button.

If the endpoint of the freehand line is to be determined using one of the Snap Menu commands, press the **spacebar** and then select the required command from the Snap Menu or enter the coordinates for the point using the Status dialog box.

Please note:

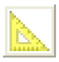
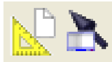
Apart from the starting and endpoints of the freehand line, no other points can be determined via numerical entry or a Snap Menu command.
Any required drawing grid is not taken into consideration.

Draw parallel to line

 This command can also be activated using this icon.

This command can be used to draw parallels to individual existing lines (lines, circles, arcs, ellipses, elliptical arcs and splines). This will apply to lines in another partial drawing .

The distance defined in the Status dialog box is constant and can be used for other parallel objects until a new value is defined. Parallels with variable distance to an object can be created using the ***Parallel Line/Contour*** command.

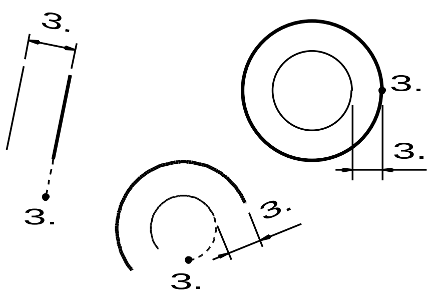
1. If you wish to create a parallel to an object on a different partial drawing to the one currently active, please take note that the respective partial drawing is active. If necessary, activate the partial drawing by selecting all  or using the list  to select the respective partial drawing.

2. Identify the line which is to have a parallel drawn to it.

3. Determine the position of the parallel line dynamically using the cursor or use one of the following options:

Determine the position by defining a point on the parallel line or enter the value for the distance to the original line in the ***Distance 1*** text box in the Status dialog box.
This is carried out on the side of the original line on which the cursor is positioned.

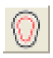
4. If, in the previous step, the value for the ***Distance 1*** has been entered, this value is retained.
Further lines can be identified, which are to be drawn parallel at the distance entered.
In this case, The parallels are drawn on the side of the original line on which the cursor was positioned during identification.



Please note:

The distance will be determined in accordance with the scale applicable to the current partial drawing.

Draw parallels to continuous line/contour

  This command can also be activated using these icons.

This command is for drawing lines that are parallel to individual objects and open or closed continuous lines that consist of 2D line objects in the current partial drawing (lines, circles, arcs, ellipses, elliptical arcs and splines).

This command can be used to define parallels to individual objects with any distances without having to reselect the object. The direction in which the parallels are created is determined via the sign preceding the entry in the Status dialog box.

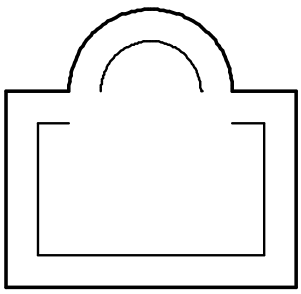
A continuous line must be just so, i.e. two adjoining line objects must share an endpoint.
In addition, a continuous line must not branch at any point, i.e. the endpoint of one line object must always be the endpoint of a second line object.

1. If the selection list is empty, take over the objects that are to be used from the action list.
It is important to remember that the only objects that can be dealt with are those in the current partial drawing.

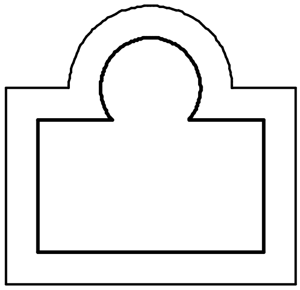
2. If you select more than one continuous line, choose the one that is to be dealt with by means of the marker at a point.

3. This step opens a context dialog box containing the options. Select the one for specifying in what form the "sharp" corners of the initial continuous line are to be drawn for the parallel continuous line.

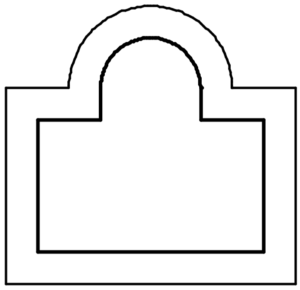
An example is shown in the illustration. If a new continuous line is to be generated as a parallel inside the original continuous line, you must use the options in the list to determine how the spaces are to be closed:



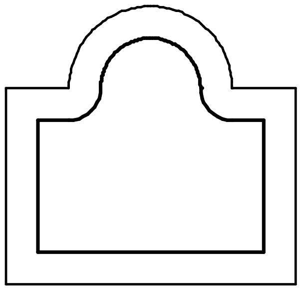
Extend lines
Curved line objects will be extended in accordance with their geometrical shape:



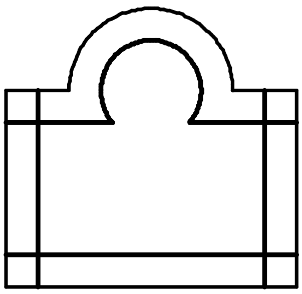
Insert lines
Curved line objects will be extended by tangential lines to close the gaps:



Insert arcs
In the case of external contours an arc will be inserted so that its midpoint forms the corner point between two adjacent line objects:

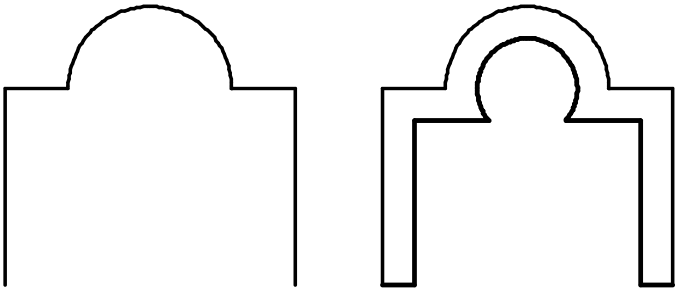


4. If necessary, deactivate the option ***Trim lines automatically*** so that trimming in the lines in the parallel continuous line will not be dealt with.



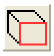
5. If the continuous line you have selected is open, you can activate the option ***Close contour***:
This means that lines will be inserted between the endpoint of the initial continuous line and those ion the parallel continuous line,

e.g.:



6. You can use the cursor to specify the position of the parallel continuous line dynamically or you can enter a value denoting the ***Distance 1*** in the Status dialog box.
   It is important to remember that the mathematical sign will always determine the side of the initial continuous line on which the new parallel continuous line will be generated.
7. Determine further parallels to the selected object/contour by defining more values using the Status dialog box.
8. The command can be quit by pressing the right mouse button or pressing the Esc key on the keyboard.

Project Edges (only 3D)

,  The command ***Draw 2D***, ***Project Edges*** is started using this icons.

This command projects a single edge or all the edges of a selected face onto the current workplane. The 2D objects are linked to the display parameters of the current layer or when deactivating the layer assignment to the object parameters.

1. In the 3D window select the command ***Activate WP*** or select the WP in the menu onto which the edges should be projected.
2. Identify a face or an edge of a solid.
3. Repeat the command or cancel it by pressing the right mouse button.

Intersection Line between Workplanes (only 3D)

,  The command ***Draw 2D***, ***Create WP Intersection Line*** is started using this icons.

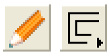
This command will create the intersection line between two workplanes in 3D space.

1. Identify the frame of the first workplane.

2. Identify the frame of the second workplane.

The length and position of the intersection line is defined by the size of the frame of the workplane.

Draw multi-line

 This command can also be activated using this icon.

The functions it contains allow you to draw polygon courses with up to four parallel sections.

You can define the settings for the number of parallels and their spacing using the menu command ***Settings***, ***Multi-line***.

Determine settings for multi-lines

The menu command ***Settings, Multi-line*** can be used to determine the three commands available for the determination of multi-lines.
Leave the dialog box open whilst carrying out the commands in order to be able to modify the settings at any time.

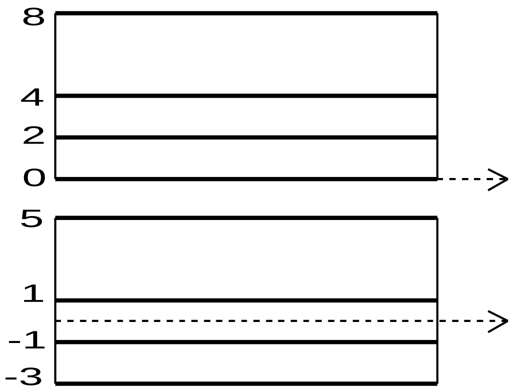
If the settings are to be permanent, save these settings in a template.

A multi-line is generated by defining a polyline:
This forms the axis line of the multi-line. Parallel to the partial lines of the axis line, up to four lines are drawn whose spacing can be determined.
The axis line itself is not drawn, but is only temporarily displayed during definition.

The following settings can be determined:

distance
If positive values are entered in the ***distance*** text boxes to the right of ***outer left***, ***inner left***, ***outer right***, ***inner right***, the parallels are generated to the left and right of the axis line (in drawing direction).
If one of the spacings has the value 0, the partial line generated with this setting will lie on the axis line.

If the spacings have negative values, the parallels are drawn on the other side of the axis line.



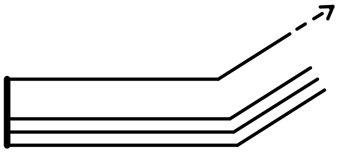
draw

A parallel to the axis line is only drawn if the respective ***draw*** check box is active.

This means that the check boxes can be used to define the number of parallels from which the multi-line should exist.

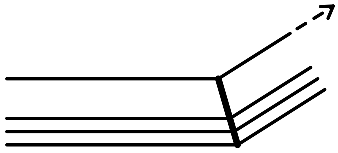
close start

If this check box is active, the start point of the outer parallels are linked to the first partial line by a connecting line.



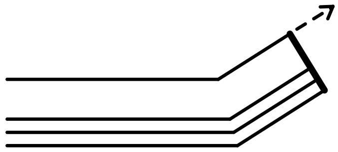
close inner

If this check box is active - apart from the last part (partial line) - the endpoints of the outer parallels are linked by a connecting line for each individual part.



close end

If this check box is active, the endpoints of the outer parallels of the last part of the multi-line are linked by a connecting line.



show at start of function

If this option is active, the settings dialog is opened at the start of the ***Draw multi-line*** function in order to be able to make a desired line setting.

Should the parameters be stored, define a ***Name*** for the parameter set und click the command button ***Store Parameter Settings***. Repeat this action to save multiple parameter sets for multi-lines. The paramter sets can be saved in a TPL file if used in other models also.

Please note:

To change a parameter set tag it in the list, change the parameters as needed and click the command button ***Update Parameter Settings***.

A parameter set can be erased using the command button ***Remove Current Parameter Set***.

The parameter sets for multi-lines can also be handled in an MPS file using the menu command ***File***, ***Read/Write Parameters***.

Draw multi-line

This command can also be activated using this icon.

This command can be used to draw multi-lines - parallel polylines - according to the currently defined settings.
Each line of such a multi-line can be processed individually.

1. Using the menu command ***Settings, Multi-Line***, determine the settings for multi-lines.
   The number of polylines as well as the spacing of the polylines to be drawn can be determined.
2. Activate the ***Draw 2D, Multi-Line, Multi-Line*** command and define a polyline. (as for the command ***Draw 2D, Polyline***). The endpoint of a partial line is also the start point of the following partial line.
   The polyline defined the run of the axis line for the multi-line. The polylines defined in the settings are drawn parallel to this axis line.
3. If the multi-line is to be closed, do not Cancel the command and select the command ***Draw 2D, Multi-Line, Close Multi-Line***.
   If the partial line which is to be generated last is to end on a line, do not Cancel the command and select the command ***Draw 2D, Multi-Line, Connect Multi-Line***.

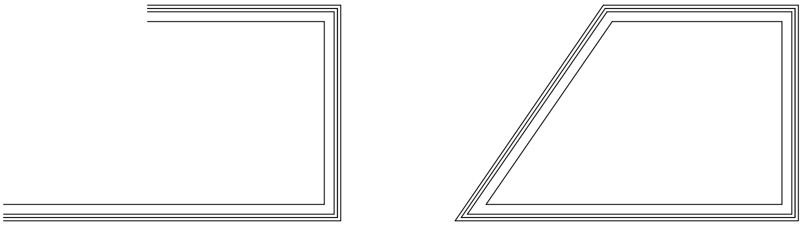
Close multi-line

 This command can also be activated using this icon.

This command can only be loaded if the command ***Draw 2D, Multi-Line, Multi-Line*** is active.

It joins the start point of the axis line to the endpoint by inserting a line. This is carried out in the same way with the parallels to the axis line.

The command ***Draw 2D, Multi-Line, Multi-Line*** can then be quit.



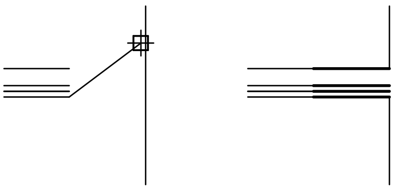
Connect multi-line

 This command can also be activated using this icon.

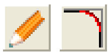
This command can only be loaded if the command ***Draw 2D, Multi-Line, Multi-Line*** is active.

It extends or shortens the partial line generated last so that it ends on an existing line or the imaginary extension.
Identify the line on which the partial line is to end.

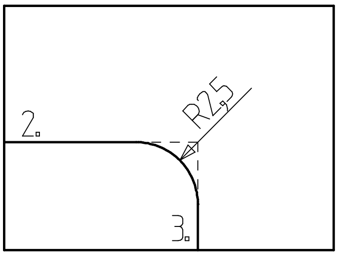
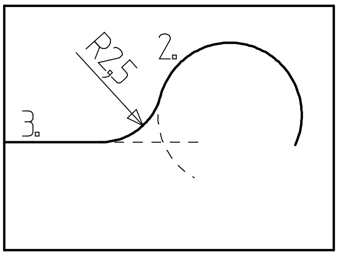
The command ***Draw 2D, Multi-Line, Multi-Line*** can then be quit.



Draw blend

 This command can also be activated using this icon.

This command can be used to connect the ends of two lines (lines, arcs) at a common intersection point and inserts an arc at this corner.

1. If a ***blend radius*** has been predefined on the ***2D draw*** option card using the menu command ***Settings, 3D/2D Commands***, this value is displayed in the *Radius* text box in the Status dialog box.

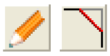
   If a different value is to be used, enter this value.

   By entering 0, it is possible to determine the radius dynamically or by entering numeric values.
2. Identify the first of the lines near the common point of intersection "within“ the corner which is to be blended.
3. Identify the second line in the same way.
4. If the value for the radius is larger than 0, the blend is drawn, if the design is possible.

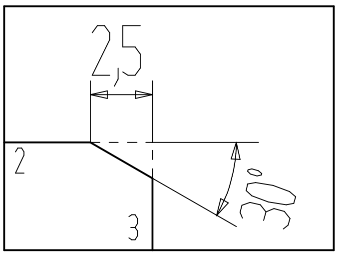
   If the value of the radius is 0, define it dynamically using the cursor or enter the required value in the ***Radius*** text box of the Status dialog box.
   If the blend is determined using the cursor, the cursor must be on the arc line when confirming with the left mouse button.

If the blend cannot be generated with the defined radius, a message is displayed.

Draw chamfer

 This command can also be activated using this icon.

This command connects the ends of two lines and inserts a chamfer at this corner.



1. If a ***chamfer length*** and ***chamfer angle*** have been predefined on the ***2D draw*** option card using the menu command ***Settings, 3D/2D Commands***, these values are displayed in the Status dialog

box.
If other values are to be used, enter these.

By entering 0 for the chamfer length, it is possible to determine both values dynamically or by entering numeric values.

2. Identify the first of the two lines where the intersection of the chamfer should be inserted.
The chamfer angle is started at the line.
3. Identify the second line.
4. If the value for the chamfer length is larger than 0, the chamfer is carried out.

   Otherwise, enter the value for the chamfer angle and then that for the chamfer length.

If the chamfer cannot be designed with the defined values, a message is displayed.

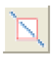
4.5 Process 2D objects

 This command can also be activated using this icon.

The commands described in the following can be used to trim, extend, interrupt, etc 2D line objects (lines, circles, arcs, ellipses, elliptical arcs and splines).

You can also disassemble the views of models and symbols, and edit or disassemble standard components. You can also work on displays showing sections.
A description of the last-named commands is to be found in the chapters describing how to use these objects.

Hide lines or partial lines

  Alternatively, the command can be started by using these symbols.

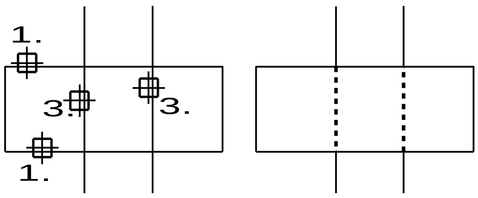
This command is for changing the display attributes for 2D line objects (line, full circle, arc, ellipse, elliptical arc, spline) or for parts of any of these.
What this means is that lines or partial lines can be changed to hidden lines by linking them to a layer displayed with the line type "broken".

If only parts of line objects are to be dealt with, the extent of the processing will be defined by the other 2D objects that intersect or touch the lines that are going to be dealt with.

The command can be used in one of the following ways:

Process with selection list:

1. You must accept the lines limiting the extent of the processing in the selection list, **before** you call up this command.
   The selection list can also include the lines (partial lines) that you are going to process.
2. Start the command and then define the colour, the line type, width or layer for the lines (partial lines) that are to be processed using the dialog that will appear on your screen.
   Click ***OK*** to accept the settings you have made.
3. Identify the lines (partial lines) that are to be allocated the properties.



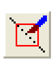
Process with action list

1. If there are no entries in your selection list, start the command and then specify the colour, line type, width or layer for the lines that you are going to process by completing the settings in the dialog.
   Click ***OK*** to accept the settings you have made.
2. Accept the lines defining the extent of the processing into your action list.
   The action list can also include the lines (partial lines) that you are going to process.
3. Identify the lines(partial lines) that are to be allocated the properties.

Please note:

When you use this command, you will generate new line objects. They will be saved both in the partial drawing and in the group actually containing the line object that was worked on.

Interrupt or trim on line(s)

  This command can also be activated using this icon.

This command can be used to cut off or cut out parts of 2D line objects (lines, circle, arc, ellipse, elliptical arc, spline).

The trimming boundaries are defined by other 2D line objects, which cut or are tangent to the 2D line objects to be processed.

In order to carry out this command, the trimming boundaries must be defined as well as the parts which are to be removed.

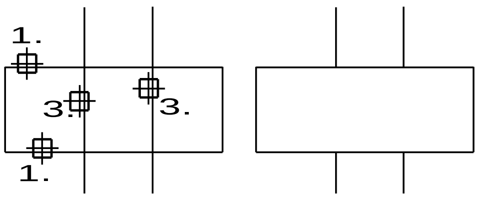
The following options are available:

Process with selection list:

1. **Before** loading the command, accept the lines which are to be used as "trimming tools" into the selection list.
   The selection list may also contain the lines to be processed.
2. Start the command and identify the parts of the lines which are to be cut off or cut out.
   All lines can be identified which have a "real" common point of intersection with at least one of the lines in the selection list.

Process with action list

1. If the selection list is empty, accept the lines which are to be used as "trimming tools" into the action list.
   The action list may also contain the lines to be processed.
2. Identify the parts of the lines which are to be cut off or cut out.
   All lines can be identified which have a "real" common point of intersection with at least one of the lines in the action list.



Trim line on line(s)

 This command can also be activated using this icon.

This command can be used to shorten or lengthen a 2D line object (line, arc, elliptical arc) dynamically to another line (line, circle, arc, ellipse, elliptical arc, spline).

Splines, full circles and full ellipses can be processed with this command.

In order to carry out this command, the trimming boundaries must be defined as well as the part-lines.

The following options are available:

Process with selection list

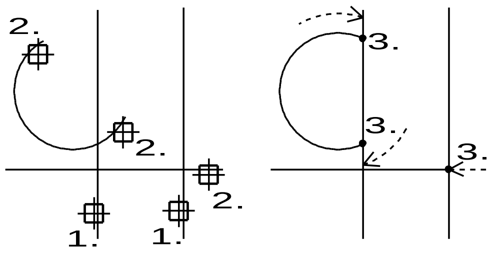
1. **Before** loading the command, accept the lines which are to be used as "trimming tools" into the selection list.
   The selection list may **not** contain the lines to be trimmed.
2. Start the command and identify one of the lines to be trimmed near the endpoint from which it is to be shortened or lengthened.
3. The position of the selected endpoint can be modified dynamically: Possible new positions are the (imaginary) intersections with lines from the selection list as well as both endpoints of the line to be trimmed.

   Move the cursor near the required new endpoint and confirm by pressing the left mouse button.
   If the command is aborted by pressing the right mouse button, the original state of the line is redisplayed.
4. Repeat steps 2 and 3 if further lines are to be trimmed on the lines contained in the selection list.

Process with action list

1. If the selection list is empty, accept the lines which are to be used as "trimming tools" into the action list.
   The action list may **not** contain the lines to be processed.
2. Identify one of the lines to be processed near the endpoint from which it is to be shortened or lengthened.
   Lines in the action list cannot be identified.
3. The position of the selected endpoint can be modified dynamically: Possible new positions are the (imaginary) intersections with lines from the selection list as well as both endpoints of the line to be trimmed.

   Move the cursor near the required new endpoint and confirm by pressing the left mouse button.
   If the command is aborted by pressing the right mouse button, the original state of the line is redisplayed.
4. Repeat steps 2 and 3 if further lines are to be trimmed on the lines contained in the selection list.

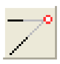


Please note:

If full circles and full ellipses are to be trimmed, use the command ***Process 2D, Divide On Line*** in order to convert them to arcs and elliptical arcs.

If the lines, arcs and elliptical arcs are to be shortened or lengthened in any way, therefore without intersecting objects, use the command ***Process 2D, Extend/Shorten***.

Join lines

,  This command can also be activated using this icon.

This command can be used to join two 2D line objects (line, arc, elliptical arc) with each other, i.e. they are shortened or lengthened so that one of the intersection points becomes the common endpoint.

The function can be used in the both modes *'connect endpoints'* and *'keep selected'*. By pressing the **X** key you switch to the other mode.

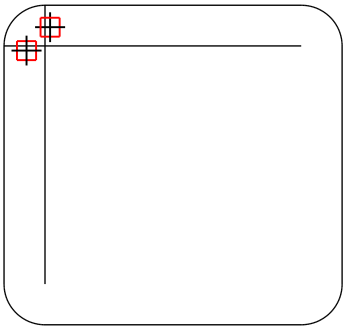
1. If the 2D drawing window is active, a dialog box will appear. You must first specify which partial drawings are to be available for selection.

2.a Mode **connect endpoints**
Identify both of the lines near the endpoint which is to be moved to the common point of intersection. The position of the respective other endpoint remains the same.

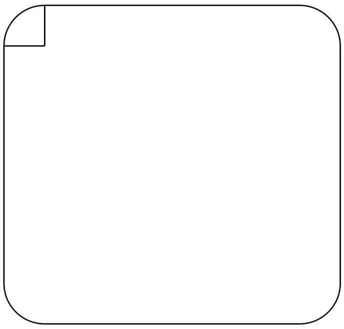
2.b Mode **keep selected**
Identify both of the lines. In this mode the identified line areas remain and are joined to an intersection point.

3. If both of the identified are actually 2D lines, only one common point of intersection exists. The identified endpoints are moved to this point of intersection.
If one of the identified lines is an arc or elliptical arc, they both have up to four common points of intersection. These are identified by markers.

Using the cursor, identify the arc near a marker in order to determine to which point of intersection the endpoints are to be moved.

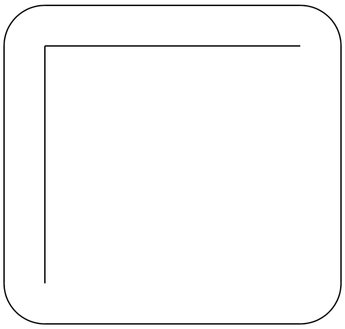


Keep Selected



The identified line areas remain.

Join end points



The identified line end points are joined to a common intersection point.

Please note:

This command can also be used with two lines, provided that they lie on one imaginary straight line.

If full circles and full ellipses are to be processed with this command, this is only possible if they are converted into arcs using the command ***Process 2D, Divide At Point*** or ***Divide On Line***.

If both of the divided objects are not to be shortened or lengthened at the same time, use the command ***Process 2D, Divide On Line***.

Extend / Shorten Lines

  The command ***Process 2D***, ***Extend / Shorten*** is started by clicking the displayed icons.

The command is used to extend or shorten lines and arcs dynamically or by a value.

1. Identify the line or the arc near the endpoint to be changed.

2. Extend or shorten the line or arc dynamically by mowing the cursor. To define the new endpoint also the snap menu can be used - except the commands ***Perpendicular to*** and ***Tangential to***.
   A perpendicular line is generated in the background from the defined point to the line or arc. The endpoint of this perpendicular line defines the new endpoint of the line or arc.

   Also a ***Length*** can be entered into the text box of the dialog box to extend or shorten the line or arc. Positive values means an extention and negative values a reduction.

   Alternatively in the field ***Tot. Length*** a value for the total length of the line object.

3. If a value is entered in the Length text box multiple lines can be extended or shortenend with this value. This lines or arcs must then be indentified near the endpoint to be changed.

Please note:

If full circles are to be processed with this command, this is only possible if they are converted into arcs using the command ***Process 2D, Divide At Point*** or ***Divide At Point***.

If single or a number of lines - even elliptical arcs - are to be shortened or extended so that they end on other lines, the command ***Process 2D, Trim On Line*** can also be used.

Extend / shorten lines to absolute value

  This command can also be activated using this icon.

After clicking on the displayed icons, the ***Process 2D, Extend/Shorten*** command is started.

This command extends or shortens lines and arcs to absolute values, if they are combined with the ***Pick menu*** in the Status dialog box.

1. If the 2D drawing window is active, in the dialog box that is displayed, determine which partial drawings are to be taken into consideration for the selection.

2. Define the absolute value on which the 2D line object is to be extended or shortened in the Status dialog box with the additional -tl.
   Example: You wish to shorten a line to the absolute value 200 mm, therefore, in the Status dialog box enter ***200-tl*** and confirm the entry with the Enter key.
   Using the abbreviation <tl> lengths of 2D line objects can be taken over from the drawing.

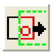
3. Identify the 2D line object to determine the current length of the object.
4. Identify the line object again at the position at which it is to be extended or shortened.

Please note:

If full circles are to be processed with this command, this is only possible if they are converted into arcs using the command ***Process 2D, Divide At Point*** or ***Divide At Point***.

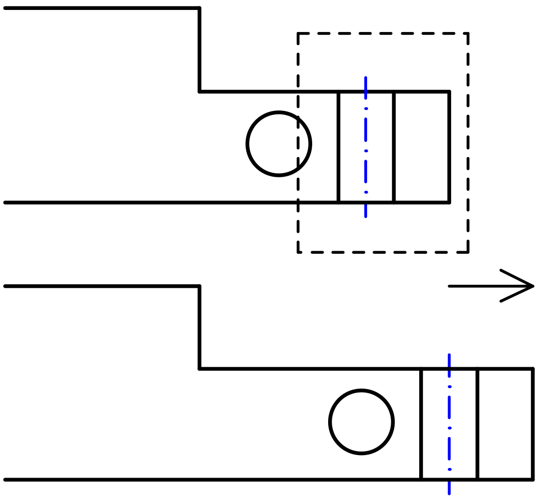
If individual or a number of lines - even elliptical arcs - are to be shortened or extended so that they end on other lines, the command ***Process 2D, Trim On Line*** can also be used.

Extend/trim lines and Contours

  Alternatively, the command can be started by these icons.

This command is for moving the endpoint of several lines along the identical vector.

If the vector runs parallel to the lines, the command can be used to extend and to trim lines, e.g.:



If there is only one endpoint of each line inside a window, this point will be moved, whilst the other will stay where it is.
Lines that are entirely inside the window will be moved along the vector.

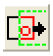
Any other objects will either be moved or stay in their original position. The parameters for selecting the object types are how they are to be dealt with are specified in ***Settings, Object Filter*** in the ***Clip selection mode*** block.

You can use either of the following procedures with this command:

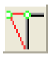
Action – Selection

1. If the 2D drawing window is active, you must first specify which partial drawings are to be available for selection.
2. Drag a window or collect a number of windows by keeping the Ctrl key pressed so that the window(s) contain endpoints of the lines to be extended or reduced.
3. Define the start of the displacement vector.
4. Define the endpoint. You can use the cursor or enter the data in the Status dialog box.

Selection – Action

1. Click the  to activate ***Reselect***.
2. If the 2D drawing window is active, you must first specify which partial drawings are to be available for selection.
3. Activate the selection mode ***Clip*** using the displayed icon  in the main toolbar.
4. Drag a window or collect a number of windows by keeping the Ctrl key pressed so that the window(s) contain endpoints of the lines to be extended or reduced.
5. Click ,  to activate the command ***Extend/Trim Lines***.
6. Define the start of the displacement vector.
7. Define the endpoint. You can use the cursor or enter the data in the Status dialog box.

Drag endpoint

,  This command can also be activated using this icon.

This command can be used to move the endpoint of one or a number of 2D line objects of the current partial drawing dynamically (line, arc, elliptical arc, open spline). The respective other endpoint remains the same.

1. Identify the line at the endpoint which is to be moved.
   If other lines project into the snapbox which also end in this point, their endpoints are also moved.
2. Determine the new position of the endpoint dynamically or use one of the options available for defining a point.

Please note, that the ***Perpendicular To*** and ***Tangent To*** commands in the Snap Menu cannot be used here.

Please note:

For full circles, full ellipses and closed splines, this command can only be used to dynamically move these objects.

The command ***Transform, Show Geo Points*** offers further options for modifying the position and size of individual lines.

Divide line at point

 This command can also be activated using this icon.

This command can be used to divide 2D line objects (line, circle, arc, ellipse, elliptical arc and spline) at a defined point.

1. If the 2D drawing window is active, a dialog box will appear. You must first specify which partial drawings are to be available for selection.

2. Identify the line which is to be divided.

3. Define the point at which the line is to be divided.
The actual division point is the point on the line which lies nearest to the defined point.

Please note:

The lines produced by division are embedded in the model just like the original line, i.e. they are saved in the same group, linked with the same layer and contain the same display attributes.

If a full circle or full ellipse is divided at a point, these objects become an arc or elliptical arc with an opening angle of 360 degrees. The division point can be snapped as the endpoint.
Such an arc can be processed with the command ***Process 2D, Divide On Line*** and ***Extend/Shorten***.

If a closed spline is divided at a point, the division point can be snapped as the endpoint.

Divide line on line

 This command can also be activated using this icon.

This command can be used to divide 2D line objects (line, circle, arc, ellipse, elliptical arc, spline) on other 2D line objects.

Before loading the command, the lines to be divided and the lines used as "trimming tools" must be defined.

Use one of the following options for carrying out the command:

Divide a number of lines on a line

1. **Before** loading the command, accept the lines which are to be divided into the selection list.
2. If the 2D drawing window is active, a dialog box will appear. You must first specify which partial drawings are to be available for selection.
3. Start the command and identify the dividing line, on which all of the lines in the selection list are to be divided.
   The dividing line itself can also be contained in the selection list.
   The lines in the selection list are divided at all (also imaginary) points of intersection which they have in common with the dividing line.

Divide one line on one or a number of lines

1. If the selection list is empty, identify the line which is to be divided. You can first specify which of the partial drawings are to be considered during the selection procedure.
2. If the line is only to be divided at one of the individual lines, identify this.
   If the line is to be divided by a number of dividing lines, accept the dividing lines in the action list.
   The line selected in step 1 is divided at all (also imaginary) points of intersection which are common with the lines in the action list.

Please note:

The lines produced by division are embedded in the model just like the original line, i.e. they are saved in the same group, linked with the same layer and contain the same display attributes.

Divide line n-times

 This command can also be activated using this icon.

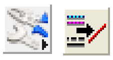
After clicking on the displayed icons, the ***Process 2D, N-Divide*** command is started.

This command can be used to divide 2D line objects (lines, circles, arcs, ellipses, elliptical arcs and splines) into a definable number of segments of the same length.

1. If the 2D drawing window is active, in the dialog box that is displayed, determine which partial drawings are to be taken into consideration for the selection.

2. Define the number of required divisions in the Status dialog box. Confirm the entry with the Enter key.
3. Identify the line(s) to be divided.
4. Quit the command using the right mouse button.

Transfer properties from 2D objects

 After selecting the displayed icons, the ***Transfer properties*** command is started.

This command can be used to transfer properties concerning colour, line type, line width, layer, order of 2D objects, text parameters and dimensions to other 2D objects.

1. Identify the starting object whose properties you wish to transfer to other 2D objects.
2. With a single click, or by dragging a window whilst keeping the mouse button pressed, identify the 2D objects which are to take over the new properties.
   All the objects which have been touched will take over the new properties.

Please note

The object type is not changed when taking over the properties. For example, should you take over the properties of a centre line and you allocate these to a *“normal“* line, the line takes over the object or layer properties of the centre line but not the object type.

This command can also be used to transfer the respective layer for the 3D objects.

Besides the text parameters also other parameters like colour, line type, line width, layer and drawing order are transferred to other objects – like lines, dimensions and hatches - if the start object is a text.

Generate symbol

 After clicking on the displayed icons, the ***Process 2D, Generate Symbol*** is started.

Use one of the following methods to save objects in a symbol.

By selecting the objects that are to be saved

1. Before loading the command , transfer the 2D objects that you want to save to the selection list. The only objects to be saved will

be those in the current partial drawing. If you do not define a selection list, you can move the objects into the action list later.

2. The dialog box ***Generate Symbol*** is displayed.
3. In the line ***in library*** you will see the name of the most current symbol library now suggested as the target library. If you want to use a different library, select one from the list or load an additional library by clicking .
   Note:
   The name of a library can directly be entered in the drop down list ***in library:.*** Confirm the input by pressing the Enter key. If the symbol library is found in the actual library path the library will be loaded. If not, a new library is generated.
4. Select the directory to be used to save the symbol by clicking it.
5. Type the name of the symbol in the ***symbol name*** box.
6. If you want to define two additional reference points for the symbol in addition to the insertion point, activate this setting. If you want to define two ***additional reference points*** for the symbol in addition to the insertion point, activate this setting.
   With symbols that are inserted you can use these points as Geo points, i.e. the symbols can be processed with the command ***Transform, Show Geo Points***.
7. Choose a creation mode:

   Save
   A copy of the 2D elements are stored in the current library as a symbol.

   Save and erase
   The geometry is erased from the drawing and stored as symbol in the current library.

   Save and replace
   The 2D elements are stored in the current library as a symbol. At the same time the 2D elements in the drawing are exchanged by the symbol.
8. Confirm the settings by clicking ***OK***.
9. If the selection list is empty, transfer the objects that are to be saved into the action list (e.g. by window selection or a group selection/SHIFT key).

10. Define the insertion point, i.e. the point with which the symbol is to be inserted in drawings later, using point definition or making an entry in the Status dialog box.
11. Depending on the setting made in Step 6, specify two further reference points if necessary.

    Afterwards the symbol will be saved in the symbol library. The symbol contains a group with the same name. After inserting the symbol and disassembling, all 2D objects of this symbol lie in this group.

By selecting the group that is to be saved

1. The dialog box *Generate Symbol* is displayed.
2. In the line *in library* you will see the name of the most recently selected symbol library now suggested as the target library. If you want to use a different library, select one from the list or load an additional library by clicking .

   Note:
   The name of a library can directly be entered in the drop down list ***in library:.*** Confirm the input by pressing the Enter key. If the symbol library is found in the actual library path the library will be loaded. If not, a new library is generated.
3. Select the directory to be used to save the symbol by clicking it.
4. Type the name of the symbol in the *symbol name* box.
5. If you want to define two *additional reference points* for the symbol in addition to the insertion point, activate this setting. With symbols that are inserted you can use these points as Geo points, i.e. the symbols can be processed with the command ***Transform, Show Geo Points***.
6. Activate the setting ***save defined group as symbol***.
7. Choose a creation mode:

   Save
   : A copy of the 2D elements are stored in the current library as a symbol.

   Save and erase
   : The geometry is erased from the drawing and stored as symbol in the current library.

   Save and replace
   : The 2D elements are stored in the current library as a symbol.

At the same time the 2D elements in the drawing are exchanged by the symbol.

8. Confirm the settings by clicking ***OK***.

9. The dialog box for selecting groups will appear.
Tag the group that is to be saved a s a symbol and the confirm this step by clicking ***OK***.

10. Define the insertion point, i.e. the point with which the symbol is to be inserted in drawings later, using point definition or making an entry in the Status dialog box.

11. Depending on the setting made in Step 6, specify two further reference points if necessary.

Afterwards the symbol will be saved in the symbol library. The symbol contains a group with the same name. This contains all saved 2D objects.

Please note:

When entering the symbol name, upper case, lower case, digits and umlauts are permitted also the special characters ~, $, Ø, -, <, >, ., (,)
Spaces are automatically replaced by underlines. However, please take note that: if you are exporting data into the DXF or DWG format, the symbol name must only contain the characters A-Z, 0-9, -, _.

Groups and objects as well as layers, are saved in a symbol with which the objects are linked, as well as the information as to whether the display attributes are to be object or layer specific.

If a symbol is scaled whilst being positioned, the geometry of all the objects it contains will be enlarged or reduced for the display. This applies in particular to standard parts and to dimensions; for this reason these will not have their correct measurements or dimension numbers. For this reason 2D standard parts should never be saved in symbols that are going to be scaled when positioned. Dimensions in scaled symbols will, on the other hand, be corrected automatically, if the symbols are disassembled.

Disassemble symbol

  This command can be used to disassemble symbols, i.e. disassemble them into their original 2D objects. The group structure contained in a symbol is moved to the group in which the symbol is included.

The symbol is disassembled one step at a time, i.e. if further symbols are contained in a symbol, these are not disassembled.

Use one of two options for selecting the required symbol to be disassembled:

With selection list If symbols already exist in the selection list, these are disassembled directly after loading the command.

Without selection list Using the ***Access to ... partial drawing***  command at the top border of the 2D window, define in which partial drawings the symbols to be disassembled are to be determined in. Accept the symbols to be disassembled in the action list.

Please note:

If a symbol has been allocated attribute values, these are removed when the symbol is disassembled. This is not valid for attribute values which the symbol contains.

If standard sheets have been allocated attributes, there should be **no** disassembly before the DWG/DXF export. Otherwise the attributes defined in the text field will not be taken over into the DWG/DXF drawing.

If you disassemble a symbol that has been scaled, the dimensions, finish characteristics, and shape and position tolerances as well as weld and seam symbols in it will be regenerated. The dimension numbers will be adapted to their true values and the objects will be given their original measurements.

 Symbols can also be disassembled in the first level using the command ***Process 2D, Disassemble 2D object***.

Completely disassemble symbols in model

  After clicking on the displayed icons, the ***Process 2D, Disassemble Symbol Completely*** command is started.

This command can be used to disassemble boxed symbols, i.e. disassemble them into their original 2D objects. The group structure contained in a symbol is subordinate to the group in which the symbol is embedded.

The symbol is disassembled with a number of steps, i.e. if further symbols are contained in a symbol, these are also disassembled.

Use one of two options for selecting the required symbols:

If symbols already exist in the selection list, these are disassembled directly after loading the command.

In the context dialog box which then appears, determine in which partial drawing the symbols to be disassembled are to be determined. Then identify the symbols to be disassembled.

Please note:

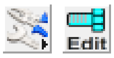
If a symbol has been allocated attribute values, these are removed when the symbol is disassembled. This is not valid for attribute values contained in the symbol.

If standard sheets have been allocated attributes, there should be **no** disassembly before the DWG/DXF export. Otherwise the attributes defined in the text field will not be taken over into the DWG/DXF drawing.

If you disassemble a symbol that has been scaled, the dimensions, finish characteristics, and form and positional tolerances as well as weld and seam symbols in it will be regenerated, i.e. the dimension numbers will be adapted to their true values and the objects will be given their original measurements.

 Symbols can also be disassembled in the first level using the command ***Process 2D, Disassemble 2D object***.

Edit 2D standard part

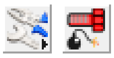
 After clicking on the displayed icons, the ***Process 2D, Edit 2D Standard Part*** command is started.

2D standard parts which have been inserted into drawings can be edited, i.e. their dimensions, view and position can be modified.

1. Identify the standard part to be processed.
2. The dialog box for the selection of 2D standard parts is then displayed. The standard part is displayed with the previously allocated dimensions.
3. If an individual part of a bolt is to be edited, click on this in the ***Selection tree*** window and, if necessary, select another individual part from the ***Selection*** window.
4. Under the ***Parameters*** heading, click on the line with the required data or, for *roller bearings*, the type name.
5. If necessary, change the view by selecting the new one from the list below the preview window.

6. Modify the position, by clicking on one of the options with the predefined angles or enter the required value in the ***angle*** text box.
7. Accept the modifications with ***OK*** or lose them with ***Cancel***. Then the old standard part will be erased and replaced by a new one in the same position.

Disassemble 2D standard parts

 After clicking on the displayed icons, the **Process 2D, Disassemble 2D Standard Part** command is started.

If 2D standard parts are to be processed, e.g. individual lines are to be erased, this is only possible if this standard part is disassembled.

Action - selection

1. If the selection list does not contain any 2D standard parts, load the command ***Process 2D Object, Disassemble 2D Standard Part***. Accept the 2D standard parts which are to be disassembled into the action list.

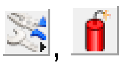
Selection - action

1. Accept the 2D standard parts which are to be disassembled into the selection list.
2. Load the command ***Process 2D Object, Disassemble 2D Standard Part***.

Please note: After disassembling, standard parts cannot be edited.

The exception: fastenings. After any of these has been disassembled once, you will still be bale to edit the separate components as standard parts.

After you have disassembled a fastening, a group will be created within the group structure and given the name 2DFea that will contain all the separate components used for the fastening. If a single standard part disassembled, a group with the name 2DFea will be created and it will contain all the objects in this standard part.

 2D standard parts can also be disassembled using the command ***Process 2D, Disassemble 2D object***.

Disassemble 2D Objects

,  By clicking the displayed icon the command ***Process 2D, Disassemble 2D Objects*** is started.

Using this command the following 2D objects can be disassembled:

- **Symbol**
  A group is created with the name of the symbol containing all the 2D objects or even symbols again, if the selected symbol contains other symbols. If the group structure was used to save the symbol, the original structure is restored.
- **2D standard part**
  2D standard parts are disassembled step by step. If, for example, a tthrough bolt is disassembled, the single components (srew, washer, nut etc.) are inserted in a new group named *2DfeaGruppe.* Single 3D standard parts are disassembled into their 2D objects and inserted in a new group named *2Dfea.*
- **Dimension**
  The single 2D objects are inserted in a new group named *Annotation.*
- **Text**
  A new group *TrueType-Kontur* is created which contains the 2D objects.
- **Referenced Text**
  A new group *Annotation* is created which contains the 2D objects.
- **Hatch**
  A new group *Hatch* is created which contains the lines.
- **Model views / Section views**
  The objects are inserted in the current partial drawing in the active group. Depending on the settings defined using the menu command ***Settings***, ***3D/2D Commands***, ***Model view*** the model is erased or not.
- **Details**
  Identify the frame, the ident text or the reference line of a detail view to disassemble the complete content (original and detail) including the contained view into 2D line objects. The ident text will remain as text. The elements of the detail will be inserted into the subgroup Annotation in the current partial drawing. The lines of the view will be created in the partial drawing / group in which the view has been before.

- **Finish Characteristics, Form/Position Tolerance, Welding Symbols**
  Groups are created with the name Gruppe_Oberfläche, Group_Fola, WeldSymbol where the 2D objects are inserted.
- **Section view annotation**
  A group is created with the name *Annotation,* where the 2D objects are inserted.

Identy a single object or multiple objects by a selection window or with the pressed ***Ctrl***-key. Unlock the ***Ctrl***-key before identifying the last element.

Please note:

Be sure to deactivate those objects not supposed to be disassembled when using the selection window, for example by excluding them in the menu command ***Settings***, ***Object Filter*** or be deactivating the relevant Layers, Groups or Partial Drawings.

4.6 Transform objects

 This command can also be activated using this icon.

Transformations are the commands for moving, for rotating, for scaling (=enlarging, reducing the size) and for mirroring objects.
When these commands are being used, there is always a choice of completing a transformation without or with a copy of the object.

Notes for copying:

In the case of **2D standard parts** or other objects forming part of a component only the geometry will be copied.
In addition the selection mode ***Group Selection*** can be activated in the main icon bar. In this mode and by pressing the ***Shift key*** it can be defined from a list wether to copy objects or groups.

If groups <u>and</u> the contained objects should be copied they can be identified in the ***Standard Selection*** by pressing the ***Alt key***. In the ***Group Selection*** mode press the ***Shift key*** to choose which structure level should be copied.

If you <u>only</u> want to select the objects contained in a group without copying the group, keep the *Shift* key pressed during the transformation with copy. If the ***Group Selection*** mode is active choose ***Elements of 2D group*** from the list.

Move object

Both of the commands described in the following can be used to move 2D objects or copies of these in the selected IXI 2D drawing or in the currently selected workplane of the 3D window.

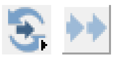
This command can also be used to move 3D objects. In this case, the definition of the move vector is always carried out in the current workplane.

A movement results in an actual modification of the object position, not just a simple move of the view.

Before hand, select objects so that you can carry out a dynamic move, then this command can also be used to move/copy objects from one context to another.
Move/copy between two ***partial drawings*** or two ***workplanes***, by clicking on the current partial drawing/workplane in the box and using the up/down arrow keys to select the required context.
Move/copy objects between 2D and 3D windows (***partial drawing*** and ***workplane)*** whilst they hang dynamically on the cursor and then pressing the Ctrl+TAB keys for example.

The respective, defined transformation vector is displayed in green during the action and then suppressed again once the command has been completed.

Move original

 This command can also be activated using this icon.

This command can be used to move 2D objects and 3D objects.

Selection - action (dynamic move)

1. **Before** loading the command, accept the objects to be moved in the selection list.
2. If solids are to be moved in the 3D-view window, select the workplane in which the move vector is to be defined.
3. Define the start point of the move vector (= reference point for the move).

   Please note:
   The start point of the moving vector must only be defined in the drawing / work plane if the vector is defined by the point definition menu.

4. If necessary, change to the partial drawing or workplane into which the objects are to be moved.
5. Define the target position e.g. by entering the values for ***Length*** and ***Angle*** in the status dialogue window, using absolute or relative coordinates or using the point definition menu.
   If the cursor is used to do so, the objects contained in the selection list are moved dynamically.

   If the command is aborted, the objects are redisplayed at their original position.

   Please note:
   This procedure is also for moving 2D objects from your current partial drawing into the current WP of the 3D-view window.
   Between ***partial drawings*** or ***workplanes***, by clicking on the current partial drawing/workplane in the box and using the up/down arrow keys to select the required context.
   Change between ***partial drawing*** and ***workplane***, by using the key combination Ctrl+TAB to scan between the opened windows.

Move with action - selection

1. If solids are to be moved in the 3D-view window, select the workplane in which the move vector is to be defined.
2. If the selection list is empty, define the start point of the move vector (= reference point for the move).

   Please note:
   > The start point of the moving vector must only be defined in the drawing / work plane if the vector is defined by the point definition menu.
3. Define the target position e.g. by entering the values for ***Length*** and ***Angle*** in the status dialogue window, using absolute or relative coordinates or using the point definition menu.
4. Accept the object to be moved in the action list.
   If objects are selected which have already been moved, these are moved again by the entered vector.

Please note:

The move vector is displayed in green once it has been defined and remains visible during the transformation procedure. You can change the values in the status dialogue window within the current function. By pressing ESC once or the right mouse button, the vector is erased, but the command remains active so that you can define a new vector straight away.

, , , When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.
The selection mode ***Clip*** is offered in a 3D window only in the top view.

Move copy

This command can also be activated using this icon.

This command is for copying 2D and 3D objects and moving them. You can also deal with groups and the objects in them. Copies of groups will always be saved in the group structure parallel to the original group.

Copy with selection - action (dynamic copy)

1. **Before** loading the command, accept the objects or groups to be copied in the selection list.
After you have selected the command, copies are generated of the objects contained in the selection list.

2. Load ***Transform, Move with copy***. After you have called up this command, the program will generate copies of the objects that are in the selection list.

3. If **solids** are to be copied and moved in the 3D-view window, select the workplane in which the move vector is to be defined.

4. Define the start point of the move vector (= reference point for the move).

Please note:
The start point of the moving vector must only be defined in the drawing / work plane if the vector is defined by the point definition menu.

5. If necessary, change to the partial drawing or workplane into which the objects are to be copied.

6. Define the target position e.g. by entering the values for ***Length*** and ***Angle*** in the status dialogue window, using absolute or relative coordinates or using the point definition menu.
If the cursor is used to do so, the copied objects contained in the selection list are moved dynamically.

The starting point of the move vector is retained in this command. One after the other, you can place any number of copies at any position until this command is quit.

If the command is aborted, the copies are erased.

Please note:
This procedure is also for copying 2D objects from your current partial drawing into the current WP of the 3D-view window.
Between ***partial drawings*** or ***workplanes***, by clicking on the current partial drawing/workplane in the box and using the up/down arrow keys to select the required context.
Change between ***partial drawing*** and ***workplane***, by using the key combination Ctrl+TAB to scan between the opened windows.

Copy with action - selection

1. If solids are to be copied and moved in the 3D-view window, select the workplane in which the move vector is to be defined.

2. If the selection list is empty, define the start point of the move vector (= reference point for the move).

Please note:
The start point of the moving vector must only be defined in the drawing / work plane if the vector is defined by the point definition menu.

3. Define the target position e.g. by entering the values for ***Length*** and ***Angle*** in the status dialogue window, using absolute or relative coordinates or using the point definition menu.

4. Accept the objects or groups to be copied, in the action list.
If objects are selected which have already been copied, these are moved again by the entered vector.

The copies accept the display attributes of the original object. They can be modified using the menu command ***Process, Object Display***.

The copies of the 2D objects are

- saved in the current group determined in group selection
- linked with the layer of the original object
- allocated the display attributes of the original object

The group allocation can later be modified in the ***Model Explorer***, the link to a layer with the menu command ***Process, Object Display***.

Please note:

The move vector is displayed in green once it has been defined and remains visible during the transformation procedure. You can change the values in the status dialogue window within the current function. By pressing ESC once or the right mouse button, the vector is erased, but the command remains active so that you can define a new vector straight away.

, , , When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.
The selection mode ***Clip*** is offered in a 3D window only in the top view.

In the 3D-view window you cannot use this option except in the top view of the current workplane.

You can use the ***Settings / Object Filter*** menu to define the ***copy mode for objects***. Depending on the setting made here, only objects are copied or group structures are also copied.

Move or copy 2D objects in another partial drawing

Alternatively, the commands can also be loaded using these icons.

The commands for moving originals and copies can also be used to move or copy 2D objects or 2D groups from a partial drawing into another.

Proceed as follows:

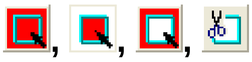
1. **Before** loading the command, accept the objects to be moved or copied in the selection list.
   If you select a group, you can copy both the group and the objects in it.
2. Define the start point of the move vector (= reference point for the move).
3. Activate the target partial drawing using the selection list in the 2D toolbar.
4. Determine the target position of the reference point.
   If the cursor is used to do so, the objects contained in the selection list are moved dynamically.
5. If the target partial drawing is aborted by pressing the left mouse button, the 2D objects are positioned where they were initially in the partial drawing (s).

The 2D objects are

- saved in the current group of the active partial drawing
- linked with the layer of the original object

- allocated the display attributes of the original object

Please note:

 When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.
You can use the ***Settings, Object Filter*** menu to define the ***copy mode for geometry objects***. Depending on the setting made here, only objects are copied or group structures are also copied.

Move or copy 2D objects in a workplane

 Alternatively, the commands can also be loaded using these icons.

The commands for moving originals and copies can also be used to move or copy 2D objects or 2D groups from the selected 2D drawing in a workplane of the 3D window.
You can also use them to move or to copy 2D objects from one workplane to another.

Proceed as follows:

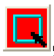
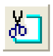
1. **Before** loading the command, accept the 2D objects to be moved or copied in the selection list.
   If you have selected a group both it and the objects in it will be copied or moved.
2. Define the start point of the move vector (= reference point for the move).
3. Activate the 3D-view window e.g. by using the two keys ***Ctrl Tab.***
4. 

   Using the list box or icon displayed here from the 3D toolbar, select the workplane in which the 2D objects are to be taken over.
5. Determine the target position of the reference point.
   If the cursor is used to do so, the objects contained in the selection list are moved dynamically.
6. If the target workplane is aborted with the right mouse button, the 2D objects are positioned where they were initially in the 2D window.

The 2D objects are

- linked with the layer of the original object
- allocated the display attributes of the original object

Please note:.

, , ,  When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.
In the 3D-view window user defined linetypes always will be displayed as a continuous line.

You can use the ***Settings / Object Filter*** menu to define the ***copy mode for objects***. Depending on the setting made here, only objects are copied or group structures are also copied.

Move/copy objects dynamically

Both of the following commands can be used to move 2D and 3D objects or their copies dynamically in the currently selected 2D drawing or currently selected workplane in the 3D window – attached to the cursor.

This command can also be used to move/copy objects between different workplanes and/or partial drawings.

In relation to movement, there are further additional transformation options available.

  **or**  After clicking on the displayed icons, the ***Transform, Dynamic Move*** or ***Dynamic Copy*** command is started.

This command is for moving or copying 2D and 3D objects dynamically within the current partial drawing or 3D workplane.

1. Accept the objects to be moved/copied in the action list after loading this command.

2. If solids are to be copied and moved in the 3D-view window, select the workplane in which the move vector is to be defined.

3. If necessary, define the starting point of the move vector (= reference point for the move) using the Snap Menu.

4. If need be, change the context whilst the object hangs on the cursor.
Between ***partial drawings*** or ***workplanes***, by clicking on the current partial drawing/workplane in the box and using the up/down arrow keys to select the required context.
Change between ***partial drawing*** and ***workplane***, by using the key combination Ctrl+TAB to scan between the opened windows.

5. If necessary, use one of the following additional options before you position the objects to be transformed in the target position.

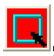
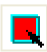
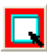
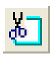
Keyboard entry

X	-	The objects found on the cursor are mirrored on the X-axis.
Y	-	The objects found on the cursor are mirrored on the Y-axis.
Left arrow key ←	-	The objects found on the cursor are rotated in an anticlockwise direction by the angle defined in the Status dialog box.
Ctrl + left arrow key ←	-	The objects found on the cursor are rotated in an anticlockwise direction by 90°.
Right arrow key →	-	The objects found on the cursor are rotated in a clockwise direction by the angle defined in the Status dialog box.
Ctrl + right arrow key →	-	The objects found on the cursor are rotated in a clockwise direction by 90°.
Up arrow key↑	-	The objects found on the cursor are scaled larger by the factor defined in the Status dialog box.
Down arrow key ↓	-	The objects found on the cursor are scaled smaller by the factor defined in the Status dialog box.
Ctrl+R	-	The objects found on the cursor are rotated dynamically about the cursor point.
Ctrl+S	-	The objects found on the cursor are scaled dynamically.

6. Determine the target position of the reference point in the same way (if necessary, using the Snap Menu).

7. For the case of dynamic copying, a further copy is kept at the end, on the cursor at the position of the first copy and this can then be transformed afresh and positioned.

Please note:

, , ,  When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.
The selection mode ***Clip*** is offered in a 3D window only in the top view.

You can use the ***Settings / Object Filter*** menu to define the ***copy mode for objects***. Depending on the setting made here, only objects are copied or group structures are also copied.

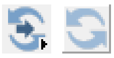
Rotate object

Both of the commands described in the following can be used to rotate 2D objects or copies of these in the selected 2D drawing about a point in the active partial drawing of the 2D drawing area or in the currently selected workplane of the 3D-view window.

This command can also be used to rotate 3D objects. In this case, the definition of the midpoint and rotation point always take place in the active partial drawing or in the current workplane.

A rotation results in an actual modification of the object position, not just a simple rotation of the view.

Rotate original

 This command can also be activated using this icon.

This command can be used to rotate 2D and 3D objects.

Rotate with selection - action (dynamic rotate)

1. **Before** loading the command, accept the objects to be rotated in the selection list.
2. If **solids** are to be rotated in the 3D-view window, if necessary, select the workplane in which the midpoint and rotation angle should be defined.
3. Define the midpoint about which is to be rotated.
4. Determine a point which is to be used as the reference point for the rotation.
   The connection between the midpoint and this point defines the origin from which the open angle is detracted.
5. Determine the rotation angle.
   This can be entered in the text box of the Status dialog box.

   Angle= The value entered in this field is relative to the x axis of the global coordinate system.

   Open Angle = The value entered in this field is relative to the start point of the rotation vector.

   It can also be determined by defining the new angle position with the end point of the rotation vector.
   If the cursor is used, the objects are rotated dynamically.

If the command is aborted, the objects are drawn at their original positions.

Rotate with action - selection

1. If **solids** are to be rotated in the 3D-view window, if necessary, select the workplane in which the midpoint and rotation angle should be defined.
2. If the selection list is empty, define the midpoint about which the rotation is to take place.
3. In the same way, determine a point which is to act as the reference point for the rotation.
   The connection between the midpoint and this point defines the origin from which the open angle is detracted.

Please note:

This step can be skipped, if the open angle is entered in the status dialog window.

4. Determine the rotation angle.
   This can be entered in the Status dialog box.

   Angle= The value entered in this field is relative to the x axis of the global coordinate system.

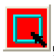
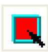
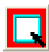
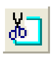
   Open Angle = The value entered in this field is relative to the start point of the rotation vector.

   It can also be determined by defining the new angle position with the end point of the rotation vector.

5. Transfer the objects to be rotated into the action list.
   If objects are selected which have already been rotated, these are rotated again by the entered midpoint and angle.

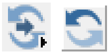
Please note:

The rotate vector is displayed in green once it has been defined and remains visible during the transformation procedure. You can change the values in the status dialogue window within the current function. By pressing ESC once or the right mouse button, the vector is erased, but the command remains active so that you can define a new vector straight away.

, , ,  When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.
The selection mode ***Clip*** is offered in a 3D window only in the top view.

In the 3D-view window you cannot use this option except in the top view of the current workplane.

Rotate copy

 This command can also be activated using this icon.

This command is for copying and rotating 2D objects and 3D objects. In a similar way you can copy groups and the objects in them. Copies of groups will always be saved in the group structure parallel to the original group.

Rotate with selection - action (dynamic rotate)

1. **Before** loading the command, accept the objects or groups to be copied in the selection list.
   After you have loaded the objects, the copies are generated of the objects contained in the selection list.
2. Load ***Transform, Rotate with copy***. After you have called up this command, the program will generate copies of the objects that are in the selection list.
3. If **solids** are to be copied and rotated in the 3D-view window, if necessary, select the workplane in which the midpoint and rotation angle should be defined.
4. Define the midpoint about which is to be rotated.
5. In the same way, determine a point which is to act as the reference point for the rotation.
   The connection between the midpoint and this point defines the origin from which the open angle is detracted.

Please note:

This step can be skipped, if the open angle is entered in the status dialog window.

6. Determine the angle about which the copies are to be rotated.
   This can be entered the Status dialog box.

   Angle= The value entered in this field is relative to the x axis of the global coordinate system.

   Open Angle = The value entered in this field is relative to the start point of the rotation vector.

   It can also be determined by defining the new angle position with the end point of the rotation vector.
   If the cursor is used, the objects are rotated dynamically.

The starting point of the rotation vector is retained in this command. One after the other, you can place any number of copies at any position until this command is quit.

If the command is aborted the copies are erased.

Rotate with action - selection

1. If **solids** are to be copied and rotated in the 3D-view window, if necessary, select the workplane in which the midpoint and rotation angle should be defined.

2. If the selection list is empty, define the midpoint about which the copies are to be rotated.

3. In the same way, determine a point which is to act as the reference point for the rotation.
The connection between the midpoint and this point defines the origin from which the open angle is detracted.

Please note:

This step can be skipped, if the open angle is entered in the status dialog window.

4. Determine the angle about which the copies are to be rotated.
This can be entered the Status dialog box.

Angle= The value entered in this field is relative to the x axis of the global coordinate system.

Open Angle = The value entered in this field is relative to the start point of the rotation vector.

It can also be determined by defining the new angle position with the end point of the rotation vector.

5. Transfer the objects that are to be rotated or the group into the action list.
If objects are selected which have already been rotated, these are copied again and rotated by the entered midpoint and angle

The copies accept the display attributed of the original object. They can be modified using the menu command ***Process, Object Display***.

The copies of the 2D objects are

- saved in the current group determined in group selection
- linked with the layer of the original object

Please note:

The rotate vector is displayed in green once it has been defined and remains visible during the transformation procedure. You can change the values in the status dialogue window within the current function. By pressing ESC once or the right mouse button, the vector is erased, but the command remains active so that you can define a new vector straight away.

You can use the ***Settings / Object Filter*** menu to define the ***copy mode for objects***. Depending on the setting made here, only objects are copied or group structures are also copied.

, , , When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.
The selection mode ***Clip*** is offered in a 3D window only in the top view.

In the 3D-view window you cannot use this option except in the top view of the current workplane.

Scale objects

Both of the commands described in the following can be used to move 2D objects or copies of these in the selected 2D drawing or in the currently selected workplane of the 3D window, i.e. enlarge or reduce.

Scaling results in an actual size modification of the object position, not just a simple move of the view.

Please note:

These commands should not really be used on solids.
Since the management of solid data is carried out with an exactness of 6 decimal places, solid scaling by necessary rounding up of calculation results could lead to increased imprecision. This could result in inconsistencies in your model. This is particularly valid for enlargements of solids.

Scale original

This command can also be activated using this icon.

This command is for enlarging or reducing the size of 2D objects. When you are in the 2D window you can use an option to specify different factors for scaling along the X and Y directions.

Scale with selection - action (dynamic scale)

1. **Before** loading the command, accept the objects to be copied in the selection list.
2. Define the fixpoint of the scale.
3. Determine the reference for the scale.
   The connection from the fixpoint to this point defines a ray.
4. The 3D window is for specifying the individual scaling factor and the 2D drawing window for specifying the two factors.
   The fields to use for your entries in the Status dialog box are those for ***Factor XY*** (3D) or ***Factor X*** and ***Factor Y*** (2D).
   In addition you can specify another point to determine the length and orientation of the line for determining the scaling factors.
   If the cursor is used, the copied objects are scaled dynamically.

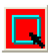
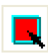
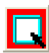
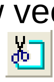
If the command is aborted, the objects are redrawn in their original size and position.

Scale with action - selection

1. If the selection list is empty, define the fixpoint of the scale.
2. In the same way, determine a point which is to act as the reference point for the scale.
   The connection between the fixpoint and this point defines a ray.
3. Specify one scaling factor or both.
   The fields to use for your entries in the Status dialog box are those for ***Factor XY*** (3D) or ***Factor X*** and ***Factor Y*** (2D).
   In addition you can specify another point to determine the length and orientation of the line for determining the scaling factors.
4. Transfer the objects that are to be scaled into the action list.
   If objects are selected which have already been scaled, these are scaled again from the same fixpoint about the same factor.

Please note:

The scaling vector is displayed in green once it has been defined and remains visible during the transformation procedure. You can change the values in the status dialogue window within the current function. By pressing ESC once or the right mouse button, the vector is erased, but the command remains active so that you can define a new vector straight away.

**,** **,** **,**  When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.

The selection mode ***Clip*** is offered in a 3D window only in the top view.

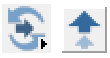
In the 3D-view window you cannot use this option except in the top view of the current workplane.

Neither full circles nor arcs will be distorted: the will be scaled using the larger of the two scaling factors.

The shape and position tolerances as well as the finish characteristics will not be scaled. The position will be adapted provided that they have been positioned with reference to a line object.

This command does not operate with standard parts.

Scale copy

 This command can also be activated using this icon.

This command is for copying and enlarging or reducing the size of 2D objects.
In a similar way you can copy groups with the objects in them. The copies of the groups will always be saved in the group structure parallel to the original group.

When you are in the 2D window you can use an option to specify different factors for scaling along the X and Y directions.

Scale with selection - action (dynamic scale)

1. **Before** loading the command, accept the objects or groups to be copied in the selection list.
After you have loaded the command, copies are generated of the objects contained in the selection list.
2. Load the command ***Transform, Scale with Copy***. After you have loaded the command the program will generate the copies of the objects that are in the selection list.
3. Define the fixpoint of the scale.
4. Determine the reference for the scale.
The connection from the fixpoint to this point defines a ray.
5. The 3D window is for specifying the individual scaling factor and the 2D drawing window for specifying the two factors.
The fields to use for your entries in the Status dialog box are those for ***Factor XY*** (3D) or ***Factor X*** and ***Factor Y*** (2D).
In addition you can specify another point to determine the length

and orientation of the line for determining the scaling factors.
If the cursor is used, the copied objects are scaled dynamically.

If the command is aborted, the copies are erased again.

Scale with action - selection

1. If the selection list is empty, define the fixpoint of the scale.
2. Determine a point which is to act as the reference point for the scale.
   The connection between the fixpoint and this point defines a ray.
3. Specify one scaling factor or both.
   The fields to use for your entries in the Status dialog box are those for ***Factor XY*** (3D) or ***Factor X*** and ***Factor Y*** (2D).
   In addition you can specify another point to determine the length and orientation of the line for determining the scaling factors.
4. Transfer the objects that are to be scaled or the groups into the action list.
   If objects are selected which have already been copied, these are copied again and scaled from the same fixpoint about the same factor

The copies of the objects are

- saved in the current group determined in group selection
- linked with the layer of the original object
- allocated the display attributes of the original object

Please note:

The scaling vector is displayed in green once it has been defined and remains visible during the transformation procedure. You can change the values in the status dialogue window within the current function. By pressing ESC once or the right mouse button, the vector is erased, but the command remains active so that you can define a new vector straight away.

You can use the ***Settings / Object Filter*** menu to define the ***copy mode for objects***. Depending on the setting made here, only objects are copied or group structures are also copied.

, , , When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.
The selection mode ***Clip*** is offered in a 3D window only in the top view.

In the 3D-view window you cannot use this option except in the top view of the current workplane.

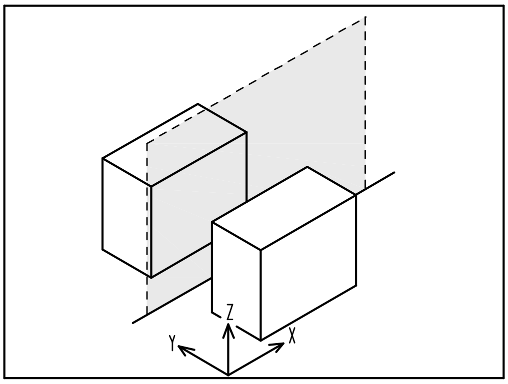
Neither full circles nor arcs will be distorted: the will be scaled using the larger of the two scaling factors.

This command does not operate with standard parts.

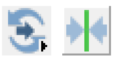
Mirror objects

Both of the commands described in the following can be used to mirror move 2D objects or copies of these in the selected 2D drawing on a freely definable axis or in the currently selected workplane of the 3D window.

These commands can also be used to mirror 3D objects on a plane which runs vertical to the current workplane. Their position is determined by the definition of an axis:



Mirror original

 This command can also be activated using this icon.

This command can be used to mirror objects.

This command can be used to mirror 2D objects and 3D objects.

Mirror with selection - action (dynamic mirror)

1. **Before** loading the command, accept the objects to be mirrored in the selection list.
2. Define the first point on the mirror axis.
3. Determine the position of the mirror axis completely:
   The required value can be entered in the ***Angle*** text box in the Status dialog box.
   This can also be done by defining another point.
   When mirroring solids, define a mirror plane in this way whose axis

is defined by the mirror axis and is aligned vertically to the current workplane.

If the cursor is used, the objects are mirrored dynamically.

If the command is aborted, the objects are redrawn in their original position.

Mirror with action - selection

1. If the selection list is empty, define the first point on the mirror axis.
2. Determine the position of the mirror axis completely:
   The required value can be entered in the *Angle* text box in the Status dialog box.
   When mirroring solids, define a mirror plane in this way whose axis is defined by the mirror axis and is aligned vertically to the current workplane.

   The run of the mirror axis or mirror plane can be determined by defining another point.
3. Transfer the objects that are to be mirrored into the action list.

Please note:

The mirror axis is displayed in green once it has been defined and remains visible during the transformation procedure. By pressing ESC once or the right mouse button, the axis is erased, but the command remains active so that you can define a new mirror axis straight away.
, , , When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.
In the 3D-view window you cannot use this option except in the top view of the current workplane.

Mirror copy

This command can also be activated using this icon.

This command can be used to generate and mirror copies of objects. You can also copy groups with the objects in them. Copies of groups will always be saved in the group structure parallel to the original group.

Mirror with selection - action (dynamic mirror)

1. **Before** loading the command, accept the objects or groups to be copied in the selection list.
   After you have loaded the command, copies are generated of the objects contained in the selection list.

2. Load the command ***Transform, Mirror with Copy***. After you have loaded the command, the copies of the objects in the selection list will be generated.
3. Define the first point on the mirror axis.
4. Determine the position of the mirror axis completely:
   The required value can be entered in the *Angle* text box in the Status dialog box.
   This can also be done by defining another point. When mirroring solids, define a mirror plane in this way whose axis is defined by the mirror axis and is aligned vertically to the current workplane.

   If the cursor is used, the copied objects are mirrored dynamically.

If the command is aborted, the copies are erased.

Please note:

Since the copies are generated at the position of the origin when the command is loaded, the points on the mirror axis can be defined on these copies.

Mirror with action - selection

1. If the selection list is empty, define the first point on the mirror axis.
2. Determine the position of the mirror axis completely:
   The required value can be entered in the *Angle* text box in the Status dialog box.
   This can also be done by defining another point.
3. Transfer the objects that are to be mirrored into the action list.

The copies accept the display attributed of the original object. They can be modified using the menu command ***Process, Object Display***.

The copies of the 2D objects are

- saved in the current group determined in group selection
- linked with the layer of the original object

Please note:

The mirror axis is displayed in green once it has been defined and remains visible during the transformation procedure. By pressing ESC once or the right mouse button, the axis is erased, but the command remains active so that you can define a new mirror axis straight away.

You can use the ***Settings, Object Filter*** menu to define the ***copy mode for objects***. Depending on the setting made here, only objects are copied or group structures are also copied.

, , , When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.
In the 3D-view window you cannot use this option except in the top view of the current workplane.

If texts should be mirrored, their behaviour can be defined using the menu ***Settings, Text***.

Multiply objects

After you have created 2D- or 3D-objects you may want to multiply them and arrange the copies evenly. Use the following commands in the 2D toolbox: ***Transform, Multiply*** with ***sort in longitudinal, matrix, wreath*** or ***circular spacing***.

The required parameters are defined in the current drawing or in the current WP of the 3D view window.

The number of copies for the ***length X***, the ***length Y*** and the ***angle***, which was last entered in the status window, are saved with the general settings when the program is terminated and are thus available again when the command is restarted.

If a group is selected as object to be copied, new groups including their complete content are created on the same structure level.

If you multiply solids in the 3D window, note the additional options for solids.

Multiply objects with sort in longitudinal spacing

, Alternatively, the command can be started with these icons.

This command is for multiplying objects and groups.
The copies will be arranged along a line (imaginary line) at the same distance from each other; the starting point of the line is the original object.

If you are multiplying solids in the 3D window, note the additional options for Fixed Solids

1. Select the type of arrangement for the copies in the dialog box.

Depending on the option you select, use one of the following procedures:

Length and number

Defines the arrangement along the entire length of the line of the copies, and specifies the number of copies.

2. Define the starting point of the (imaginary) line on which the objects are to be placed.
3. Enter the number of copies in the ***Number*** field of the Status dialog box.
4. Define the endpoint of the line and thus the ***Angle*** and ***Length*** (=total length) of the line: use either the cursor or enter the value in the Status dialog box.
   The value resulting from the total length and the quantity is used to determine the ***Distance*** between each pair of copies is displayed only for your information. You cannot enter a value yourself.
5. Select the objects that are to be multiplied.

Distance and number

Defines the way the copies are arranged by means of a specific distance between each pair of copies and the quantity of copies.

2. Enter the quantity of copies as a number against ***Number*** in the Status dialog box.
3. Define the starting point for the vector to specify the spacing and the orientation from one copy to the next.
4. Define the endpoint of the line and thus the ***Angle*** and ***Length*** (= distance from one copy to another) of the line: use either the cursor or enter the value in the Status dialog box.
   The value resulting from the number and distance is used to determine the ***Tot Length*** is displayed only for your information. You cannot enter a value yourself.
5. Select the objects that are to be multiplied.

Divide length

Defines the arrangement of the copies along the entire length of the line, and their distribution along it: you must specify the distance or the number of copies.

2. Define the starting point of the (imaginary) line on which the copies are to be arranged.
3. Define the endpoint of the line and thus the ***Angle*** and ***Length*** (=total length) of the line: use either the cursor or enter the value in the Status dialog box.

4. By dragging the cursor towards the starting point of the line you can dynamically determine the ***Number*** of copies and their ***Distance***.
The values can, alternatively, be entered in the Status dialog box.
If the result of the division of the length by the ***Distance*** does not result in a whole number, the next largest value for the spacing will be displayed for you to accept or reject.

5. Select the objects that are to be multiplied.

Multiply distance

Defines the arrangement by means of a spacing between one copy and the next and its duplication by specifying the total length or the quantity of copies.

2. Define the starting point of the (imaginary) line on which the copies are to be arranged.

3. Define a point that will thus specify the ***Angle*** and ***Distance*** between one copy and the next: use the cursor or enter the value in the Status dialog box.

4. By dragging the cursor towards this point and over it you can duplicate the spacing and so change the total length (= ***Distance***) and the ***Number*** of copies dynamically.
If the result of the division of the spacing duplication by the value entered does not result in a whole number for the quantity, the next smallest value for the total length will be displayed for you to accept or reject.

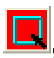
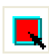
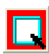
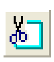
5. Select the objects that are to be multiplied.

The copies will be generated starting where the original object is on the basis of the parameters that you have specified.

Please note:

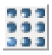
The objects to be multiplied can also be taken over in the selection list before loading the command.
The group can be selected by keeping the ALT key pressed whilst selecting objects or alternatively using the ***Group Manager*** to select one or more groups with the command ***Edit >>> Select Group***.

You can use the ***Settings / Object Filter*** menu to define the ***copy mode for objects***. Depending on the setting made here, only objects are copied or group structures are also copied.

, , ,  When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.

In the 3D-view window you cannot use this option except in the top view of the current workplane.

Multiply objects with sort in matrix spacing

,  Alternatively, the command can be started with these icons.

This command is for multiplying groups, 2D line objects, text and dimensions, which are contained in the active partial drawing.
The copies will be arranged with identical spacing on a matrix defined along two (imaginary) lines. The shared starting point is on the original object.

If you are multiplying solids in the 3D window, please note the additional options for Fixed Solids.

1. A dialog box will appear for you to specify the type of arrangement of the copies on the matrix axes.
   The selection in the upper section of the box is for specifying the type for the first of the matrix axes, and the lower section for the type for the second of them.

   The options are more or less the same as those used in ***Fixed solid, Multiply with sort in longitudinal spacing***.
2. Specify the parameters for the first of the matrix axes.
3. If the second axis is to form a right angle with the first, change the value presented in the box for ***Angle 2*** in the Status dialog box.
4. Specify the parameters for the first of the matrix axes.
5. Select the objects that are to be multiplied.

Starting at the position of the original objects the command will create the copies in accordance with the parameters that you have specified.

Please note:

The objects to be multiplied can also be taken over in the selection list before loading the command.
The group can be selected by keeping the ALT key pressed whilst selecting objects or alternatively using the ***Group Manager*** to select one or more groups with the command ***Edit >>> Select Group***.

You can use the ***Settings / Object Filter*** menu to define the ***copy mode for objects***. Depending on the setting made here, only objects are copied or group structures are also copied.

, , , When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.
In the 3D-view window you cannot use this option except in the top view of the current workplane.

Multiply objects with sort in Crown Spacing

, After clicking the displayed icons the command ***Transform, Copy on Crown Spacing*** is started.

This command is for multiplying groups, 2D line objects, text and dimensions, which are contained in the active partial drawing.

Contrary to the Copy on Matrix Spacing the copies are only placed on the outer crown, which is defined by two (ficticious) lines. The common start point of both lines is at the original object.

If you are multiplying solids in the 3D window, please note the additional options for Fixed Solids.

1. A dialog box will appear for you to specify the type of arrangement of the copies on the matrix axes.
   The selection in the upper section of the box is for specifying the type for the first of the matrix axes, and the lower section for the type for the second of them.

   The options are more or less the same as those used in ***Fixed solid, Multiply with sort in longitudinal spacing***.
2. Specify the parameters for the first of the matrix axes.
3. If the second axis is to form a right angle with the first, change the value presented in the box for ***Angle 2*** in the Status dialog box.
4. Specify the parameters for the first of the matrix axes.
5. Select the objects that are to be multiplied.

Starting at the position of the original objects the command will create the copies in accordance with the parameters that you have specified.

Please note:

The objects to be multiplied can also be taken over in the selection list before loading the command.
The group can be selected by keeping the ALT key pressed whilst selecting objects or alternatively using the ***Group Manager*** to select one or more groups with the command ***Edit >>> Select Group***.

You can use the ***Settings / Object Filter*** menu to define the ***copy mode for objects***. Depending on the setting made here, only objects are copied or group structures are also copied.

, **,** **,** When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.

In the 3D-view window you cannot use this option except in the top view of the current workplane.

Multiply objects with sort circular spacing

, Alternatively, the command can be started with these icons.

This command is for multiplying groups, 2D line objects, symbols, text and dimensions, which are contained in the active partial drawing. The copies will be arranged at identical distances from each other along an (imaginary) arc: the arc starts at the original object.

If you are multiplying solids in the 3D window, please note the additional options for Fixed Solids.

1. Select whether the circle for circular spacing is to be defined ***using centre/radius*** or ***using 3 points***.
Please note:
The distance of the centre of the circle to the original object defined, if the ***Also rotate objects*** option is deactivated, the radius of the circumference on which the copies are to be sorted.

2. Define for

- circular spacing ***using centre/radius,*** the rotation midpoint and then the radius of the circular spacing. As long as you do not wish to determine the copy vector from the drawing using snap points, the radius can be positioned anywhere in the drawing with the cursor.
- circular spacing ***using 3 points***, the circle of the circular spacing using 3 points in the drawing or by entering values.

3. Activate the setting ***also rotate objects*** inside the dialog box, if you want to arrange the copies on the centre of the arc.

4. Specify how the type of arrangement for the copies is to be defined.

Depending on the option, use one of the following procedures:

Angle and number
Defines the arrangement by means of the total angle of the arc on

which the copies are to be positioned, and by means of the quantity of copies.

5. Define a point in the drawing for the starting angle of the (imaginary) arc on which the objects are to be sorted. As long as you do not wish to determine the copy vector in the drawing by using snap points, you can position the starting point of the copy vector anywhere in the drawing using the cursor or by entering any value in the ***Angle*** text box of the Status dialog box.
   However, if you determine the vector using snap points from the drawing or if the objects are also to be rotated, select the respective snap point using the Snap Menu and identify the required object.
6. Type a number against ***Number*** in the Status dialog box to indicate the quantity of copies.
7. Define the endpoint that is the ***Open angle*** of the arc: use either the cursor or type a value in the Status dialog box.
   The value resulting from the opening angle and the number is the ***Angle*** between each pair of copies, but is displayed only for your information. You cannot enter a value yourself in this option.
8. Select the objects that are to be multiplied.

Angle distance and number
Defines the arrangement by means of the angle distance between each pair of copies and by means of the number of copies.

5. Define a point in the drawing for the starting angle of the (imaginary) arc on which the objects are to be sorted. As long as you do not wish to determine the copy vector in the drawing by using snap points, you can position the starting point of the copy vector anywhere in the drawing using the cursor or by entering any value in the ***Angle*** text box of the Status dialog box.
   However, if you determine the vector using snap points from the drawing or if the objects are also to be rotated, select the respective snap point using the Snap Menu and identify the required object.
6. In the ***Quantity*** text box of the Status dialog box, enter the number of copies.
   With the selected **angle distance and quantity** option, the number of copies are distributed over the maximum 360°. If the defined number exceeds 360° with the present angle distance, an error message appears.
   Remedy: reduce the angle distance between the copies by moving

the cursor clockwise about the rotation midpoint (without clicking with the mouse), so that the required number can be distributed under the maximum of 360°.

7. Define the endpoint of the arc and thus the ***Angle*** (lower): use either the cursor or enter a value in the Status dialog box.

If you want to copy objects over a full circle, simply move the cursor anticlockwise until you obtain a full circle of the circular spacing (opening angle = 360°) and confirm the angle by cursor in the drawing.

The value resulting from the ***Number*** and the ***Angle*** is the ***Open.angle***, but is displayed only for your information. You cannot enter a value yourself in this option.

8. Select the objects that are to be multiplied.

Divide angle

Defines the arrangement of the copies over the entire angle where they are to be placed, and the division of the angle by specifying an angle distance.

5. Define a point in the drawing for the starting angle of the (imaginary) arc on which the objects are to be sorted. As long as you do not wish to determine the copy vector in the drawing by using snap points, you can position the starting point of the copy vector anywhere in the drawing using the cursor or by entering any value in the ***Angle*** text box of the Status dialog box.
However, if you determine the vector using snap points from the drawing or if the objects are also to be rotated, select the respective snap point using the Snap Menu and identify the required object.

6. Define the endpoint and thus the ***Open.angle*** for the arc; use the cursor or enter a value in the Status dialog box.

7. By dragging the cursor towards the starting point of the angle you can dynamically determine the ***Number*** of copies and their *distance* by dividing the angle.
The values can, alternatively, be entered in the Status dialog box. If the result of the division of the opening angle by the ***Angle*** (lower) does not result in a whole number, the next largest value for the angle distance will be displayed for you to accept or reject.

8. Select the objects that are to be multiplied.

Multiply angle distance

Defines the arrangement by the angle distance between at least one of the copies and the next and the duplication by specifying the entire opening angle.

5. Define a point in the drawing for the starting angle of the (imaginary) arc on which the objects are to be sorted. As long as you do not wish to determine the copy vector in the drawing by using snap points, you can position the starting point of the copy vector anywhere in the drawing using the cursor or by entering any value in the ***Angle*** text box of the Status dialog box.
However, if you determine the vector using snap points from the drawing or if the objects are also to be rotated, select the respective snap point using the Snap Menu and identify the required object.

6. Define a point and thus the ***Angle*** between one copy and the next: use the cursor or enter a value in the Status dialog box.

7. By dragging the cursor towards and past this point you can duplicate the angle distance to determine the ***Open.angle*** dynamically and the ***Number*** of copies.
The values can, alternatively, be entered in the Status dialog box. If the result of the division of the duplication of the opening angle by the value of the ***Open.angle*** does not result in a whole number for the quantity, the next smallest value for the opening angle will be displayed for you to accept or reject.

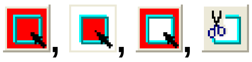
8. Select the objects that are to be multiplied.

Starting at the position for the original objects the copies will be generated in accordance with the parameters that have been specified.

Please note:

The objects to be multiplied can also be taken over in the selection list before loading the command.
The group can be selected by keeping the ALT key pressed whilst selecting objects or alternatively using the ***Group Manager*** to select one or more groups with the command ***Edit >>> Select Group***.

You can use the ***Settings / Object Filter*** menu to define the ***copy mode for objects***. Depending on the setting made here, only objects are copied or group structures are also copied.

 When selecting 2D objects by a window, the modes ***All, Inside, Outside*** or ***Clip*** can be chosen.
In the 3D-view window you cannot use this option except in the top view of the current workplane.

Modify 2D objects using geo points

This command can also be activated using this icon.

This command can be used to modify the size and position of 2D objects by modifying their geometry points.

The size in which the geo points are displayed fir this command can be determined using the menu command ***Settings, Options***.

1. If the selection list contains objects, these objects will be processed.
   If the selection list is empty, accept individual or a number of objects in the action list. The geo points are then shown by a marker for all the selected objects.
2. If required, load the following context menu by pressing the middle mouse button or using Ctrl + the right mouse button:

   Rotate About Geo Point
   Move Geo Point
   This can be used to determine whether the following objects are to be rotated about their geometry points or whether the geometry points are to be moved. The selected setting remains active until it is modified or until the command is aborted.

   Each time the command ***Transform, Geo Points*** is loaded, the ***Move Geo Point*** setting is active.
3. Identify the required geometry point.
4. Modify the position of the object or the geometry point depending on the setting:

   Rotate About Geo Point
   The identified geometry point is the centre for the rotation of the object. Define the rotation angle dynamically using the cursor. The Snap Menu commands are available for this - except ***Perpendicular To*** and ***Tangent To***.
   The Status dialog box can also be changed to and the value for the rotation angle entered in the ***Angle*** text box. The value 0 always refers to the position of the object before the modification.

   Move Geo Point: move objects
   The following geometry points enable objects to be moved:

   Midpoint of a line, centre of a circle, arc, ellipse, elliptical arc, base point of a symbol as well as reference point centre-centre of a text.

Move Geo Point: modify size and position of objects
All geometry points, except those mentioned above, enable the values of the respective object to be modified, by moving the identified point:

line	endpoints	length and angle
circle	quadrant points	radius
arc	endpoints	opening angle
	midpoint	radius
ellipse	quadrant points	radii
elliptical arc	endpoints	opening angle
	quadrant points	radii
spline	control points	interpolation or corner points of supporting point polyline
text	corners text box	height, width, distance
symbol	reference point	scale factor, rotation angle
pixel drawing	corner points	width, height

For a part of the object, the size value to be modified can be defined by moving to the Status dialog box and entering the respective value.

4.7 Draw construction aids

 This command can also be activated using this icon.

The commands within the Const Aids menu can be used to generate 2D objects (line, infinite line, circle, arc) in the drawing area of a 2D drawing window or in the current workplane (WP) of the 3D-view window.
If the 2D objects are drawn in a WP of the 3D window, they are linked with this WP. If the WP is processed or erased, all the 2D objects contained on it are affected by this action.

If the 3D-view window is activated, points which are defined for drawing construction lines or snapped on solid edges are generally projected in the selected WP.

The generated 2D objects are included and drawn into the structure of a model in the following way:

Group
The generated objects are saved in the current group. This can

previously be selected from the existing groups in the group selection in the main toolbar.

Colour, line type, line width
The 2D objects created with the commands in this menu are linked to the ***Construction Aid Layer*** defined in the ***Layer Explorer*** inside the ***Model Explorer***. This can be defined in the Layer Explorer by clicking the ***Construction Aid*** command button.

The objects are allocated the display attributes colour, line type and line width which are currently set for the layer.

The link with a layer as well as colour, line type and line width can later be modified using the menu command ***Process, Object Display***.

To ensure that when processing your drawing, only specific objects can be identified, deactivate the snap possibilities for all other object types using the menu command ***Settings, Object Filter***. Construction aids are managed as "normal" lines, circles, ellipses.

Draw construction aid - line

This command can also be activated using this icon.

This command can be used to draw construction lines.
If this is to be done using the cursor to specify angles at specified regular intervals, first select the menu command ***Settings, 3D/2D Commands***, and then use the ***2D draw*** option card to specify the ***Angle grid for construction aids***.
When you are using the Snap Menu or by pressing the ***Shift key*** the angle grid will be ignored.

1. Define the start point of the construction line.
2. Define the endpoint of the construction line.
   If you have activated an angle grid and require a direction which deviates from the defined line direction, the value can be entered in the ***Angle*** text box in the Status dialog box.

This is the standard procedure - other procedures available are those which can also be used for the command ***Draw 2D, Line***.

Draw construction aid - polyline

This command can also be activated using this icon.

This command can be used to draw polylines whose partial lines are construction lines.

The endpoint of a partial line is also the start point of the next partial line.
Each partial line of a construction polyline can be processed individually.

For the transition from one line to the next a rounding can be defined by entering a ***Radius*** in the status dialogue window. With this radius all transitions are rounded, as long as the radius can be defined due to the minimum length of a line.

This command can be used to draw construction lines.
If this is to be done using the cursor to draw angles at specified regular intervals, first select the menu command ***Settings, 3D/2D Commands***, and then use the ***2D draw*** option card to specify the ***Angle grid for construction aids***.
When you are using the Snap Menu or by pressing the ***Shift key*** the angle grid will be ignored.

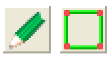
1. Define the start point of the construction line.

2. Define the endpoint of the construction line.
If you have activated an angle grid, but want a partial line that is to deviate from the direction already specified, the value can be entered in the ***Angle*** text box in the Status dialog box.

3. Repeat step 2 for all following partial lines.

This is the standard procedure - other procedures available are those which can also be used for the command ***Draw 2D, Polyline***.

Draw construction aid - rectangle

 This command can also be activated using this icon.

This command can be used to draw rectangles whose partial lines are construction lines.

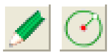
In order to define a construction rectangle, two diagonally opposite corner points should be determined as well as the (rotation) angle of the base line, if necessary. A rectangle can also be defined by 3 points, when first the 2 points in one axis are defined and then the third point in the second axis.

After starting the command a ***radius*** can be defined in the status block for the edges of the rectangle. These are created as long as the side length of the rectangle is at least double the value of the entered radius.

Each partial line of such a construction rectangle can be processed individually.

1. Define a corner point of the rectangle.
2. If the rectangle sides are not to run parallel to the coordinate axes, enter the angle in the ***Angle*** text box of the Status dialog box under which the base line of the rectangle.
   The angle always refers to the direction of the X-axis of the global coordinate system.
3. In the ***Length X*** and ***Length Y*** text boxes, enter the side lengths or define the diagonally opposite corner point.

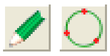
Draw construction aid - circle/arc dynamic Draw

 This command can also be activated using this icon.

Construction circles can be defined using the centre and a point or a perpendicular or tangent condition as well as by entering the radius. Construction arcs need the additional entry of start and opening angle.

The options and procedures available correspond to those in the CAD Menu command ***Draw 2D***, ***Dynamic Circle/ Arc***.

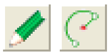
Draw construction aid - circle using 3 points

 This command can also be activated using this icon.

Construction circles are drawn using 3 points which do not lie on an infinite line.

The options and procedures available correspond to those in the CAD Menu command ***Draw 2D, Circle 3 Points***.

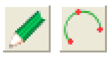
Draw construction aid - arc dynamic

 This command can also be activated using this icon.

Construction arcs can be defined using the centre and a point, by entering the radius as well as entering the start and opening angle.

The options and procedures available correspond to those in the CAD Menu command ***Draw 2D, Dynamic Arc***.

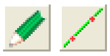
Draw construction aid - arc using 3 points

 This command can also be activated using this icon.

Construction arcs are drawn using 3 points which do not lie on an infinite line.

The options and procedures available correspond to those in the CAD Menu command ***Draw 2D, Arc 3 Points***.

Draw construction aid - infinite line

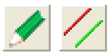
 This command can also be activated using this icon.

A construction infinite line is a "line" which stretches over the entire drawing area. Neither endpoints or a midpoint can be snapped on such a line.
If this is to be done using the cursor to draw angles at specified regular intervals, first select the menu command ***Settings, 3D/2D Commands***, and then use the ***2D draw*** option card to specify the ***Angle grid for construction aids***.
When you are using the Snap Menu or by pressing the ***Shift key*** the angle grid will be ignored.

Proceed as follows:

1. Define any point which is to lie on the infinite line.
2. Determine the direction of the infinite line:
   If you have activated an angle grid, but want a partial line that is to deviate from the direction already specified, the value can be entered in the ***Angle*** text box in the Status dialog box.

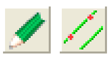
Draw construction aid - parallel to line

 This command can also be activated using this icon.

A construction parallel for a 2D line object (line, circle, arc, ellipse, elliptical arc, spline) is defined by the distance to this line as well as the side on which the parallel is to be drawn.

The options and procedures available correspond to those in the CAD Menu command ***Draw 2D, Parallel To Line***.

Draw construction aid - parallel infinite line to line

 After clicking on the displayed icons, the command ***Construction Aids, Parallel Straight To Line*** can be started.

This command can be used to draw parallels to individual existing lines (lines, circles, arcs, ellipses, elliptical arcs and splines) as well as ***parallel infinite lines to*** individual, existing ***line and infinite lines***. This is also valid for line objects in other partial drawings.

A parallel infinite line to a 2D line object (line, circle, arc, ellipse, elliptical arc, spline) is defined by the distance to this line as well as by the side on which the parallel is to be drawn.

1. If the 2D drawing window is active, you can use the dialog box to specify which partial drawings are to be considered during selection.

2. Identify the line which is to have a parallel drawn to it.

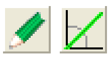
3. Determine the position of the parallel line dynamically using the cursor or use one of the following options:

Determine the position by defining a point on the parallel line. This point can also be on the imaginary extension of the parallel to be drawn.

The value for the distance to the original line can also be entered in the ***distance*** text box in the Status dialog box.
This is carried out on the side of the original line on which the cursor is positioned.

4. If, in the previous step, the value for the ***distance*** has been entered, this value is retained.
If this is the case, further lines can be identified to which parallels are to be drawn with the entered distance.
In this case, the parallels are drawn on the side of the original line on which the cursor was positioned during identification.

Draw construction aid - bisector

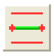
 This command can also be activated using this icon.

This command draws a construction infinite line, which halves the angle formed from two existing lines, construction lines or construction infinite lines.

A construction bisector is an infinite line which stretches over the entire drawing area. Neither endpoints or a midpoint can be snapped on such a line.

1. One after the other, identify two of the above mentioned objects (line, construction line, construction infinite line) which have a common "real" point of intersection.

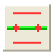
Help Line to Parallel Lines

,  The command ***Construction Aids***, ***Help Line to Parallel Lines*** is started using this icons.

This command is an equivalent to centre lines but creates a help line.

1. Identify two parallel lines after each other.

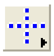
Straight Help Line to Parallel Lines

,  The command ***Construction Aids***, ***Straight Help Line to Parallel Lines*** is started using this icons.

This command is an equivalent to centre lines but creates a straight help line.

1. Identify two parallel lines after each other.

4.8 Draw centre lines

 This command can also be activated using this icon.

For the identification of symmetrical view or cuts of workpieces, the commands in the *Centre Lines* menu can be used to draw lines, circles and arcs as centre lines or to convert existing 2D line objects of these types into centre lines.

The commands generate 2D objects in the active partial drawing and in not active partial drawings when defined in the ***Select on listed Sketchboards*** command.or in the current workplane (WP) of the 3D-view window.

If the 2D objects are drawn in a WP of the 3D window, they are linked with this WP. If the WP is processed or erased, all the 2D objects contained on it are affected by this action.

If the 3D-view window is activated, points which are defined for drawing 2D centre lines or snapped on solid edges are generally projected in the selected WP.

The generated 2D objects are included and drawn into the structure of a model in the following way:

Group
The generated objects are saved in the current group. This can previously be selected from the existing groups in the group selection in the main toolbar.

The group allocation of the objects can be modified later using the ***Model Explorer***.

Colour, line type, line width
The 2D objects drawn with the commands from this menu are linked with the ***layer for centre lines***. This layer can be defined using the ***Layer Explorer*** inside the ***Model Explorer***.

The objects are allocated the display attributes colour, line type and line width which are currently set for the layer. This is not valid if the display attributes are defined object specific using the main toolbar.

The display of line type and line width is carried out according to the definitions made using the menu command ***Settings, Options***.

The link with a layer as well as the colour, line type and line width can later be modified using the menu command ***Process, Object Display***.

To ensure that when processing your drawing, only centre lines can be identified, deactivate the snap possibilities for ***Centre lines/3D axes*** using the menu command ***Settings, Object Filter*** and deactivate for all other object types.

Determine settings for centre lines and solid axes

Start the menu command ***Settings, 3D/2D Commands*** and change to the ***2D/3D*** *axes* option card.

define 2D-axes with overhang
define 3D-axes with overhang
The check boxes in front of this can be used to determine whether the value entered here for the overhang should be displayed in the Status dialog box when drawing 2D centre lines and solid axes. This value can be changed by loading one of the commands in the centre line menu or by loading the command ***Centre Lines, Define 3D Axes***.

If you generally wish to use a different value in your drawings to the one preset with the installation, simply define the required value in your template.

If the endpoint that is snapped on an axis with an overhang, the point does not actually lie on a visible end of the axis: it will be at the same distance as the length of the overhang.

Display properties of 2D axes
The display settings of 2D axes can be defined by layer or by object.

If colour, line type and line width is by layer (Icon coloured), all objects defined using the menu ***Centre Lines*** get their display attributes from the layer assigned to the layer for Centre_Lines.

When changing settings the attributes by layer are active (Icon greyed). This setting has a higher priority than the one defined in the layer Centre_Lines.

The assignment of 2D axes to a layer as well as to a colour, a line type and a line width can be edited using the menu ***Process, Object Display***.

display 3D axes
The check box in front of this can be used to determine whether solid axes are to be displayed in the 3D window.
The display attributed of the solid axes are not defined as layer specific, but just object specific. For colour selection, click on the respective icon to the right and for the definition of the line type, click on the respective list box.

Draw centre lines - line

This command can also be activated using this icon.

Proceed as follows:

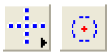
1. A centre line - line is drawn with an overhang on both sides (this differs from a normal line).
   Enter the respective value in the ***Overhang*** text box in the Status dialog box.
2. Draw the centre line - line in the same way as a normal line is drawn when using the command ***Draw 2D, Line***.

Please note:

The endpoints of the defined centre lines are only on the visible endpoints if the overhang has the value 0.
Otherwise they lie on the overhang value away from the visible endpoints on the lines.

Later, you can modify the overhang of centre lines using the ***Edit Object*** command from the ***Information*** command menu. In the command, identify a centre line and define the required overhang. This command and also the ***Centre Line To Parallels*** command can be used to define centre lines in the model views and sectional views from model views. With this, centre lines can be placed on the line objects using the Snap Menu.

Draw centre lines - circle

 This command can also be activated using this icon.

This command draws a centre line circle or arc.

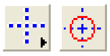
1. Define the centre of the arc (the circle).
2. If a full circle is to be drawn, specify its radius by an entry in the Status dialog box or by specifying a point on its circumference.

   If an arc is to be drawn, enter values in the two boxes ***Angle*** and ***Open.angle*** to specify the start and opening angle of the arc.

Please note:

If the start and/or the opening angle of the arc is to be changed, use the commands ***Process 2D Objects, Lengthen/Shorten*** or ***Information, Edit Object***.

Draw centre cross to circle

 This command can also be activated using this icon.

This command draws a centre cross to a circle or arc in the active partial drawing or in a not active partial drawings when defined in the ***Select on listed Sketchboards*** command whose lines run parallel to the coordinate axes.

1. Both axes of the centre cross are given a overhang on both sides. Enter this value in the ***Overhang*** text box in the Status dialog box.
2. Identify the circumference.

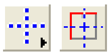
Please note:

The endpoints of these centre lines are only at the visible endpoints if the overhang has the value 0.
Otherwise, they lie the value of the overhang away from the visible endpoints on the lines, i.e. on the intersections with the circumference.

Later, you can modify the overhang of centre lines using the ***Edit Object*** command from the ***Information*** command menu. In the command, identify a centre line and define the required overhang.

This command can be used to bring in centre crossed to drill holes and tapped holes into model views by identifying the drill holes.

Draw centre cross to 2 lines

 This command can also be activated using this icon.

This command draws a centre cross to two non parallel lines in the active partial drawing or in a not active partial drawings when defined in the ***Select on listed Sketchboards*** command whose lines run parallel to the lines.

1. Both centre lines are given a overhang on both sides.
Enter this value in the ***Overhang*** text box in the Status dialog box.

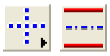
2. Identify both lines.

Please note:

The endpoints of the defined centre lines are only on the visible endpoints if the overhang has the value 0.
Otherwise they lie on the overhang value away from the visible endpoints on the lines.

Later, you can modify the overhang of centre lines using the ***Edit Object*** command from the ***Information*** command menu. In the command, identify a centre line and define the required overhang.

Centre lines to parallels

 After clicking on the displayed icons, the ***Centre Lines, Centre Lines To Parallels*** command is started.

This command draws a centre line to two parallel lines on the active partial drawing or in a not active partial drawings when defined in the ***Select on listed Sketchboards*** command or on the active workplane.

1. The centre line is allocated an overhang on both sides – this overhang is defined under ***Settings, 3D/2D Commands*** on the ***2D/3D Axes*** option card. For each centre line, once the command has been started, this value can be edited in the Status dialog box.

2. Identify the 2 parallel lines.

Please note

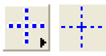
The endpoints of these centre lines only actually find themselves on the visible endpoints if the overhang has a value of 0.
Otherwise they will be positioned on the lines by the value of the overhang determining the distance from the visible endpoints.

The overhang of centre lines can later be edited using the command ***Edit Objects*** from the ***Information*** command menu.

When using this command, identify a centre line and define the required overhang.

This command can also be used to later define centre lines in model views and drill holes in the sectional view from model views.

Draw centre cross dynamic Draw

 This command can also be activated using this icon.

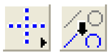
This command draws a centre cross whose lines run parallel to the coordinate axes.

1. Define the centre (= intersection of centre lines).
2. Determine the total length of the centre lines dynamically or by entering the value in the ***Length*** text box of the Status dialog box.
   The length can also be defined by defining another point:
   This corresponds to the distance to the centre.

Please note:

The centre lines drawn with these commands will be drawn without any overhang. This means that the endpoints are the visible endpoints.

Convert line into centre line

 This command can also be activated using this icon.

This command converts 2D line objects (line, circle and arc) of the active partial drawing or in a not active partial drawings when defined in the ***Select on listed Sketchboards*** command into centre lines, i.e. these lines are linked to the layer for centre lines and allocated the display attributes of this layer, if the graphic attributes are defined as by layer in ***Settings, 3D / 2D Commands, 2D / 3D Axes***.

1. If required, modify the value displayed in the ***Overhang*** text box of the Status dialog box.
2. Identify one or multiple lines by window to be converted to centre lines.
   Lines are lengthened on both sides by the value entered for the overhang.

Please note:

This command can also be proceeded in the Selection-Action mode to select the objects first and then start the command.
Please be ware that only those lines can be converted to centre

lines, which have no reference to other objects, e.g. no dimension is allowed to be depending on this line.

Convert centre line to line

, By clicking the shown icons the command ***Centre Lines, Centre Line to Line*** is started.

This command transforms centre lines in the active sketchboard or also in non active sketchboards, where the access is given through the command ***Select on ... sketchboards***, in 2D line objects (lines, circles and arcs). This line objects are then linked to the current layer and with the display properties of this layer or when the layer link is deactivated with the current object parameters.

1. Start the command with the displayed icons.

2. Identify a single centre line or multiple centre lines by dragging a rectangle with the pressed mouse button to transform them to 2D line objects.

Please note:

This command can also be executed in the Selection – Action mode, where first the objects are selected and then the command is started.
Please be aware to transform only those centre lines which have no reference to other objects, e.g. dimensions.

Centre line as parallel to a line

This command can also be activated using this icon.

This command is for drawing centre lines as parallels to other lines, circles or arcs.
A parallel is determined by its distance from another line and by the side on which the parallel is to be drawn.

1. Specify the partial drawings that are to be considered for selecting the original objects. Complete this step in the context dialog.

2. Identify the line to which the parallel is to be drawn.

3. Determine the position of the parallel line either by using the cursor or by using one of the methods described below:

Specify the position by defining the point on the parallel line or type a value for the ***Distance 1*** in the Status dialog box indicating the distance from the original line.

This will be calculated on the side of the original line where the cursor was positioned as the entry was made.

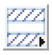
4. If a numerical value for the ***Distance 1*** has just been entered, this value will still apply.
   In this case you can identify other lines so that lines can be drawn parallel to them at the distance already specified.
   The parallels will be drawn on the side of the original line where the cursor was positioned during the identification procedure.

Please note:

If the object that has been identified is either a line or a construction line, the parallel centre line will be lengthened at both its endpoints by the value specified in ***Settings, 3D/2D commands*** on the option card for ***2D/3D axes*** or in the Status dialog box as the ***overhang***.

Later, you can modify the overhang of centre lines using the ***Edit Object*** command from the ***Information*** command menu. In the command, identify a centre line and define the required overhang.

4.9 Define and process faces

 This command can also be activated using this icon.

2D faces form the basis in order to *hatch*, *fill* and *clip* 2D objects. They can also be used to generate *swept solids* and *rotational solids*.

2D faces can be defined on sections which are bordered on all sides by 2D line objects (line, circle, arc, ellipse, elliptical arc, spline), i.e. they can be generated like these objects in the 2D window or in a workplane of the 3D window.

These commands can be used to generate, hatch, fill 2D faces as well as use them for the generation of swept solids and rotational solids. Commands such as ***Face Definition*** can be used specifically to generate an object of type 2D face.
These 2D faces can then directly be used to generate hatching, filling, etc.

2D faces are linked to the lines which border them, i.e. if the position of these lines is modified, the faces act **associative**, as long as these lines border the 2D faces completely. The associativity will be retained, if a rounding or chamfering is added to the corners of the border lines.

2D faces are destroyed if one of the bordering lines is erased. In addition to the 2D face any hatching and fillings on it will also be erased, and clipped sectors will be destroyed.

If it is possible to reconnect the endpoints of the boundary lines in such a way that an enclosed area will result, the program will normally be able to re-construct the 2D face.

The generated 2D faces are:

- saved in the current group determined using  *Select current 2D group from list.*
- linked with the active layer determined using  *layer selection*.
  If 2D faces are later highlighted in another colour using the command ***Face Definition, Fill Face*** this is carried out with the colour determined for the layer or object.

The group membership can be edited using the ***Model Explorer***, the assignment to a layer as well as the colour can be edited using the menu ***Process, Object Display***.

To ensure that whilst processing your drawing, only 2D faces can be identified, in the object filter, activate the snap possibilities for 2D faces and deactivate the other object types.

Please note:

2D faces that are not hatched or not filled will not appear on screen. This applies in particular to clip faces.
Such faces can be made visible by selecting the ***Settings, Options*** command button and by then activating the ***tagged*** option in the ***2D face display*** block.

Determine settings for hatch

After starting the ***Face Definition, Generate Hatch*** command, you can determine the hatching type, value and display attributes for hatchings.

If the settings are saved in a template, they are also available in all models generated on the basis of this template.

Generally, the dialog box differentiates line hatchings and symbol hatchings. By combining two lines, alternating line and cross hatchings can be generated. On the basis of these combination options, the following hatch types are available:

Simple line hatch

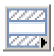
Alternating line hatch

Cross hatch

Symbol hatch

Depending on the selected hatch type, the following parameters are available for the definition of hatchings:

__Attributes Line 1__ and __Attributes Line 2__

,  and  In the dialog windows of the commands ***Create Hatch*** and ***Edit Hatch*** hatch parameters can be taken from existing hatches in the graphic.

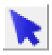
Especially when editing hatches not all the parameters have to be changed only the desired ones.

Start one of the commands and select ***Get parameter from graphic***. Then identify a hatch line from a hatch to take over their parameter.

Please note:
When rotating a hatched contour the hatch will also rotate if it is also selected. If only the contour is selected, the hatch is not rotated.

distance
The value entered here determines the distance between two neighbouring hatch lines.
The display of the distance is carried out in relation to the dimensions of the drawing, therefore is independent of the selected scale of the partial drawing.

Angle
Determines the hatch angle.

Start point
Activate this option in order to determine a point through which the hatching line should run. This is necessary if a number of hatchings are to be aligned. The starting point can be determined using absolute coordinates or by using the  command for point definition.

Display attributes
The standard display attributes colour, line type and line width which have been determined for the hatch layer using the ***Layer Explorer*** inside the ***Model Explorer*** are displayed.

If the following hatch lines are to be allocated their own display attributes independent of those predetermined, these can be set here.

Please note:

> Alternating line and cross hatchings exist of two individual hatchings, whose attributes are defined by ***Attributes_Line 1*** and ***Attributes Line 2***. Since only one current layer can be defined for hatchings, a separate layer allocation for ***Line 2*** is not possible.

Any display attribute that differ from ***Line 1*** must be defined as object attributes.

Layer attribute **Object attribute**
For each of the three display attributes it can be determined whether they are to be taken over from the hatch layer or they are to be allocated individual settings independent of those predetermined.
The active setting can be determined by clicking on the icon after the respective text box.

If a colour, line type or line width is determined using these icons, an object specific attribute is always defined.

If the display attributes of the layer are to be modified, use the menu command ***View, Model Explorer***, option card ***Layer Explorer***.

In the box *layer:*, the hatch layer is displayed which was defined using the menu command ***View, Model Explorer***, option card ***Layer Explorer***.
If another layer is to be selected for the hatch, open the list and select the respective layer in the dialog box which appears.

Please note:

By using the menu command ***File, Save/Load Parameter Sets***, the settings for hatchings can be loaded and taken over from a MPS file.

<u>Attribute symbol hatch</u>

distance x; distance y
The values entered here determine the distance between two neighbouring symbols in the X and Y directions.
The display of the distance is carried out in relation to the dimensions of the drawing, therefore is independent of the selected scale of the partial drawing.

angle
Enter the rotation angle for the symbols.

scale x; scale y
The values entered here determine the scale of the symbols in the X and Y direction.

Current symbol
This allows for the allocation of a symbol from one of the existing libraries whereby the hatching is to be generated on the basis of this

symbol. Select this command in order to select a symbol from a library.

Please note:
The use of symbols that contain faces, clip faces, fill faces or hatch faces, is not supported.

Start point
Activate this option in order to determine a point through which the hatching line should run. This is necessary if a number of hatchings are to be aligned. The starting point can be determined using absolute coordinates or by using the command for point definition.

Display attributes
The standard display attributes colour, line type and line width which have been determined for the hatch layer using the menu command ***View, Model Explorer***, option card ***Layer Explorer*** are displayed.

If the following hatch lines are to be allocated their own display attributes independent of those predetermined, these can be set here.

Please note:

> Alternating line and cross hatchings exist of two individual hatchings, whose attributes are defined by ***Attributes Line 1*** and ***Attributes Line 2***. Since only one current layer can be defined for hatchings, a separate layer allocation for ***Line 2*** is not possible. Any display attributes that differ from ***Line 1*** must be defined as object attributes.

Layer attribute **Object attribute**
For each of the three display attributes it can be determined whether they are to be taken over from the hatch layer or they are to be allocated individual settings independent of those predetermined.
The active setting can be determined by clicking on the icon after the respective text box.

If a colour, line type or line width is determined using these icons, an object specific attribute is always defined.

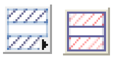
If the display attributes of the layer are to be modified, use the menu command ***View, Model Explorer***, option card ***Layer Explorer***.

In the box *layer:*, the hatch layer is displayed which was defined using the menu command ***View, Model Explorer***, option card ***Layer Explorer***.
If another layer is to be selected for the hatch, open the list and select the respective layer in the dialog box which appears.

Please note:

By using the menu command ***File, Save/load Parameter Sets*** you can load the settings for hatchings from a MPS file and take them over into other drawing files.

Generate hatch

 After clicking on the displayed icons, the ***Face Definition, Generate Hatch*** command is started.

This command hatches faces which are bordered completely by 2D line objects (line, circle, arc, ellipse, elliptical arc, spline) according to the current settings.
The only 2D line objects to be treated as borders are those that are in the current partial drawing or in the current workplane.

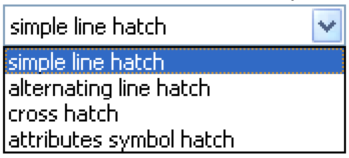
Use one of the following options to generate a hatching:

Generate hatch with selection list

1. Take over the respective bordering lines into the selection list **before** loading this command.
   The bordering lines must completely border at least one section of the drawing area completely.

2. In the dialog box, determine the settings required for the hatch.
   To get the parameters from an existing hatch start one of the commands and select ***Get parameter from graphic***. Then identify a hatch line from a hatch to take over their parameter.

3. In the selection list, determine the hatch type.

4. Start the command using the ***Hatch*** command button. The dialog box is closed automatically and appears again once the hatch has been generated and the command is quit using the right mouse button.

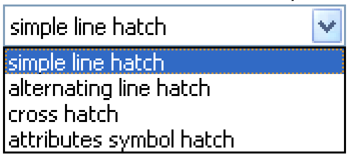
   Position the cursor within the section to be hatched and press the left mouse button to confirm.
   If **one** hatching is to be defined on sections divided in space, keep the *Ctrl key* pressed during this work step in order to collect the individual faces.

If the lines contained in the selection list do not completely border any section of the drawing area, an error message is displayed and no hatching is drawn.

Generate hatch with action list

1. In the dialog box, determine the required settings for the hatch. To get the parameters from an existing hatch start one of the commands and select ***Get parameter from graphic***. Then identify a hatch line from a hatch to take over their parameter.

2. In the selection list, determine the hatch type.



3. If the boundary objects for the defined hatching are selected using window selection and if hatchings or 2D faces already lie within this area, deactivate the ***Activate 2D Face*** option. Otherwise additional hatches are generated on every selected face.

4. If the selection list is empty, **after** loading the command determine the border object, i.e. an action list is defined for this command. The 2D line objects in the action list must completely border at least one section of the drawing area. Since as individual objects, only full circles, full ellipses and closed splines completely border a section, the selection of border objects is usually determined by collection (*Ctrl key*) or by defining a window.

If the 2D line objects contained in the action list, do not completely border a section of the drawing area, an error message is displayed and no hatching is drawn.

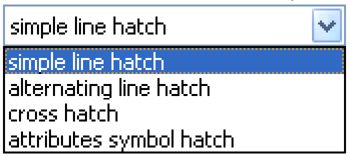
5. Position the cursor within the section to be hatched and press the left mouse button to confirm. If **one** hatching is to be defined on sections divided in space, keep the *Ctrl key* pressed during this work step.

Generate hatch on existing 2D faces

If the ***Define Face*** command has been used to generate one or more 2D faces, the hatching can be generated directly within such a 2D face. This is also valid for 2D faces which have already been hatched.

1. In the dialog box, determine the settings for the hatch. To get the parameters from an existing hatch start one of the commands and select ***Get parameter from graphic***. Then identify a hatch line from a hatch to take over their parameter.

2. In the selection list, determine the hatch type.



3. Using the ***Activate 2D Faces*** option or the ***Settings, Object Filter*** menu command, ensure that 2D faces can be snapped.

4. Position the cursor within the 2D face to be hatched and press the left mouse button to confirm.

Consider existing 2D faces

Activate 2D Faces determines whether new hatchings are to be generated within the window selection when touching existing hatchings or faces. Deactivate this option in order to deactivate the object type ***2D Face***.

Please note:

Generate hatch with selection list or *Generate hatch with action list* a 2D face is generated within the selected section. This forms the basis for the hatching. If a hatching already exists, a 2D face has also been generated.

If a 2D face is to be rehatched, use the procedure *Generate hatch on existing faces* since no new 2D faces are generated in this way.

A hatch is an object whose basis is an object of type 2D face. Like the 2D face, the hatch acts associatively if the border lines of the face are moved and it is destroyed if one of the border lines is erased and the 2D face is therefore destroyed.

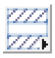
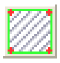
- The generated 2D faces and hatches are saved together in the current group determined using *group selection*.
- The generated 2D faces are linked with the active layer determined using ***layer selection***, the hatching with the set hatch layer using ***Settings, Hatch***.
- The hatching is allocated the display attributes colour, line type and line width which are currently set for the hatch layer or object.

The lines of a hatching cannot be processed individually. In order to modify the settings of a hatching, the command ***Face Definition, Edit Hatch*** can be used. Alternatively, a hatching can be disassembled using the commands ***Disassemble Hatch*** and ***Disassamble 2D Objects*** in order to be able to process individual objects from the hatching. All hatching attributes such as e.g. the associativity of the hatchings are therefore lost.

Group allocation of 2D faces and hatchings can be modified using the ***Model Explorer***, the link with a layer as well as the display attributes of hatchings using the menu command ***Process, Object Display.***

To ensure that whilst processing your drawing, only hatchings or 2D faces can be identified, use the menu command ***Settings, Object Filter*** to activate the snap possibilities for hatchings or 2D faces and deactivate the other object types.

Generate hatching polyline

  After clicking on the displayed icons the ***Face Definition, Generate Hatch Polyline*** command is started.

In the 2D window or the current workplane of the 3D window, this generates a closed polyline.
Within this closed polyline, a 2D face and a hatching is generated. The line and the object 2D face are placed in the current working layer and take over the determined properties or the currently determined object properties. The hatching is placed on the layer predefined for hatchings and takes over those properties or those determined using the menu ***Settings, Hatch***. This option can also be used to predefine alternating, cross or symbol hatchings.

1. Generate a closed polyline from at least two partial lines.
The start point of the first partial line is connected like a rubberband to the endpoint of the last partial line. A closed polyline is therefore generated.
Design the closed polyline using the cursor, using snap points or by entering values in the Status dialog box. Even value definition is possible using relative coordinates (key combination Ctrl+f for the snap mode with relative coordinates).

2. Finish the generation of further partial lines by pressing the right mouse button.
The 2D face within the closed polyline is then generated and allocated a hatching.

The generated polyline objects, the 2D faces and the hatchings are saved together in the current group determined using *group selection.*

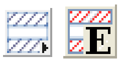
The generated polyline objects and 2D faces are linked with the active layer and the hatchings are linked to the layer predefined for hatchings.

The colour display of the hatchings is carried out with the predetermined colour set using ***Settings, Hatch***.

The group allocation of the polyline partial lines, the 2D faces and the hatchings can be modified in the Model Tree of the ***Model Explorer***, the link to a layer as well as the display attributes of the partial lines and the colour of the hatchings with the menu command ***Process, Object Display***.

To ensure that whilst processing your drawing, only 2D faces or partial lines of the bordering polyline can be identified, use ***Settings, Object Filter*** to activate the snap possibilities for 2D faces or lines and deactivate the other object types.

Process face hatch

 After clicking on these icons, the ***Face Definition, Modify Hatch*** command is started.

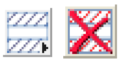
This command can be used to modify the settings of hatchings.

1. After starting the command, a dialog box is opened which contains the settings used for the last hatching.
To get the parameters from an existing hatch start one of the commands and select ***Get parameter from graphic***. Then identify a hatch line from a hatch to take over their parameter.

2. Edit the parameters in the same way as for ***Create Hatch***.

3. Identify the hatch, the parameters should be assigned to.

Disassemble hatch

 After clicking the displayed icons, the ***Face Definition, Disassemble Hatch*** command is started.

This command can be used to disassemble line hatchings into individual line objects. The hatching attributes such as e.g. the associativity are lost.
In the command, identify the line of a hatching that is to be disassembled. Then decide whether the 2D face is to be erased.

Please note:

A group with the name "Hatch" is set up automatically for the disassembled hatching. In this way, the connection between the hatching lines is retained using group selection (SHIFT / ALT keys).

As long as the hatching has not been defined with object attributes, the lines become the attributes defined on the hatching layer.

, Hatches can also be disassembled using the ***Process 2D***, ***Disassemble 2D Objects*** command.

Fill face

This command can also be activated using this icon.

This command fills 2D faces which are bordered completely by 2D line objects (line, circle, arc, ellipse, elliptical arc, spline) with the colour which is preset for the currently selected layer or for the individual object.

The only 2D line objects to be treated as borders are those that are in the current partial drawing or in the current workplane.

A filling is simply the colour display of a 2D face.

If a 2D face already exists, use the first procedure.
Otherwise use one of the following options.

Fill existing 2D faces

If a 2D face has previously been generated, this can be filled directly.

This is also valid for 2D faces which have been hatched.

1. Position the cursor within such a 2D face and press the left mouse button to confirm.
   The requirement for this is that 2D faces can be snapped - determined using the menu command ***Settings, Object Filter***.

Generate face with filling using selection list

1. **Before** loading the command, accept the respective border lines into the selection list. The border lines must completely border at least one section of the drawing area.
2. Start the command.
   Position the cursor within the section where the 2D face is to be generated and press the left mouse button to confirm.
   If **one** object of type 2D face is to be generated and filled on sections divided in space, keep the Ctrl key pressed during this step.

If the 2D line objects contained in the selection list do not completely border any section of the drawing area, an error message is displayed and no filling is drawn.

Generate face with hatching using action list

1. If the selection list is empty, **after** loading the command determine the border lines, i.e. an action list is defined for this command.
The 2D line objects in the action list must completely border at least one section of the drawing area.
Since as individual objects, only full circles, full ellipses and closed splines completely border a section, the selection of border objects is usually determined by collection (*Ctrl key*) or by defining a window.

If the 2D line objects contained in the action list, do not completely border a section of the drawing area, an error message is displayed and no filling is drawn.

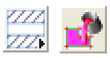
2. Position the cursor within the section where the 2D face is to be generated and press the left mouse button to confirm.
If **one** 2D face is to be generated and filled on sections divided in space, keep the Ctrl key pressed during this step.

Please note:

To ensure that whilst processing your drawing, only 2D faces can be identified, use ***Settings, Object Filter*** to activate the snap possibilities for 2D faces and deactivate the other object types.

The colour of the face and therefore the filling can be modified using the menu command ***Process, Object Display***.

Generate fill polyline

 This command can also be activated using this icon.

This command generates a closed polyline in the 2D window or in the current workplane of the 3D window.
Within the closed polyline, a 2D face is generated which is filled with the currently set colour.
A filling is simply the colour display of a 2D face.

1. Generate a polyline out of at least two different partial lines.
The start point of the first partial line is connected like a rubberband to the endpoint of the last partial line. A closed polyline is therefore generated.

2. Finish the generation of further partial lines by pressing the right mouse button.
The 2D face within the closed polyline is then generated and filled with a predetermined colour.

The generated polylines and 2D faces are saved together in the current group determined using *group selection.*

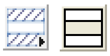
The generated polylines and 2D faces are linked with the active layer determined using *layer selection*.

The colour display of the 2D faces is carried out with the predetermined colour for the layer or object.

The group allocation of the polyline partial lines and 2D faces can be modified in the ***Model Explorer***, the link to a layer as well as the display attributes of the partial lines with the menu command ***Process, Object Display***.

To ensure that whilst processing your drawing, only 2D faces or partial lines of the bordering polyline can be identified, use ***Settings, Object Filter*** to activate the snap possibilities for 2D faces or lines and deactivate the other object types.

Define face

 This command can also be activated using this icon.

This command generates objects of type 2D face within sections which are bordered on all sides by 2D line objects.

The only 2D line objects to be treated as borders are those that are in the current partial drawing or in the current workplane.

Generate face with selection list

1. **Before** loading the command, accept the respective border lines into the selection list. The border lines must completely border are least one section of the drawing area.

2. Start the command.
Position the cursor within the section in which the 2D face is to be generated and press the left mouse button to confirm.
If **one** 2D face is to be defined on sections divided in space, keep the *Ctrl key* pressed during this step.

If the lines contained in the selection list do not completely border any section of the drawing area, an error message is displayed and no 2D face is generated.

Generate face with action list

1. If the selection list is empty, **after** loading the command determine the border lines, i.e. an action list is defined for this command.
The objects in the action list must completely border at least one

section of the drawing area.
Since as individual objects, only full circles, full ellipses and closed splines completely border a section, the selection of border objects is usually determined by collection (*Ctrl key*) or by defining a window.

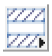
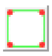
If the 2D line objects contained in the action list, do not completely border a section of the drawing area, an error message is displayed and no 2D face is generated.

2. Position the cursor within the section in which the 2D face is to be generated and press the left mouse button to confirm.
If **one** 2D face is to be defined on sections divided in space, keep the *Ctrl key* pressed during this step.

Please note:

2D faces that are not hatched or not filled will not appear on screen. This applies in particular to clip faces.
Such faces can be made visible by selecting the ***Settings, Options*** command button and by then activating the ***tagged*** option in the ***2D face display*** block.

Define Face by Polyline

,  After clicking on the displayed icons the ***Face Definition, Face by Polyline*** command is started.

In the 2D window or the current workplane of the 3D window, this generates a closed polyline. Within this closed polyline, a 2D face and a hatching is generated. The line and the object 2D face are placed in the current working layer and take over the determined properties or the currently determined object properties.

1. Generate a closed polyline from at least two partial lines.
The start point of the first partial line is connected like a rubberband to the endpoint of the last partial line. A closed polyline is therefore generated.
Design the closed polyline using the cursor, using snap points or by entering values in the Status dialog box. Even value definition is possible using relative coordinates (key combination Ctrl+F for the snap mode with relative coordinates).

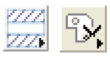
2. Finish the generation of further partial lines by pressing the right mouse button.
The 2D face within the closed polyline is then generated and allocated a hatching.

The generated polyline objects and the 2D face are saved together in the current group determined using *group selection.*

The group allocation of the polyline partial lines, the 2D faces and the hatchings can be modified in the Model Tree of the ***Model Explorer***, the link to a layer as well as the display attributes of the partial lines and the colour of the hatchings with the menu command ***Process, Object Display***.

To ensure that whilst processing your drawing, only 2D faces or partial lines of the bordering polyline can be identified, use ***Settings, Object Filter*** to activate the snap possibilities for 2D faces or lines and deactivate the other object types.

Use clip face

 This command can also be activated using this icon.

When generating clip faces and clip polyline, objects of type 2D face are allocated a drawing order of 60 and are displayed in the currently set background colour.
They can be used to make other 2D objects which lie in their face completely or partially "invisible".

Clip faces are by their very nature not visible and so it is advisable to tag them before they are created or processed, and to tag the objects concealed by them.
This is done by using the ***Settings, Options*** command button and then by activating the ***tagged*** option in the ***Line objects and markers*** block.
Please Note:
The drawing order values ***-1*** and ***-2*** are reserved for special layer functionallity and should not be used for 2D objects.

Generate clip face

 This command can also be activated using this icon.

This command generates 2D faces within section bordered on all sides by 2D line objects (line, circle, arc, ellipse, elliptical arc, spline) and clips these 2D faces:
2D objects which lie completely or partially within this 2D face, are hidden. The result is that the 2D face is allocated the drawing order 60.
This display cannot be generated unless you have used the ***Settings, Options*** command button to deactivate the display of the faces.

The only 2D line objects to be treated as clip faces are those that are in the current partial drawing or in the current workplane.l

Like the object of type 2D face, the clip acts associatively if the border lines of the 2D face are moved and it is destroyed if one of the border lines is erased and the 2D face is therefore destroyed.

If a 2D face already exists, use the first procedure.
Otherwise use one of the following options.

Clip existing 2D faces

If a face has previously been generated with the command ***2D Face Definition, Define Face*** this can be clipped directly. This is also valid for 2D faces which have been hatched.

1. Position the cursor within the 2D face to be clipped and press the left mouse button to confirm.

2. If no other 2D faces are to be clipped, press the right mouse button.

3. If individual or a number of objects which lie in the section of the defined clip faces are still to be visible, i.e. the borders of a 2D face, accept these objects in an action list.

4. If the results of this action are not displayed correctly, use the command ***2D Face Definition, Clip Face, Regenerate Display*** or redraw the drawing.

Clip face using selection list

1. **Before** loading the command, accept the respective border lines into the selection list. The border lines must completely border at least one section of the drawing area.

2. Start the command.
Position the cursor within the section where the 2D face is to be generated and press the left mouse button to confirm.

If the 2D line objects contained in the selection list do not completely border any section of the drawing area, an error message is displayed and no 2D face will be defined: no face will be clipped.

3. If no other faces are to be clipped, press the right mouse button.

4. If individual or a number of objects which lie in the section of the defined clip faces are still to be visible, i.e. the borders of a 2D face, accept these objects in an action list.

5. If the results of this action are not displayed correctly, use the command ***2D Face Definition, Clip Face, Regenerate Display*** or redraw the drawing.

Clip face using action list

1. If the selection list is empty, **after** loading the command determine the border lines, i.e. an action list is defined for this command.
The border lines in the action list must completely border at least one section of the drawing area.
Since as individual objects, only full circles, full ellipses and closed splines completely border a section, the selection of border objects is usually determined by collection (Ctrl key) or by defining a window.

If the 2D line objects contained in the action list, do not completely border a section of the drawing area, an error message is displayed and no 2D face will be defined: no face will be clipped.

2. Position the cursor within the section where the 2D face is to be generated and press the left mouse button to confirm.

3. If no other faces are to be clipped, press the right mouse button.

4. If individual or a number of objects which lie in the section of the defined clip 2D faces are still to be visible, i.e. the borders of a face, accept these objects in an action list.

5. If the results of this action are not displayed correctly, use the command ***2D Face Definition, Clip Face, Regenerate Display*** or redraw the drawing.

If 2D faces are generated with this command, they are

- saved in the current group *determined using group selection*
- linked with the active layer determined using *layer selection*.

The group allocation of the 2D faces can be modified in the ***Model Explorer***, the link to a layer as well as the display attributes with the menu command ***Process, Object Display***.

Please note:

With *Clip face using selection list* or *Clip face using action list* a 2D face is generated within the selected section. This forms the basis for the clipping.

If all the objects hidden by a clipped face are to be made visible again, use the command ***2D Face Definition, Clip Face, Deactivate Clip***.

If only individual objects hidden by a clipped face, are to be made visible again use the command ***2D Face Definition, Clip Face, Objects To Front***.
In order to be able to see and clearly identify the 2D faces whose

borders are possible also clipped, activate the ***tagged*** setting using the menu command ***Settings, Options***.

To ensure that whilst processing your drawing, only 2D faces can be identified, use ***Settings, Object Filter*** to activate the snap possibilities for 2D faces and deactivate the other object types.

The drawing order values ***-1*** and ***-2*** are reserved for special layer functionallity and should not be used for 2D objects.

Generate clip polyline

 This command can also be activated using this icon.

This command generates a closed polyline in the 2D window or in the current workplane of the 3D window.
Within the closed polyline, a 2D face is generated which is clipped, i.e. objects which lie completely or partially within this 2D face will be hidden. The result is that the 2D face is allocated the drawing order 60. The requirement for this is that the *not tagged* setting is active using the menu command *Settings, Options*.

Like the object of type 2D face, the clip acts associatively if the border lines of the 2D face are moved and it is destroyed if one of the border lines is erased and the 2D face is therefore destroyed.

1. Generate a polyline out of at least two different partial lines.
The start point of the first partial line is connected like a rubberband to the endpoint of the last partial line. A closed polyline is therefore generated.

2. Finish the generation of further partial lines by pressing the right mouse button.
The 2D face within the closed polyline is then generated and filled in the background colour.

3. If individual or a number of objects which lie in the section of the defined clip faces are still to be visible, i.e. the borders of a 2D face, accept these objects in an action list.

4. If the results of this action are not displayed correctly, use the command ***2D Face Definition, Clip Face, Regenerate Display*** or redraw the drawing.

The generated polylines and 2D faces are saved together in the current group determined using *group selection.*

The generated polylines and faces are linked with the active layer determined using *layer selection*.

The group allocation of the polyline partial lines and 2D faces can be modified in the ***Model Explorer***, the link to a layer as well as the display attributes of the partial lines with the menu command ***Process, Object Display***.

Please note:

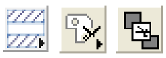
If all the objects hidden by a clipped face are to be made visible again, use the command ***2D Face Definition, Clip Face, Deactivate Clip***.

If only individual objects hidden by a clipped face, are to be made visible again use the command ***2D Face Definition, Clip Face, Objects Visible***.
In order to be able to see and clearly identify the faces whose borders are possible also clipped, activate the *tagged* setting using the menu command ***Settings, Options***.

To ensure that whilst processing your drawing, only 2D faces can be identified, use ***Settings, Object Filter*** to activate the snap possibilities for 2D faces and deactivate the other object types.

The drawing order values ***-1*** and ***-2*** are reserved for special layer functionallity and should not be used for 2D objects.

Activate clip

 This command can also be activated using this icon.

This command clips a previously defined 2D face, i.e. objects which lie completely or partially within this 2D face will be hidden. As a result the 2D face will be allocated 60 as its drawing order.

The requirement for this is that the ***tagged*** setting is active using the menu command ***Settings, Options***.

Like the 2D face, the clip acts associatively if the border lines of the face are moved and it is destroyed if one of the border lines is erased and the 2D face is therefore destroyed.

1. If the selection list contains one or more 2D faces, as soon as the command is loaded these directly become clipped faces.
2D faces are only clipped once the faces are removed from the selection list.
If the selection list does not contain any 2D faces, use the cursor to identify the 2D faces to be clipped or drag a window to a size which contains the required 2D faces.

2. If the results of this action are not displayed correctly, use the command ***2D Face Definition, Clip Face, Regenerate Display*** or redraw the drawing.

Please note:

To ensure that whilst processing your drawing, only 2D faces can be identified, deactivate the snap options for all other object types using the menu command ***Settings, Object Filter***.

If all the objects hidden by a clipped face are to be made visible again, use the command ***2D Face Definition, Clip Face, Deactivate Clip***.

If only individual objects hidden by a clipped face, are to be made visible again use the command ***2D Face Definition, Clip Face, Objects Visible***.
In order to be able to see and clearly identify the 2D faces whose borders are possible also clipped, activate the ***tagged*** setting using the menu command ***Settings, Options***.

The drawing order values ***-1*** and ***-2*** are reserved for special layer functionallity and should not be used for 2D objects.

Deactivate clip

 This command can also be activated using this icon.

This command cancels clippings on 2D faces, i.e. all objects which are completely or partially hidden will be made visible again. The result is that the 2D face is allocated the “normal” drawing order 0.

The object of type 2D face remain.

1. If necessary - activate the ***tagged*** option using the ***Settings, Options*** in menu command in order to make the selected clipped faces visible again.

2. If the selection list contains one or more clipped 2D faces, as soon as the command is loaded these directly become "normal" 2D faces, without clipping. The previously hidden objects are only displayed once the objects and 2D faces are removed from the selection list.

If the selection list does not contain any clipped faces, use the cursor to identify the faces to have the clipping cancelled or drag a window to a size which contains the required faces.

3. If the results of this action are not displayed correctly, use the command ***2D Face Definition, Clip Face, Regenerate Display*** or redraw the drawing.

Please note:

To ensure that whilst processing your drawing, only 2D faces can be identified, deactivate the snap options for all other object types using the menu command ***Settings, Object Filter***.

If only individual objects hidden by a clipped face, are to be made visible again use the command ***2D Face Definition, Clip Face, Objects Visible***.

The drawing order values ***-1*** and ***-2*** are reserved for special layer functionallity and should not be used for 2D objects.

Move object to background

 This command can also be activated using this icon.

This command moves visible 2D objects, which lie in the section of a clipped face, to the background, i.e. they are completely or partially hidden by the clipped faces. The result is that the objects are allocated the “normal” drawing order 0.

1. If the selection list contains one or more objects which are visible in sections of clipped faces, all these are completely or partially hidden directly after the command is activated.

If the selection list does not contain objects which are visible in sections of clipped faces any clipped faces, use the cursor to identify the objects to be hidden or drag a window to a size which contains the required objects.

2. If the results of this action are not displayed correctly, use the command ***2D Face Definition, Clip faces, Regenerate Display*** or redraw the drawing.

Please Note:

The drawing order values ***-1*** and ***-2*** are reserved for special layer functionallity and should not be used for 2D objects.

Move object to foreground

 This command can also be activated using this icon.

This command moves 2D objects, which are hidden completely or partially by a clipped face, to the foreground, i.e. they are made visible again. This is achieved by allocating the objects the drawing order 120.

1. If necessary - activate the ***tagged*** option using the menu command ***Settings, Options*** in order to make the selected clipped faces visible again.

2. If the selection list contains one or more objects which are hidden by clipped faces, all these are completely or partially hidden directly after the command is activated.

   If the selection list does not contain objects which are hidden by clipped faces, use the cursor to identify the required objects or drag a window to a size which contains the required objects.

3. If the results of this action are not displayed correctly, use the command ***2D Face Definition, Clip Face, Regenerate Display*** or redraw the drawing.

Please note:

If all the objects hidden by a clipped face are to be made visible again, use the command ***2D Face Definition, Clip Face, Deactivate Clip***.

The drawing order values ***-1*** and ***-2*** are reserved for special layer functionallity and should not be used for 2D objects.

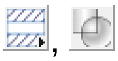
Regenerate clip display

 This command can also be activated using this icon.

This command redraws all clipped faces and all objects lying in the section of clipped faces according to the current presettings for visible and hidden.

In this way, only those objects which are concerned with clipping are updated without having to redraw the entire drawing.

Unite 2D Faces

 After clicking the displayed icons the command ***Face Definition***, ***Union*** is started.

This command is used to unite one 2D face with one other or several other 2D faces, which means that the resulting face consists of all selected faces.

1. Identify the first face.
2. Define one or several other faces by selection, drag a window or collect with the pressed CTRL key. The selection may also contain the face defined in step 1.

The display properties of the resulting face will be assigned to the first identified face.

Subtract 2D Faces

, After clicking the displayed icons the command ***Face Definition, Subtraction*** is started.

This command is used to subtract one or several faces from another face, which means that the base face is reduced by one or several other faces. The subtraction faces will remain.

1. Identify the first face.
2. Define one or several other faces by selection, drag a window or collect with the pressed CTRL key. The selection may also contain the face defined in step 1.

The display properties of the resulting face will be assigned to the first identified face.

Intersect 2D Faces

, After clicking the displayed icons the command ***Face Definition, Intersection*** is started.

This command is used to create a face which consists of the intersection which is the common part of all the selected faces.

1. Identify the first face.
2. Define one or several other faces by selection, drag a window or collect with the pressed CTRL key. The selection may also contain the face defined in step 1.

The display properties of the resulting face will be assigned to the first identified face.

4.10 Text

This command can also be activated using this icon.

Text is saved in the active partial drawing that is part of the 2D drawing already selected, or is in the current WP of the 3D window.

Text is always drawn with the same values for text height, width and spacing independent of the scale of the partial drawing.

Different text styles with different parameters (font, height, width, distance ...) can be saved using the menu command ***Settings***, ***Text***.

When defining text, these settings are taken over as suggested values. They can however be modified using the option card ***Local settings***.

The generated text is included and drawn into the structure of a model in the following way:

Group
The text is saved in the current group. This can be selected from the existing groups using group selection in the main toolbar.
Group allocation can later be modified in the ***Model Explorer***.

Colour, line type, line width
The text is linked with the layer for *text* which was previously determined using the menu command ***Layer Explorer*** *inside the* ***Model Explorer.***

When you are using vector fonts (*.FNT), the texts will be allocated the display attributes, colour and line width that are currently specified for the layer. Truetype fonts will be allocated only the specified colour.

The link with a layer as well as the display attributes can later be modified using the menu command ***Process, Object Display***.

The text content as well as the respective parameters can be modified using the command ***Text, Edit***.

To ensure that whilst processing your drawing, only text can be identified, use ***Settings, Object Filter*** to activate the snap possibilities for text and deactivate the other object types.

Settings for texts, dimension texts, referenced texts and position numbers

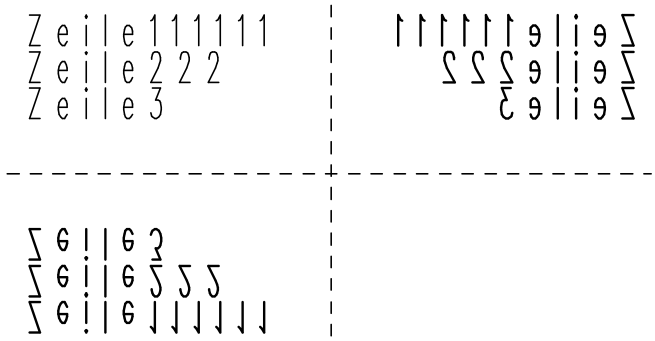
Using the menu command ***Settings, Text*** different text styles for dimensions, texts and position numbers can be saved. They can also be used for different styles of texts. For position numbers and alligned texts also the graphic properties are saved.

The settings defined in this dialog box can be saved as ***current text style***, ***current style for position numbers*** and ***current style for dimension*** texts.
Different styles can be saved and selected when needed.

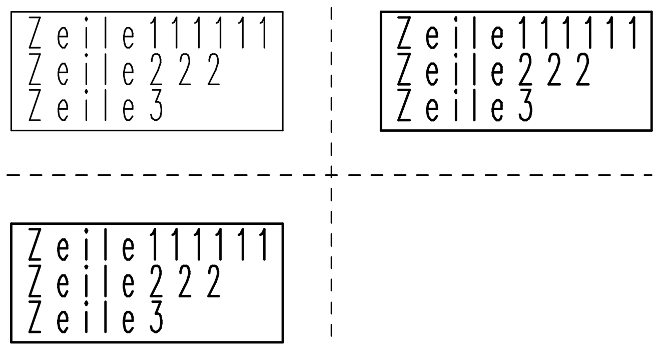
The current styles are always offered when choosing the commands Text, Define Text; in all dimensions (also in ***Surface Characteristics***, ***Form / Position Tolerances***, ***Welding Symbols*** and ***Table Dimensions***) as a suggestion.

Mirror relation
Here, one of three settings can be determined. It defines how existing text are to behave in the drawing, if they are processed with one of the commands ***Transform, Mirror*** or ***Mirror With Copy***.
These options are not valid for circular text and TrueType text.

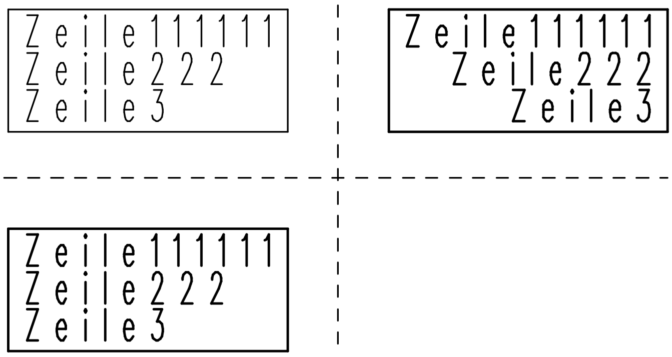
Mirror completely
With this setting, the text is mirrored completely:



Modify position
With this setting, an imaginary box that surrounds the text, is mirrored. The text is inserted in the box whilst retaining its parameters.



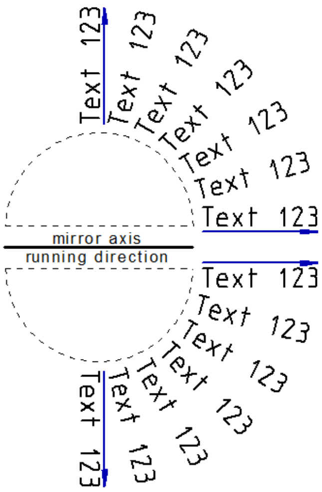
Modify position and reference point
With this setting, an imaginary box that surrounds the text, is mirrored. The text is inserted in the box. If the angle difference between the mirror axis and the base line of the text is larger than 45°, the reference point and therefore the conciseness of the text is also modified, e.g.:

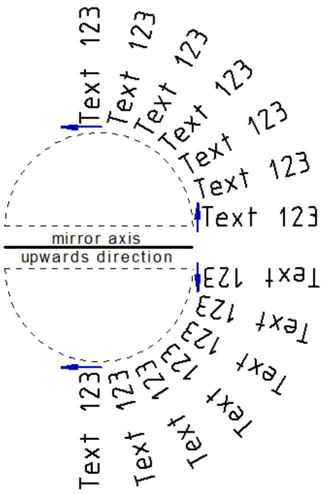


With the options for the mirror behavior of texts "modify position" and "modify position and reference point" the text was aligned in the way that it could be read "from below".

In order to mirror vertically aligned text use one of the options *upwards direction* and *running direction*. The upwards direction of is calculated from the base point of a text to the upper end; the running direction in the direction the text is written.

When mirroring a text, usually the upwards vector or the running direction is mirrored. The other direction adjusts in the way, that a readable text is created. In both cases the text position is mirrored normally. When mirroring the upwards position also the right-left-alignment is changed.
When mirroring the running direction the top-bottom-alignment is changed.





In the parameter dialog box ***Referenced Texts / Position Numbers*** the display of referenced texts and position numbers can be saved in different display styles.

Text Styles

Using the menu command ***Settings, Text*** different text styles for dimensions, texts and position numbers can be saved.

The current text style is always suggested when using the command ***Text***, ***Define Text*** as local setting.
When inserting single texts not conform to the current text style, the values can be defined in the option card ***Local settings*** using the command ***Text***, ***Define Text***.

When saving text styles in a templates, all models generated on the basis of this template are available. This is particularly valid for fonts used.

The settings described below can be entered directly in the dialog box or taken over from a text by clicking the button ***Determine from graphics***:

Text parameters
In the ***height, width, distance*** text boxes, the proportions for the characters can be determined in this specified order - since these values are sometimes dependent on each other.
Text is drawn in the size entered independent of the scale of the partial drawing. When using Windows Truetype fonts, their pixel sizes are recalculated to drawing specific units.

height
After you have entered the value for the character height, the values for the character width and spacing as well as the line spacing are also modified, so that the previously defined relation of these values remains the same.

width
After you have entered the value for the character width, the value for the character spacing is also modified, so that the previously defined relation of these values remains the same.

distance
This value defined the distance between two characters. This is evaluated depending on the ***proportional*** setting. When using Truetype fonts, this value is not taken into consideration.

angle
Enter the angle for the base line of the text.

inclination angle

If the text characters are to be written as "normal", i.e. perpendicular to the base line, an angle value of 90 should be entered.

If they are to be italic (inclining to the right), enter a value smaller than 90. With values that are greater than 90 the inclination will be to the left. The maximum permitted value is 135.

With Truetype fonts, the text is italic with angles unequal to 90° - without taking the angle value into account.

line spacing

This value defines the distance between two text lines.

If the ***Vertical*** setting is active for the alignment, it corresponds to the distance of the text columns.

proportional

If this check box is active, the text is written proportional. This means that the value entered in the ***distance*** text box is always the actual distance between two characters.

If the check box is not active, all characters are taken as imaginary rectangles of the same width and the value for the spacing defines the distance between the individual rectangles. When using Truetype fonts, this value is not taken into consideration.

Box addition

The surrounding box for texts, e.g. for Clip areas, for the addition Frame, for check dimensions or for referenced texts can be defined. The proportional value based on the text size can be defined for different text styles using the menu command ***Settings***, ***Text*** in the area ***Text values***, ***Box addition***.

 **Font Selection**

If this command button is selected is active a dialog box will be presented so that you can specify the font. The description used for it will appear in the field ***current font***.

True Type smaller

Activate this option if the height of true type texts should be including the decendes and ascenders, as done in older versions. Deactivate this option to get the correct calculation for the height of texts from the baseline.

Display of texts

In addition to the font parameters the display for the texts can be defined as follows:

addition
A list box can be used to select whether the text is to be drawn ***Without*** addition, with a frame line at the ***Top*** or ***Bottom*** or whether is should be allocated a ***Frame***. When using Truetype fonts, the ***Top*** option is not available.

alignment
If ***Horizontal*** is selected, the characters in a text line are ordered next to one another on the base line, if ***Vertical*** is selected, the characters are ordered under one another.
The direction in which the characters are written always corresponds to the direction of the base line determined in the ***angle*** text box. When using Truetype fonts, the ***Vertical*** option is not available

reference point
This list box can be used to select one of 9 options for positioning reference points.
This always refers to a box containing all the lines relating to the text.

- bottom left, centre, right
- centre left, centre, right
- top left, centre, right

This defined the reference point for positioning a text in the drawing. At the same time, this determines the justification of all lines of a text:

left	text left justified
centre	text centred
right	text right justified

All lines are aligned on an imaginary line which runs perpendicular to the base line of the text.

The reference point of a text can be snapped with the Snap Menu command ***Text Reference*** and using the menu command ***Settings, Point Definition, Text Reference*** in the point filter block.

as block
If this check box is active, instead of the existing text, only the surrounding rectangles are displayed in the drawing and plotted. If a drawing contains a lot of text, the speed of the redraw command can be increased in this way.

clipped
If this check box is active, positioned text can be detached.
This means that objects or parts of objects which project into the box which surrounds the text are not displayed on the screen or when the drawing is output. You cannot detach texts with a pen plotter, because you would then display the detached text with a filled area. For this reason the dialog box offers the button ***Print text with clip face*** for printing: you will be able to print the texts and suppress the detachment.

with color
If the option ***clipped*** is active, you can highlight the text with a color similar to a highlighter by activating the option ***with color*** and selecting a desired color via the color button.

Determine From Drawing
After you have selected this command button, the values for text already in the drawing can be taken over. Identify the text.

Please note:
By using the menu command ***File, Save/Load Parameter Sets***, the attributes for the text can be loaded and taken over from a MPS file.
The list of fonts is not modified by doing this.

Define New Text Style

Different text styles, containing the parameters for common texts, dimensions and position numbers, can be defined as follows:

1. Open the dialog box ***Text*** using the menu command ***Settings, Text***.
2. Choose the option card ***Text Styles*** and change the parameters.
3. Click the command button ***New Style***.
4. Enter a name for the new text style in the text box left of the command button.
5. Accept the changes by clicking the ***Change Style*** command button.

The saved style can be defined as current style for texts, for dimension texts and/or for position numbers.

If the different styles are saved in a template, they can be used in every model based on this template.

Change Text Style

Text styles for common texts, dimensions and position numbers can be changed as follows:

1. Open the dialog box ***Text*** using the menu command ***Settings, Text***.

2. Choose a text style from the list left of the ***New Style*** command button.

3. Change the parameters according to your needs or change the name of the text style.

4. Accept the changes by clicking the ***Change Style*** command button.

Please note:
The text style ***Standard*** cannot be renamed.

Erase Text Styles

Text styles for common texts, dimensions and position numbers can be deleted as follows:

1. Open the dialog box ***Text*** using the menu command ***Settings, Text***.

2. Choose the text style to be deleted from the list left of the ***New Style*** command button.

3. Click the ***Erase Text Style*** command button.

Please note:
The text style ***Standard*** cannot be erased.

Switch Text Style

To switch a text style to a predefined one follow this steps:

1. Open the ***Text*** dialog box using the menu commands ***Settings, Text***.

2. Choose in the upper area of the dialog box the styles for ***Common Texts***, ***Position Numbers*** and ***Dimension Texts***

3. To change the display of all existing position numbers in a drawing choose a style and activate the option ***Change style of position numbers for all existing***.

4. Click the command button Accept to make other changes in the dialog box.

Click the command button ***OK***, to accept the changes and leave the dialog box.

Display of Referenced Texts and Position Numbers

In the parameter dialog box for texts the display of referenced texts and position numbers can be saved in different display styles.

The ***Current Style for Referenced Texts*** and the ***Current Style for Position Numbers*** can be defined in the option card for ***Referenced Texts / Position Numbers***.

Change existing styles or generate new styles as follows:

1. Open the dialog box ***Settings, Text***.

2. Activate the option card ***Referenced Texts / Position Numbers***.

3. Choose one style from the list of predefined styles to be changed or define all parameters to your needs to define a **New Style**.

4. In the dialog area ***Display*** the parameters for referenced texts and / or position numbers can be defined.

Addition

Without

Bottom
On the option card ***Line*** the distance between line and text can be defined.

Rectangle
On the option card ***Rectangle*** a fixed ***Height*** and ***Width*** as well as the a fixed ***Angle*** for the rectangle can be defined.

Fitted Rectangle
The size of the rectangle will automatically fit to the size of the text. With the parameter ***Distance*** the distance between text and rectangle can be defined.

Circle
On the option card ***Circle*** a fixed radius can be defined for a circle surrounding the referenced texts and / or position numbers.

Fitted Circle
The size of the circle will automatically fit to the size of the text. With the parameter ***Distance*** the distance between text and circle can be defined

Reference Line

Choose one of the options, where the reference line should start at the text.

Without

Centre
The reference line starts at the centre of the text..

Bottom Left
The reference line starts at the bottom left of the text.

Bottom Centre
The reference line starts at the bottom centre of the text.

Bottom Right
The reference line starts at the bottom right of the text.

Top Left
The reference line starts at the top left of the text.

Top Centre
The reference line starts at the top centre of the text.

Top Right
The reference line starts at the top right of the text.

Left
The reference line starts centred left of the text.

Right
The reference line starts centred right of the text.

Snap
The reference line is generated as shortest distance between text and reference point.

Reference Symbol

Choose one of the symbols to be displayed at the end of the reference line.

Reference Symbol size

Define here the ***ref. symbol size*** (arrow, circle, line) for referenced texts and position numbers.

5. ***Change Style***
Accept the changes to the parameters by clicking the ***Change Style*** command button.

or

Define New Style
click the ***Define New Style*** command button to save the changes in the parameters as a new style. Enter a name for the new style. Accept the changes by clicking the ***Change Style*** command button.

Please note:

To change the display of all existing position numbers in a drawing choose a style and activate the option ***Change style of position numbers for all existing***. Otherwise the new style is only used for new position numbers.

When changing the style or the parameters for referenced texts, using the option ***Change style for referenced texts for all existing*** the changes can be assigned to all referenced texts in the model.
Please be aware: All referenced texts in the model will then be changed.

The current style can be deleted in the area ***Define / Process Styles*** by clicking the ***Delete Style*** command button.

, or ; ; A referenced text can be defined using the displayed icons ***Dimension***, ***Referenced Text*** or in the dialog box ***Text***, ***Define***.

, The display of single referenced text can be edited by using the command ***Information***, ***Edit Object***.

Parameters for referenced texts can also be saved in the menu command ***File***, ***Read/Write Parameters*** (*.mps) and transferred to other models. Basically all parameters for texts, dimensions, position numbers and referenced texts should be saved in Templates (*.TPL).

Determine font

The font can be determined using the menu command ***Settings***, ***Text***. In this dialog box the text styles for common texts, dimensions (also for surface characteristics, form/position tolerances, texts in welding symbols) and for position numbers can be defined.
Choose one of the styles or define a new style for which a font should be determined.

Click the command button **Font Selection**.The Font Selection dialog box can be used to load fonts which are available for text.

Vector fonts delivered with *BeckerCAD* can be used or the Truetype fonts available in Windows.

Loading of fonts should be carried out in a template. In this way, all models generated on the basis of this template are also available.

1. If you wish to add a vector font to the ***Active fonts*** list, click on the command button.

   If a Windows Truetype font is to be loaded, click on the command button.

   A selection dialog box is displayed from which the required font can be selected. Vector fonts exist as files with the extension FNT.

   Loaded fonts are entered in the ***Active fonts*** list. Only one of them can be selected as the current font.

2. To do so, tag one of the fonts in the list and then click ***As Standard Font***.

3. If the selected font should be allocated to the style click the command button ***Change Style***.

4. Click ***OK*** to close the dialog box.

Please note:

Truetype fonts can only be used in the 2D drawing window. If only their contours are needed, they can be transformed to the 3D window.

You can also delete any font from the list: tag it and then click .

Formatting of Texts

Special formattings in texts as **bold**, *italic*, underlined, ~~strikethrough,~~ special signs or cyrillic characters can be defined by **XML-Tags** in the text dialog window.
The special formattings by XML-Tags can only be used with true type fonts. The option **XML** in the dialog window ***Define Text*** or ***Edit Text*** must be activated. If generally one would like to work with XML formatted texts, the option **XML** should be activated in ***Settings***, ***Text***.

A special formatting is adjusted to a text in most cases by an opening and a closing tag surrounding the text. Opening tags have the format ***<tagname>***, in the closing tags a front slash is in the front of the tagname ***</tagname>***.

Example:

The following word is printed bold , the next word <i>italic</i>.

is shown as:

The following word is printed **bold**, the next word *italic*.

There are some tags which don´t need an opening tag. These tags are formatted **<*tagname*/>**, so the front slash is added behind the tagname. These tags are the abbreviated forms for **<*tagname*></*tagname*>.**

Example:

The line break tag,
starts a new line.
is shown as:
The line break tag,
starts a new line.

This example also shows, that in XML formatted texts is defined exclusively with this tag and not, like in „normal" texts by entering the *Return* key.

Please note: Opposite to HTML, in XML and XHTML the upper / lower case is important, which means
 and
 are different tags. Therefore all tags in *BeckerCAD* are written in lower case.

Nesting of tags

The nesting of texts is possible. Opposite to HTML, XML tags are not allowed to ‚overlap'.

Example:

correct XML:

This text is bold <i>bold+italic italic</i>.
This text is **bold *bold+italic*** *italic.*

Wrong XML:

This text is bold <i>bold+italic italic</i>.

Attributes in tags

Opening tags and standalone tags can contain attributes, describing further formattings. Attributes are formatted attributename="*value*" and are listed behind the tagname seperated by blanks.

Example:

Text in font Arial in the size 9,5.
is shown as:
Text in font Arial in the size 9,5.

Further **examples** for XML formatted texts can be found here.

Overview of the defined Tags

Formatting of texts

Font and text parameter	 ...

Attribute:
Font: **face="***fontname***"**
Height: **size="***textheight***"** or **size="***percent***%"**
Width: **width="***textwidth***"** or **width="***percent***%"**

Bold	 ...
Italic	<i> ... </i>
Underlined	<u> ... </u>
Overlined	<o> ... </o>
Strikethrough	<s> ... </s>
Bigger	<big> ... </big>
Smaller	...
Superscript	^{...}
Subscript	_…

Structuring of texts

Fraction	<fraction> … … </fraction> *)

Attribute:
align="center" | "left" | "right"
default: "center"

delimiter="line" | "slash"
default: "line"

Over each other	<over> … <br/ … <br/ …. </over> *)

Attribute:
align="center" | "left" | "right"
default: "left"

Line break	 *)

Attribute:

align="center" | "left" | "right"

Horizontal line	*)

Attribute:

align="center" | "left" | "right"
width="Width**"** or **width="**percent**%"**
size="thickness**"**

*) the following alternative is possible:

<fraction><row>...</row><row>...</row></fraction>
<over><row>...</row><row>...</row>…</over>
in this case **</br>** and **</hr>** are not necessary, the attributes **align**, **width** and **size** would be implemented in the tag **row**.
Advantage: the relation between attribute and text is clearer
Disadvantage: more complex to write

Further **examples** for XML formatted texts can be found here.

Unicode Signs/ ISO 10646

Texts in BeckerCAD are stored and displayed as 8 bit signs in the current sign coding. In most cases this is the west european sign code "ISO Latin 1" (ISO 8859-1), which contains approximately 224 signs. Most of the true type fonts supports much more signs, for example to display cyrillic, greak and chinese characters. BeckerCAD supports the two notations for Unicodes defined in XML / HTML:

- **Numeric Notation**
- **Named entities**

Precondition for the displaying of the characters is an installed and selected true type font which contains these signs.

Numeric Notation

Any Unicode sign can be imported into the text by the numeric notation **&#*number*;** (decimal) or **&#x*number*;** (hexadecimal).

Example:

π or **π** is shown as **π**

The string **太及拳** is shown as **太及拳** (tai ji chuan)

Named entities

Besides the numeric notation there are ‚named entities', which can be imported into the text in the format **&*name*;** . The greak character **α** is represented by **α**, the copyright sign © by **©**.

The names in the following tables are predefined in BeckerCAD. Please consider the syntax, shown in the following example:

Sign	Description	Name	Unicode
§	section sign	sect	#167

To display the paragraph sign in *BeckerCAD* , select a true type font and activate the option ***XML*** in the dialog window ***Define Text***. The entry in the dialog must be:

§ or **§**

Further examples for XML formatted texts can be found here.

XML-own Signs

Sign	Description	Name	Unicode
″	quotation mark	quot	#34
;	ampersand sign	amp	#38
<	less-than sign	lt	#60
>	greater-than sign	gt	#62

Character Set ISO 8859-1

Sign	Description	Name	Unicode
	no-break space	nbsp	#160
¡	inverted exclamation mark	iexcl	#161
¢	cent sign	cent	#162
£	pound sterling sign	pound	#163
¤	general currency sign	curren	#164

¥	yen sign	yen	#165
¦	broken vertical bar	brvbar	#166
§	section sign	sect	#167
¨	dieresis	uml	#168
©	copyright sign	copy	#169
ª	ordinal indicator, feminine	ordf	#170
«	angle quotation mark left	laquo	#171
¬	not sign	not	#172
	soft hyphen	shy	#173
®	registered sign	reg	#174
¯	macron	macr	#175
°	degree sign	deg	#176
±	plus-or-minus sign	plusmn	#177
^{2}	superscript two	sup2	#178
^{3}	superscript three	sup3	#179
´	acute accent	acute	#180
µ	micro sign	micro	#181
¶	paragraph sign	para	#182
·	middle dot	middot	#183
¸	cedilla	cedil	#184
^{1}	superscript one	sup1	#185
º	ordinal indicator, masculine	ordm	#186
»	angle quotation mark right	raquo	#187
¼	fraction one-quarter	frac14	#188
½	fraction one-half	frac12	#189
¾	fraction three-quarters	frac34	#190
¿	inverted question mark	iquest	#191
À	capital A, grave accent	Agrave	#192
Á	capital A, acute accent	Aacute	#193

Â	capital A, circumflex accent	Acirc	#194
Ã	capital A, tilde	Atilde	#195
Ä	capital A,dieresis	Auml	#196
Å	capital A, ring	Aring	#197
Æ	capital AE diphthong	AElig	#198
Ç	capital C, cedilla	Ccedil	#199
È	capital E, grave accent	Egrave	#200
É	capital E, acute accent	Eacute	#201
Ê	capital E, circumflex accent	Ecirc	#202
Ë	capital E, dieresis	Euml	#203
Ì	capital I, grave accent	Igrave	#204
Í	capital I, acute accent	Iacute	#205
Î	capital I, circumflex accent	Icirc	#206
Ï	capital I, dieresis	Iuml	#207
Đ	capital Eth, icelandic	ETH	#208
Ñ	capital N, tilde	Ntilde	#209
Ò	capital O, grave accent	Ograve	#210
Ó	capital O, acute accent	Oacute	#211
Ô	capital O, circumflex accent	Ocirc	#212
Õ	capital O, tilde	Otilde	#213
Ö	capital O, dieresis	Ouml	#214
×	multiply sign	times	#215
Ø	capital O, slash	Oslash	#216
Ù	capital U, grave accent	Ugrave	#217
Ú	capital U, acute accent	Uacute	#218
Û	capital U, circumflex accent	Ucirc	#219
Ü	capital U, dieresis	Uuml	#220
Ý	capital Y, acute accent	Yacute	#221
Þ	capital THORN, islandic	THORN	#222

ß	small sharp s, german	szlig	#223
à	a, grave accent	agrave	#224
á	a, acute accent	aacute	#225
â	a, circumflex accent	acirc	#226
ã	a, tilde	atilde	#227
ä	a, dieresis	auml	#228
å	a, ring	aring	#229
æ	ae, diphthong	aelig	#230
ç	c, cedilla	ccedil	#231
è	e, grave accent	egrave	#232
é	e, acute accent	eacute	#233
ê	e, circumflex accent	ecirc	#234
ë	e, dieresis	euml	#235
ì	i, grave accent	igrave	#236
í	i, acute accent	iacute	#237
î	i, circumflex accent	icirc	#238
ï	i, dieresis	iuml	#239
ð	eth, islandic	eth	#240
ñ	n, tilde	ntilde	#241
ò	o, grave accent	ograve	#242
ó	o, acute accent	oacute	#243
ô	o, circumflex accent	ocirc	#244
õ	o, tilde	otilde	#245
ö	o, dieresis	ouml	#246
÷	divide sign	divide	#247
ø	o, slash	oslash	#248
ù	u, grave accent	ugrave	#249
ú	u, acute accent	uacute	#250
û	u, circumflex accent	ucirc	#251

ü	u, dieresis	uuml	#252
ý	y, acute accent	yacute	#253
þ	thorn, islandic	thorn	#254
ÿ	y, dieresis	yuml	#255

Greak Characters

Sign	Description	Name	Unicode
Α	alpha upper	Alpha	#913
α	alpha lower	alpha	#945
Β	beta upper	Beta	#914
β	beta lower	beta	#946
Γ	gamma upper	Gamma	#915
γ	gamma lower	gamma	#947
Δ	delta upper	Delta	#916
δ	delta lower	delta	#948
Ε	epsilon upper	Epsilon	#917
ε	epsilon lower	epsilon	#949
Ζ	zeta upper	Zeta	#918
ζ	zeta lower	zeta	#950
Η	eta upper	Eta	#919
η	eta lower	eta	#951
Θ	theta upper	Theta	#920
θ	theta lower	theta	#952
Ι	Iita upper	Iota	#921
ι	iota lower	iota	#953
Κ	kappa upper	Kappa	#922
κ	kappa lower	kappa	#954
Λ	lambda upper	Lambda	#923
λ	lambda lower	lambda	#955

Μ	mu upper	Mu	#924
μ	mu lower	mu	#956
Ν	nu upper	Nu	#925
ν	nu lower	nu	#957
Ξ	xi upper	Xi	#926
ξ	xi lower	xi	#958
Ο	omikron upper	Omicron	#927
ο	omikron lower	omicron	#959
Π	pi upper	Pi	#928
π	pi lower	pi	#960
Ρ	rho upper	Rho	#929
ρ	rho lower	rho	#961
Σ	sigma upper	Sigma	#931
ς	end sigma	sigmaf	#962
σ	sigma lower	sigma	#963
Τ	tau upper	Tau	#932
τ	tau lower	tau	#964
Υ	ypsilon upper	Upsilon	#933
υ	ypsilon lower	upsilon	#965
Φ	phi upper	Phi	#934
φ	phi lower	phi	#966
Χ	chi upper	Chi	#935
χ	chi lower	chi	#967
Ψ	psi upper	Psi	#936
ψ	psi lower	psi	#968
Ω	omega upper	Omega	#937
ω	omega lower	omega	#969
ϑ	theta symbol	thetasym	#977
ϒ	ypsilon with hook symbol	upsih	#978

ϖ	pi symbol	piv	#982

Named Entities for mathematical symbols

Sign	Description	Name	Unicode
∀	for all	forall	#8704
∂	partial	part	#8706
∃	there exists	exist	#8707
∅	empty set	empty	#8709
∇	nabla	nabla	#8711
∈	element of	isin	#8712
∉	not an element of	notin	#8713
∋	contains as member	ni	#8715
∏	product	prod	#8719
∑	sumation	sum	#8721
−	minus	minus	#8722
∗	asterisk	lowast	#8727
√	square root	radic	#8730
∝	proportional to	prop	#8733
∞	infinity	infin	#8734
∠	angle	ang	#8736
∧	logical and	and	#8743
∨	logical or	or	#8744
∩	intersection	cap	#8745
∪	union	cup	#8746
∫	integral	int	#8747
∴	therefore	there4	#8756
∼	similar to	sim	#8764
≅	approximately equal to	cong	#8773

≈	almost equal to	asymp	#8776
≠	not equal to	ne	#8800
≡	identical to	equiv	#8801
≤	less than	le	#8804
≥	greater than	ge	#8805
⊂	subset of	sub	#8834
⊃	superset of	sup	#8835
⊄	not a subset of	nsub	#8836
⊆	subset of or equal to	sube	#8838
⊇	superset of or equal to	supe	#8839
⊕	circled plus	oplus	#8853
⊗	circled times	otimes	#8855
⊥	up tack	perp	#8869
⋅	dot operator	sdot	#8901
◊	lozenge	loz	#9674

Named Entities for Arrow Symbols

Sign	Description	Name	Unicode
←	leftwards arrow	larr	#8592
↑	upwards arrow	uarr	#8593
→	rightwards arrow	rarr	#8594
↓	downwards arrow	darr	#8595
↔	left right arrow	harr	#8596
↵	downwards arrow with corner leftwards	crarr	#8629
⇐	leftwards double arrow	lArr	#8656
⇑	upwards double arrow	uArr	#8657
⇒	rightwards double arrow	rArr	#8658
⇓	downwards double arrow	dArr	#8659
⇔	left right double arrow	hArr	#8660

Named Entities for diverse Symbols

Sign	Description	Name	Unicode
•	bullet	bull	#8226
′	prime	prime	#8242
‾	overline	oline	#8254
⁄	fraction slash	frasl	#8260
℘	script capital P	weierp	#8472
ℑ	blackletter capital I	image	#8465
ℜ	blackletter capital R	real	#8476
™	trade mark sign	trade	#8482
€	euro sign	euro	#8364
ℵ	alef symbol	alefsym	#8501
♠	black spade suit	spades	#9824
♣	black club suit	clubs	#9827
♥	black heart suit	hearts	#9829
♦	black diamond suit	diams	#9830

Examples for XML formatted Texts

Precondition for the displaying of the characters is an installed and selected true type font which contains these signs.The option **XML** must be activated in the dialog window ***Define Text***.
Here are some examples for the entries in the dialog window and the resulting display in the drawing:

σ=sigma α=alpha → σ = sigma α = alpha

bold and <i>italic</i> text → **bold** and *italic* text

H₂O and H₂NO₃ → H_2O und H_2NO_3

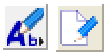
The floor space is 164.0 m² → The floor space is `164.0 m²`

(<fraction>1<hr/>n+1</fraction>) + (<fraction>2<hr/>n</fraction>) → $\boxed{(\frac{1}{n+1}) + (\frac{2}{n})}$

A fraction <fraction>1<hr/>a + b</fraction> → A fraction $\frac{1}{a+b}$

^{Above} and _{Below} is possible on top of each other: <over>Above
Below</over> → Above and Below is possible on top of each other: Above
Below

Define text

 This command can also be activated using this icon.

This command is for inserting a single-line or multiple-line text in the active partial drawing in the 2D drawing that has been selected. Use the following method:

1. In the ***Define text*** dialog box that appears, change to the ***Text entry*** option card and enter the required text. If the text is to be multi-lined, end each line of text by pressing the Enter key.

   Specify whether or not you want to ***fit*** the text between two points, or whether the attribute for the text is to be ***clipped***, or is to have a different ***angle*** for the text base line. This can be specified with the option ***along a line*** using a line already in the drawing.
   When the ***circular*** option is active, the text will not run along a straight base line; it will follow the line on a circle (imaginary). If the angle size has been set to 0 in this case, the text will run outside the circle, and if it is 180, it will be inside the circle (only possible with BeckerCAD fonts).

2. If the settings for font, font size, alignment and position of the reference point which have been determined in ***Settings, Text***, ***Current style for dimensions*** are not to be valid for this text, select the ***Local settings*** option card.
   Select another text style or determine the settings which are to be valid for the text to be entered.
   Then return to the ***Text entry*** option card.

3. Select the ***Insert*** command button in order to write the text in the drawing area.
   If the text base line is to run ***along a line***, identify a line.

4. In the case of a linear text, specify the text reference point. The position of this reference point is always relative to the box surrounding the text. This will also apply if the text is made up of several lines.

   In the case of a ***circular*** text first specify the centre and the radius of the circle, and then the angle for the position of the reference point.

If the active setting is ***fitted***, you can use the cursor to adjust the extent of the text.

Dialog box - Define Text

The option cards within this dialog box can be used to determine the content of one line or multi-line text, set the parameters and position in the drawing.

When starting the command the parameters defined in ***Settings, Text*** are loaded with the settings for font, size, alignment and position of the reference point. This means that the local settings only need to be modified if settings deviate from the general settings for individual text.

Select the respective option card depending on the required options:

Option card - Text entry

Text entry
This text box can be used to enter the text content. If the text is to be multi-lined, end each line of text by pressing the Enter key.

angle
Enter the angle for the base line of the text. If text is arranged in a circle, the value specifies the position inside or outside the line forming the circle.

along a line
This setting is used to transfer the angle for the linear texts base line from one of the lines already in the drawing.
Identify the a line in the drawing area, before you position the text. The text will then be placed in such a way that it can be read from below or from the right.

clipped
If this check box is active, the text will be allocated ***clipped*** as its attribute.
This means that objects or parts of "normal" line objects which project into the box which surrounds the text are not displayed on the screen or when the drawing is output.

with color
If the option ***clipped*** is active, you can highlight the text with a color similar to a highlighter by activating the option ***with color*** and selecting a desired color via the color button.

fitted
This setting can be used so that the cursor can be used to lengthen or to shorten the extent of a text positioned to run along a straight line or

along the line formed by a circle after a reference point has been specified. During this procedure the width and spacing of the characters will be adjusted, but the height will remain unchanged.

circular
This setting is only for the vector fonts. If it is active, the text will be placed around an (imaginary) circle.
If the entry in the *angle* field is 0, the text will be positioned outside the line forming the circle, and inside if the angle is 180.

XML
The setting is only available for true type texts. Activate this option, if you wish to define special formattings like **bold**, *italic*, underlined, ~~strikethrough,~~ special signs or cyrillic characters by **XML tags**.
Detailed informations can be found in the chapter ***Special Formatting of Texts***.

only generate contour lines
If a TrueType font is selected, this option can be activated. If this is active, no text is generated but 2D line objects (lines and splines) which form the boundaries of the individual characters.
The contour lines of a text, which can be positioned in the 2D as well as the 3D window, are saved in a group with the description *TrueTypeContour* which is generated under the current group.

Text reference
With this command, text that has already been positioned can be allocated a reference to any geometry or a point.

The display of the referenced text is depending on the style defined in the option card ***Referenced Text/Position Number*** using the menu command ***Settings***, ***Text***.

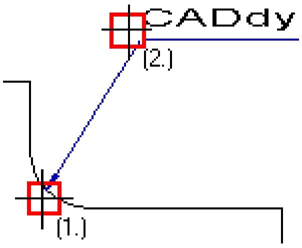
1. Activate the command.

2. Determine the starting point of the reference line using the cursor or the Snap Menu options.

3. Identify the text to be referenced. (2.)
A text reference can also be alligned to multiple texts.
If, for example, a complete text block, which was defined by multiple texts in the drawing, should get one frame and one reference line, then all texts should be selected to align the reference with.

4. A reference line is generated between the defined point and the text. The connection to the text is depending on the settings in the

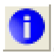
Current style for referenced texts.



5. The allocation of the reference line to layers is defined in the optioin card ***Referenced Text/Position Number*** using the menu command ***Settings***, ***Text***. The size of the reference symbol is depending on the current dimension settings.

Please note:

The same command can also be activated using the ***Dimension*** menu in the toolbox icon strip. The properties of a text reference align themselves to the current dimensioning parameters (colour, line type and width, delimiter size). Modifications to these parameters. Can be assigned to all referenced texts in the model using the option ***Change style for referenced text for all existing***.

,  The display of single, already existing text references can be edited using the command ***Information, Edit Object***. After starting this command identify a referenced text and change the desired parameters in the ***2D Object Editor*** dialog.

The ***Dimension / Disassemble Dimension*** command can be used to disassemble a text reference into its individual objects. The associativity to the reference object and to the text will be lost in doing so.

Option card - Local settings

In the ***height, width, distance*** text boxes, the proportions for the characters can be determined in this specified order - since these values are sometimes dependent on each other.
Text is drawn in the size entered independent of the scale of the drawing area.

height
After you have entered the value for the character height, the values for the character width and spacing as well as the line spacing are also modified, so that the previously defined relation of these values remains the same.

width
After you have entered the value for the character width, the value for

the character spacing is also modified, so that the previously defined relation of these values remains the same.

distance
This value defined the distance between two characters. This is evaluated depending on the ***proportional*** setting. When using Truetype fonts, this value is not taken into consideration.

proportional
If this check box is active, the text is written proportional. This means that the value entered in the ***distance*** text box is always the actual distance between two characters.
If the check box is not active, all characters are taken as imaginary rectangles of the same width and the value for the spacing defines the distance between the individual rectangles. When using Truetype fonts, this value is not taken into consideration.

Box addition
The surrounding box for texts, e.g. for Clip areas, for the addition Frame, for check dimensions or for referenced texts can be defined. The proportional value based on the text size can be defined for different text styles using the menu command ***Settings***, ***Text*** in the area ***Text values***, ***Box addition***.

line spacing
This value defines the distance between two text lines.
If the ***Vertical*** setting is active for the alignment, it corresponds to the distance of the text columns.

inclination angle
If the text characters are to be written as "normal", i.e. perpendicular to the base line, an angle value of 90 should be entered. With any value greater than 90 they will be inclined to the left. The maximum permitted value is 135.
If they are to be italic (inclining to the right), enter a value unequal to 90. With Truetype fonts, the text is italic with angles smaller than 90° - without taking the angle value into account.

addition
A list box can be used to select whether the text is to be drawn ***Without*** addition, with a frame line at the ***Top*** or ***Bottom*** or whether is should be allocated a ***Frame***. When using Truetype fonts, the ***Top*** option is not available.

alignment
If ***Horizontal*** is selected, the characters in a text line are ordered next to one another on the base line, if ***Vertical*** is selected, the characters

are ordered under one another.
The direction in which the characters are written always corresponds to the direction of the base line determined in the ***angle*** text box. When using Truetype fonts, the ***Vertical*** option is not available.

reference point
This list box can be used to select one of 9 options for positioning reference points.
This is with reference to a box enclosing all the lines forming a text.

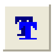
- bottom left, centre, right
- centre left, centre, right
- top left, centre, right

This defines the reference point for positioning a text in the drawing. At the same time, this determines the justification of all lines of a text:

left	text left justified
centre	text centred
right	text right justified

All lines are aligned on an imaginary line which runs perpendicular to the base line of the text.

The reference point of a text can be snapped with the Snap Menu command ***Text Reference*** and using the menu command ***Settings, Point Definition, Text Reference*** in the ***Point Filter*** block.



Font Selection
After you have selected this command button, the *Font Selection* dialog box appears in which a different standard font can be selected for the text.

True Type smaller
Activate this option if the height of true type texts should be including the decendes and ascenders, as done in older versions.
Deactivate this option to get the correct calculation for the height of texts from the baseline.

Interpret related to paper size

This option determines whether text is to be adapted to the current scale or not. If the option is activated, the text scale is generated independently in constant size. This means that the text display is identical for all scales.

If the button is deactivated, text is dealt with the same as all other geometry and adapts itself to the current scale. A text in a scale of 1:1

will be displayed respectively smaller than an identical text in the scale 10:1.

If for example, text is to be used in symbols and adapted to the current scale respective tot he symbol geometry, deactivate the button.

Text style
Select a text style previously defined in ***Settings***, ***Text*** to accept the text parameters defined in this style.

Layer
To define the layer, in which the next text should be placed, select the layer by clicking the button .

Please note:

The text attribute ***Interpret related to paper size*** can also be modified for existing text. For this, select the ***Text***, ***Edit Text*** command and modify the current setting on the ***Local settings*** option card. Alternatively, the attribute can be modified using the ***Information***, ***Any Element*** command.

Edit text

  After clicking on these icons, the ***Text, Edit*** command is started.

This concerns the text content as well as the parameters.

1. **Selection - Action**
   This is for taking over a text (or a number of text) into the selection list prior to activating the command. There can also be other objects apart from text.
   With this procedure you can only process text that has been selected.

   Action – Selection
   After calling up this command, you can accept the text that are to be processed into the action list.

   If the contents of a text are to be modified in some way, this must be the only text that is selected.
   When a number of texts are selected, the text parameters relating to all these texts can be altered. If a text is entered into the text entry field, which in this case is empty, they all have the same contents.

2. After you have selected the command, the ***Edit Text*** dialog box is displayed.

If only one text has been selected in order to edit the content, edit the text in the ***Text entry*** option card. If the text is to be multi-lined, end each line by pressing the Enter key.
If one or more text has been selected, the same parameters can be determined for all the text either on this option card and the ***Local settings*** option card (size, alignment, reference point position, etc.).

If you have selected a single text and want to ***move*** or ***fit*** it, or want to alternate between ***linear/circular*** orientation, click the ***Execute*** command button afterwards.
Then move the cursor into the drawing area to specify the change to the text. On completion of the change you will exit the command.

3. If none of the above options has been activated, the chosen texts will be changed in the drawing, if you click ***Accept*** to use the alteration.
If you are working with the *Action – Selection* procedure, you can deal with other texts by first accepting the changes to the texts selected so far by clicking ***OK***.

Dialog box - Edit Text

The option cards within this dialog box can be used to edit the content of one line or multi-line text and reset their parameters.

The parameter values of the selected text are always displayed in the text boxes. If a text box is empty or it contains a "?", no text has been selected or a number of text has been selected whose parameter values are not the same.
If a value is entered in one of these text boxes, this value is then allocated to all selected text.

<u>Option card - Text</u>

text entry
The contents of the text can be changed in this box.

angle
The entry specifies the angle for the text base line: the text reference point will be retained.

X

Determines the X-coordinate of the text reference point.

Y

Determines the Y-coordinate of the text reference point.

clipped
After you have activated this option the texts that have been selected will be allocated ***clipped*** as their attribute.

with color
If the option ***clipped*** is active, you can highlight the text with a color similar to a highlighter by activating the option ***with color*** and selecting a desired color via the color button.

XML
The setting is only available for true type texts. Activate this option, if you wish to define special formattings like **bold**, *italic*, <u>underlined</u>, ~~strikethrough,~~ special signs or cyrillic characters by **XML tags**.
Detailed informations can be found in the chapter ***Special Formatting of Texts***.

move
After you have selected a single text, you can change it to a different position by clicking ***Execute***.

fit
After you have selected a single text, you can lengthen or shorten its base line after clicking ***Execute***. If this is done, the width and spacing of the letters will be adjusted, but the height will remain unchanged.

linear/circular
If a single text has been selected, you can specify a base line for it as a line or circle after clicking ***Execute***. If the ***Move*** option is not active, the position of the reference point will remain unchanged.

<u>Option card local settings</u>

The boxes on this option card are similar to the those for the command ***Text, Define Text***.

In the ***height***, ***width***, ***distance*** text boxes the proportions for the characters can be specified in the prescribed sequence: the values are in part inter-related. A change to the height will also have a direct effect on the ***line spacing***.

inclination angle
This setting can be used to specify a character inclination that is relative to the text base line.
In the case of Truetype fonts the text will be in italics at any angle other than 90° - whatever the size of angle specified.
proportional
If this option is active, the text will be written proportionally, the value entered in the field ***Distance*** will really be the distance between two characters.

Otherwise the characters will be seen as rectangles with the same width and the value is used for the distance between those rectangles. When using true type texts this setting is unconsidered.

Box addition

The surrounding box for texts, e.g. for Clip areas, for the addition Frame, for check dimensions or for referenced texts can be defined. The proportional value based on the text size can be defined for different text styles using the menu command ***Settings***, ***Text*** in the area ***Text values***, ***Box addition***.

addition

This can be used to specify the frame lines for the texts. Select one of the options from: ***Without, Top***, ***Bottom*** or ***Frame***.
When you are using Truetype fonts, you cannot select the ***Top*** option.

alignment

Select either of the options ***Horizontal*** or ***Vertical***, for arranging the characters relative to the text base line.
When you are using Truetype fonts, you cannot select the ***Vertical*** option.

reference point

This is for choosing a difference reference point and so changing the text justification. The change will result in the same text being moved to a different position.

 **Font selection**

After you have selected this command button, the *Font Selection* dialog box appears in which a different font for the text.

True Type smaller

Activate this option if the height of true type texts should be including the decendes and ascenders, as done in older versions.
Deactivate this option to get the correct calculation for the height of texts from the baseline.

Text size related to paper size

This option determines if the text size is related to the current drawing scale. When the option is active the text size is always the same independent from the scale.
When the option is deactivated texts are handled as all other geometry and will change when the scale is changed. A text in scale 1:1 is displayed smaller then an identical text in scale 10:1.

If, for example, texts are used in symbols and should change their size according to the size of the symbol, this option should be deactivated.

Text style
Select a text style previously defined in ***Settings***, ***Text*** to accept the text parameters defined in this style.

Layer
Using the icon  the layer manager is offered and you can move the text to another layer.

Using special characters with vector fonts

When you are using vector fonts (*.FNT), you can make use of one of the following special characters by typing the underscore character "_" before the actual character:

Entry	Result
_D	∅
_G	°
_O	Ω
_P	Π
_S	Σ
_a	α
_b	β
_d	δ
_e	ε
_f	φ
_m	μ
_n	ν
_o	ω
_p	π
_ß	§
_+	±
_Digit	Superscript digit

Numerate

,  Using the command ***Numerate*** from the Text toolbar, consecutive numbers can be placed in the drawing or in the current workplane (3D window).

The texts are created according to the text parameters defined in ***Settings***, ***Text*** for the Current Text Style. The display properties are assigned to the current layer for ***Texts***.

The default value after the start of the command is 1. This can be edited in the status dialogue window. All further texts are numbered consecutively.

You can use the status dialog window to enter text in two additional input fields, which should appear before or after the number. Confirm the entry with the ***Enter*** key.
You can also use spaces when entering in the status window. It is therefore not possible to deactivate the status window with the space bar key.
The last text entered is retained during the active work session.

General process

1. Define the parameters (font, height, reference point etc.) in ***Settings***, ***Text*** in the ***Current Text Style***.
2. Select ***Numerate*** in the ***Text*** toolbar.
3. Enter the start value in the status dialogue window.
4. Place the texts one after another.

Please note:

Placed numbers can be edited using the command ***Text***, ***Edit***.

Alphanumeric Labeling

,  The command ***Alphanumerate*** from the icon bar ***Text >*** will count up the first small or capital letter in a string.
When starting the command the letter **A** is offered to be placed. When necessary, this letter can be changed in the status dialogue window.

If a string is defined as start text only the first letter will be changed. (A.1; B.1; C.1).

The characters will be created according to the settings made in the menu ***Settings, Text*** for ***Current text style***. The display properties will be according to the current layer for ***Texts***.

You can use the status dialog window to enter text in two additional input fields, which should appear before or after the number. Confirm the entry with the ***Enter*** key.
You can also use spaces when entering in the status window. It is therefore not possible to deactivate the status window with the space

bar key.
The last text entered is retained during the active work session.

General procedure

1. Define the desired parameters (text height, Font, reference point etc.) using the menu command ***Settings, Text*** for the ***Current text style***.
2. Select the command ***Alphanumerate*** from the ***Text*** icon bar.
3. Enter the start string in the status dialogue window.
4. Place the strings displayed in the status dialogue window after each other in the drawing.

Please note:

Already placed strings can be changed using the command ***Text >, Edit***.

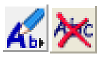
Align text to line

 After clicking on the displayed icons, the ***Text, Align Text to Line*** command is started.

This command can be used to align existing text in the model to line objects.

1. Identify the text to be positioned.
2. Identify an object of type line. Depending on the nearest endpoint of the line, the text is positioned on or under / right or left of the line.
   The text can then be dynamically moved to the line with its text reference point.
3. If necessary, in the Status dialog box, define a distance for the text to the line or press the **Shift key** in order to position the text freely.
4. Define the final position using the cursor or by selecting from the Snap Menu.

Disassemble text

 After clicking on these icons, the command ***Text, Disassemble*** is started.

This command can be used to disassemble existing text into 2D line objects (lines and splines) which form the boundaries of the individual characters.

Selection – action

1. Take over the required text into the selection list and then load the command.

Action - selection

1. Load the command and then select the text.

The contour lines of a text are saved in a group with the description *TrueTypeContour* which is generated under the group in which the text is found.

Please note:

 Texts can also be disassembled using the command ***Process 2D, Disassemble 2D Objects***.

4.11 Dimension

 This command for generating and processing dimensions can also be activated using this icon.

The dimensions in a 2D drawing window will be generated in the partial drawing containing the first object that you identify. If the dimensions are specified not by objects but by points, the dimensions will be in the current partial drawing.

In the 3D-view window the dimensions will be saved in the WP for the 3D window that is currently selected.
If they are drawn in a WP of the 3D window, they are linked with this WP. If the WP is erased, all the dimensions on this WP are also erased.

Each dimension is treated as an **object** that will be generated according to the currently valid ***Settings, Dimension***. The parameters for dimension texts can be defined and saved using the menu command ***Settings***, ***Text***.

Tolerances containing tolerance abbreviations can be computed with the help of the command ***Fit dimensions table*** and then inserted in the drawing as a table.

Dimensions completed for 2D objects that are all **in the same partial drawing**, or those that have been defined by points on these objects, are **object-related**. They will react associatively with the reference objects whenever a position or a size is changed, and they will be erased as soon as any of the dimensioned 2D objects is erased.

Dimensions without relation to 2D objects are **point related.** This also applies to the spacing dimensions whose reference points or reference objects have been defined or identified in **different partial drawings**.

Using the dimension commands line objects in Symbols can be also identified, e.g. to dimension the length of a line, the distance between parallel lines, the angle between non parallel lines or the diameter of arcs and circles.

Dimensions created to lines in symbols in the same partial drawing, or to points of this elements are **object related**. They behave associative when changing the position or size of their related objects and they are deleted when the related symbol will be disassembled, updated, exchanged by another symbol or erased.

In ***Model Views*** and derived ***Section Views*** lines, circles and arcs and distances between objects can be dimensioned. Also dimensions can be generated as distance between points.
If any solid is changed inside the 3D model, the model views and the section views will interact associatively. If the edges and their projection lines are not changed, the program will attempt to retain the dimensions and to adjust them. Dimensions related to points will remain unchanged.
Dimensions on model views which have lost their reference due to changes in the 3D model and therefore are not associative any more, are displayed lowlighted. The dimension is displayed in the original colour again, when the reference to the 3D model is valid again. The ***Lowlightcolour*** for 2D objects can be defined using the menu command ***Settings, Options*** in the tab ***General Settings***.

Apart from the modifications made to an existing dimension by an alteration that could affect the position and/or size of the reference objects, dimensions can be altered in quite specific ways. This can be done with the following commands: ***Dimension, Move Dimension/Dimension Text***, ***Edit Dimension Properties**, **Edit Dimension Text, Define Reference Point, Add/Remove Additives, Insert Partial Dimension***, ***Delete Partial Dimension***.

The generated dimensions are

- saved in the current group determined in group selection
- allocated to the ***Dimension Layer*** defined in ***the Layer Explorer*** inside the ***Model Explorer***
- assigned to the display properties of the ***Dimension Layer***.

The group allocation can later be modified in the ***Model Explorer***.

The display attributes for the dimensions can be changed by using the ***Layer Explorer*** inside the ***Model Explorer*** or by using ***Dimension, Edit Dimension Properties***.

To ensure that whilst processing your drawing, only dimensions can be identified, use ***Settings, Object Filter*** to activate the snap possibilities for dimension and deactivate the other object types.

Please note:

The lines and text in a dimension cannot be dealt with separately. Only endpoints can be snapped on the dimension lines and the dimension help lines.

Determine dimension settings

The dimension settings can be determined using the menu command ***Settings, Dimension***.

This is where it is possible to utilise global settings to modify dimensions that have already been positioned.

Different settings can be saved in the dimension styles. Dimensions are always generated with the settings of the current style. Changes to the settings in a style always affects all dimensions generated with this dimension style.

If the general settings are saved in a template, all models generated on the basis of this template are available.

When **global settings** are changed, the changes will also affect dimensions that have already been completed and that have the attribute ***according to defaults***.
Global settings are indicated by the entry (global).
The attribute ***according to defaults*** can be changed by means of the command ***Dimension, Edit Dimension Properties***.

Please note:

By using the menu command ***File, Save/Load Parameter Sets***, the settings for hatchings can be loaded and taken over from a MPS file.

Dimension Styles

Different parameter settings for dimensions can be saved as ***Dimension Styles***.

New dimensions are generated according to the parameters in the dimension style defined in the menu command ***Settings, Dimension***.

Define New Dimension Style

To define a new dimension style follow this steps:

1. Open the dialog box ***Dimension*** using the menu command ***Settings, Dimension***.
2. Press the command button ***New Style.***
3. Enter a name for the new dimension style in the text box right besides the command button.
4. Change the parameters in the dialog boxes according to your needs.
5. Confirm the changes by clicking the ***Accept*** command button.

Please note:

Existing dimensions can be assigned to a different dimension style using the command ***Dimension, Edit Dimension Properties***.

Change Dimension Style

To change the parameters of a dimension style and with this all dimensions in a model generated with this dimension style choose the following:

1. Open the dialog box ***Dimension*** using the menu command ***Settings, Dimension***.
2. Choose the style from the list to be changed.
3. Define the new parameters for this style in the option card ***Common***.
5. Confirm the changes by clicking the ***Accept*** command button, if other changes should be made or by clicking the ***OK*** command button to accept the changes and leave the dialog box.
6. Other parameters can be changed using the option card ***Dimension geometry***.
7. Confirm the changes by clicking the ***Accept*** command button, if other changes should be made or by clicking the ***OK*** command button to accept the changes and leave the dialog box.

Please note:

Parameters of dimensions which were changed using the command ***Dimension***, ***Edit Dimension*** and deactivating the option ***accorind to defaults*** are not changed.

Erase Dimension Style

Dimension styles can be erased as follows:

1. Open the dialog box ***Dimension*** using the menu command ***Settings, Dimension***.

2. Choose the style from the list to be erased.
3. Click the command button ***Erase Dimension Style***.
4. Choose from the list the style for new dimensions.
5. Click the ***OK*** command button to accept the changes and leave the dialog box.

Please note:

Dimensions which were generated with the erased style before, are not assigned to any style then. Using the menu command ***Dimension***, ***Edit Dimension*** they can be assigned to a different style.

Determine general dimension settings

Dimension form (global)
By selecting the dimensioning type, a number of basic settings are activated for the dimension text and dimension geometry, which can still be modified:

Standard

The unit for the dimension length or distance is mm.

The point is set as the decimal separator.

An open arrow-head is set as the dimension limit.

Construction lines are displayed up to the contour (possibly with a specified distance).

Architecture 0,5/0,7

The unit for the dimension length or distance is cm or m, i.e. al values under 100 are dimensioned in cm, all values over 100 are dimensioned in m. Any values that are less than 1 cm will be inserted as superscripts. Their size will be 0.5 or 0.7 times the size of the dimension text.

For values under one centimetre it is suggested to round off to 0.25 cm.

The comma is set as the decimal separator.

A slash of 60 degrees is set as the dimension limit.

Construction lines are displayed with a fixed length.

Dimension text

Positioning
One of the following methods can be used to position the dimension text:

Freely definable
When this button is active, the cursor will have to be used to position the dimension text after each dimension has been positioned.
Apart from the first part dimension, this setting is not valid for incremental, reference and NC dimensions.

After a dimension has been defined, the position of the dimension text can be changed by means of *Dimension**,** Move Dimension/Dimension Text*.

Centred
When this button is active, the dimension text will be centred on the dimension line.
If the distance separating the dimension boundaries of simple dimensions and of "outer dimensions" in incremental and reference dimensions is less than the length of the dimension text, the text will be positioned outside the extension line.

Superscript
When this button is active, the dimension text will be changed to superscript.
The distance separating the text base line from the dimension line will be arrived at by adding together the following values: ***distance to dimension line*** (***General*** option card) and ***overhang*** (***Dimension Geometry*** option card).

Fitted
When this button is active, the dimension text will be positioned so that it will break up the dimension line.
The value specified as ***distance to dimension line*** will be ignored.

Automatic
When this button is active, the result will be a combination of ***Centred*** and ***Superscript***: If the dimension text does not fit between the dimension boundaries, it will appear as a superscript in the case of "interior dimensions".

distance to dimension line
This entry will specify the distance separating the base line for the dimension text from the dimension line.
If the ***fitted*** button is active, this entry will be ignored.

with fixed angle (global)
If this setting is active, the base line of all dimension text is aligned according to the entered angle value.

Dimension Text Style (global)
A text style for dimension texts can be selected from the list. The text parameters like Font, Height, Width defined in the different styles can be edited using the menu ***Settings, Text***. More information can be found in the chapter ***Settings for texts, dimension texts, referenced text and position numbers***.

Underline edited dimension text
If this option is activated, all edited dimension texts are underlined as long as the edited text is in the text box for dimension numbers.

Check dimension
If this command is activated, the following defined dimensions are identified as check dimensions.

length dimension unit (global)

The value of the dimension number is always computed from the actual lengths and distances in the drawing.

This option will determine whether the length and distance dimensions are to be ***Metric*** or in ***Inch***.

If one or other of the options chosen is ***Architecture 0,5 / 0,7***, the value of the dimension number will be calculated in the following way: With lengths and distances that are less than 100, the value will be in the true number of centimetres. If they are more than 100, the value shown will be the value divided by 100 in metres.
The *inches* option will be ignored.

angle dimension unit (global)
This will determine whether or not text for angle dimensions are to be shown in ***degrees*** with decimal places or in ***degrees, minutes, seconds***.

decimal places (global)
This entry is for specifying the number of decimal places (0-10) for length and angle dimensions for rounding the dimension values.
If the option ***trailing zeros*** is not active, fewer decimal places will be displayed.

If the dimensioning chosen is one of the alternatives ***Architecture 0,5/0,7***, all dimensions in excess of one metre will be displayed with, at the most, two decimal places.

decimal separator (global)
This can be used to determine the character separating a whole number and the decimal places in a dimension value: *a point* or a c*omma*.

trailing zeroes (global)
If this button is active, all the decimal places will always be displayed.
If this button is not active, no zeros will appear at the end of the dimension value.

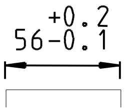
round off architecture (global)
If the dimensioning form is one of the alternatives ***Architecture 0,5/0,7***, all the values less than 1 will be rounded to 0.1, 0.25 or 0.5. They will be part of the dimension value as a superscript.
Text size in this case will be equivalent to 0.5 or 0.7 times the "normal" text sizes, depending on whatever has been specified here as the setting.

<u>Tolerance entries</u>

type

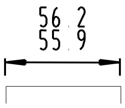
If the dimensions are to include all tolerance values, the types can be selected and the value(s) entered in the appropriate fields.
The size of the tolerance text will correspond to the one for the dimension text multiplied by the entry made in the field for the **text size factor**.

Deviation



The dimension text will have the upper and lower deviations added to it.
Their values can be inserted with the appropriate sign in the two fields ***Top value*** and ***Bottom value***.

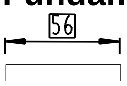
Limit dimension



The dimension text will be replaced by two text indicating the upper and lower limiting dimensions.
These will be computed from the actual value by adding to it the values entered in the two fields ***Top value*** and ***Bottom value***.

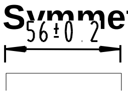
It is important that the values are entered as numbers with a decimal point. Leading zeros need not be included.

Fundamental



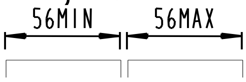
The dimension text will contain no tolerance deviation information.
The text will be enclosed by a rectangular box.

Symmetric deviation



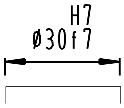
The dimension text will have the deviation placed adjacent with the ± symbol.
The value of the deviation must be entered in the *deviation* field.

Min, Max



The text MIN, MAX will be added to the dimension text.

Fit dimensions



The abbreviations indicating the tolerance classes will be added to the dimension text.
When a table of fit dimensions is to be created and added to the tolerance information for inclusion in the drawing, the command to be used is ***Dimension, Generate Fit dimension table***.

If in the template, it has already been defined that dimensions can be allocated tolerance entries, then you have the opportunity to define the tolerances in the Status dialog box with preset tolerance types whilst the dimension is still positioned on the cursor for dynamic positioning.

Tolerance by value

Tolerances on dimensions are interpret as character string (texts) and displayed in the drawing. Therefore it is warranted that the defined tolerance value is displayed consistently, independent from the current model units.
In the case, tolerances should be recalculated when changing the model units (mm, cm, m, inch), the tolerance types *deviation*, *limit dimension* and *symmetric deviation* can also be handled as numerical values in a dimension.
The display of the tolerance values in this case adjusts to the global or local settings made in the area *Dimension number*.

Tolerances to individual dimensions in dimension strings (NC, incremental, reference dimensions) can be brought in using the ***Dimension /Edit Dimension Text*** command.

Layers

These fields are for displaying the layer(s) activated by means of ***Define, Layers*** have been specified for the dimensions.
It will be from these layers that the partial objects in the subsequent

dimensioning will be allocated their display attributes. This can be changed by using the menu command ***Define, Layers*** or by using the command ***Dimension, Edit Dimension Properties***.

Please note:

Using the ***File***, ***Save/Load Parameter Sets*** menu command, you can load the settings for hatchings from a MPS file and take them over.

Determine dimension geometry settings

Using the menu command ***Settings, Dimension*** settings for the display of dimension geometry can be determined.

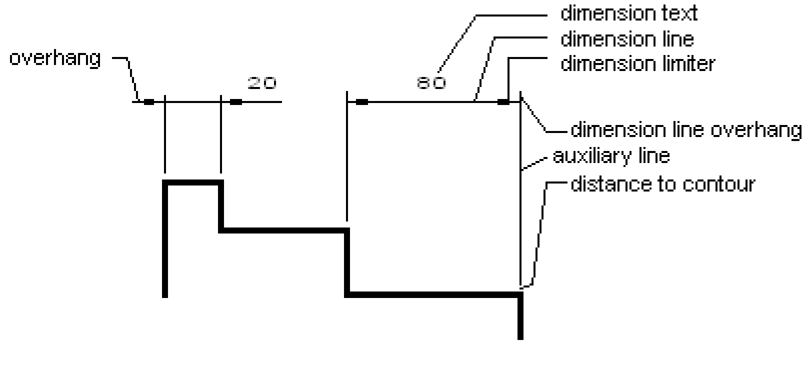
Different settings can be saved in different dimension styles. Dimensions are always generated according to the current style. Changes in the settings will affect all dimensions generated using this style.

In addition, the global settings made for dimensions already in place can be altered.

If these settings are saved in a template, all models generated on the basis of this template are available.

Whenever ***global settings*** are altered, the alterations will apply to existing dimensions as well, provided that their attribute is ***according to default***.
Global settings are indicated by (global).
The attribute according to default can be altered by means of the command ***Dimension, Edit Dimension Properties***.

The sizes that have been entered on the option card will always be displayed in relation to the measurements in the drawing, which means that they are independent of the scale used for a partial drawing.



Dimension lines
The parameters listed below can be used to draw different kinds of line.

overhang
This parameter will only have any effect if the specified ***Dimension limits type*** is an arrow-head. The size of this parameter specifies the length of the dimension line projection beyond its intersection with the extension line.
The dimension line will not be lengthened, unless the dimension text or dimension arrow-head are outside the extension line.

distance with reference dimension
This parameter indicates the constant spacing between each pair of dimension lines that refer to two neighbouring partial dimensions for a reference dimension.
The command ***Dimension, Edit Dimension Properties*** is for changing the spacing for dimensions already specified and/or releasing the spacing so that partial dimensions can be moved to a different position.

Constant distance with reference dimensions
If this option is deactivated, each individual dimension of a reference dimensioning can be freely positioned. The value in the ***distance with reference dimensions*** text box has no affect on the distance between the individual dimensions.
The ***Dimension, Edit Dimension Properties*** command can be used to later modify the distances of existing dimensions and/or clip the distances in order to be able to move individual partial dimensions.

return with radius dimension
This value determines the shortening of the dimension line for radius dimensions within the framework of the command **dimension radius**. The defined return shortens the dimension line starting from the defined symmetry axis/the defined symmetry point. This option should be used if the symmetry axis/symmetry point of the radius lie outside the active drawing area.
With the command **Dimension, Edit Dim Properties**, you can modify this distance for existing dimensions and/or release the distances in order to be able to move the individual dimensions as required.

Radius dimension line up to midpoint
If this command is activated, the dimension line is drawn from the radii dimensions to the midpoint of the radius.

Draw dimension line
If this switch is deactivated, dimension lines are not drawn.

Auxiliary lines
The settings listed here can be selected to change the display of the auxiliary lines.

distance to contour (global)
When this button is not active, the construction lines will start at the 2D objects that are to be dimensioned/at the points specified.
When the button is active, however, the starting point will be at the distance indicated. The **dimension line overhang** value can be entered to specify how far the construction lines are to run beyond the dimension line on the side away from the contour.

draw with fixed length (global)
When the button is active, the extension lines will be drawn the same length as is indicated in this field.
The list **relation to dimension line** can be used to specify the way in which the construction line is to be divided at the dimension line:
With 1:1 the construction line will be drawn half its length on either side of the dimension line.
With 1:2 the part closer to the contour will be twice the length of the section away from the contour.

display options
These options are used with construction lines for specifying ***none***, or one ***only left***, or ***only right*** or ***both***.

Parallel construction lines with arc dimensions
If this option is activated, the dimension construction lines of arc dimensionings are drawn parallel.

Dimension limits
The entries will determine the settings applicable to dimension limiters:

type (global)
The standard limit type is to be chosen from the list of symbols.

When the choice is either an open or a closed arrow-head, the "interior dimensions" in an incremental dimension will automatically be limited by a small circle should there be insufficient space. When a filled arrow-head is chosen, a filled circle will replace it.

size (global)
This entry determines the size of the *dimension limiter* that has been chosen: the length of the shaft of the dimensioning arrow or half the length of the line.
The diameter of the circles or filled circles is equivalent to one third of the value that is entered.

position
This will determine the orientation of the arrow-heads using in dimensions: either the text position itself will be disregarded completely and they will be placed ***inside/outside*** the construction lines, or, with the option ***text fixed***, they will correspond to the dimension text position.

display options
In this option it can be defined if the dimension limits should only be displayed on the left, on the right or both or not.

NC dimensions
Specify the special settings required for NC dimensions:

dimension text parallel to dimension line
If the setting is active, the dimension text for NC dimensions will run parallel to the dimension line. If the setting is not active, they will run perpendicular to it.

text displacement
When the dimension text orientation is perpendicular, the spacing from the dimension line is derived from the total of the construction line overhang beyond dimension line and the value to be entered here for the displacement.

limit type origin
You can specify whether the origin for an NC dimension is to be denoted by a small circle or by a filled circle.

Moved text with lead
If this option is activated, the position of the dimension text to the dimension line is identified by a dimension reference line. The dimension line that also underlines the dimension text, is described as the ***lead***.
NC dimension text is automatically adjusted so that it can no longer conceal itself.

Length of the lead
Using this value, define the overhang length of the lead to the dimension text by a positive value, a negative value shortens the lead. Is the value is equal to 0, the length of the lead corresponds to the dimension text.

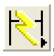
Text distance
The value in this text box defines the distance between the dimension text and the dimension line.

Coordinate Dimension
Define special settings for coordinate dimensions:

With Reference Line
If this option is active, an arrow to the origin will be added to moved coordinates. The arrow will be drawn with the parameters defined in ***Dimension Limits***.

Please note:
By using the menu command ***File, Save/Load Parameter Sets***, the settings for hatchings can be loaded and taken over from a MPS file.

Orthogonal multi-dimensioning

,  This command can be used to create different types of dimensioning.

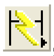
The dimensions listed below will be computed, in each case, orthogonal, i.e. horizontally or vertically. They can be specified as simple dimensions, incremental dimensions or as reference dimensions:

Length of lines
Distance separating points
Distance separating parallel lines

The dimensions listed below will be computed as simple dimensions without alignment.

Diameter of circles
Radius of arcs
Angle between two lines

NC dimensioning

,  This command can be used to create various types of dimensioning.

The dimensions listed here will be computed orthogonally in each case, i.e. horizontally or vertically. They can be defined as a simple dimension or as an NC dimension:

Length of lines
Distance separating points
Distance separating parallel lines

The dimensions listed below will be computed as simple dimensions without alignment.

Diameter of circles
Radius of arcs
Angle between two lines

Aligned multi-dimensioning

, This command can be used to create different types of dimensioning.

The dimensions listed below will be oriented, in each case, to the line or points that have been defined. They can be specified as simple dimensions, incremental dimensions or as reference dimensions:

Length of lines
Distance separating points
Distance separating parallel lines

The dimensions listed below will be computed as simple dimensions without alignment.

Diameter of circles
Radius of arcs
Angle between two lines

Dimensioning distances and lengths

For dimensioning the distances between points and the lengths of lines the commands to be used are ***Dimension, Multidimension Orthogonal*** and ***Multidimension Aligned***.

, The ***Multidimension Orthogonal*** icon can be used for dimensioning distances separating points or lines either vertically or horizontally.
The dimension line will be oriented to run either vertically or horizontally.

, The command ***NC Dimension*** can be used in the same way as those for ***Multidimension Orthogonal***. If the dimension includes more than one partial dimension, the NC dimension will be created in such a way that the dimension line runs either horizontally or vertically.

, The ***Multidimension Aligned*** icon is for dimensioning the actual distances between points or the lengths of lines.
The dimension line will be oriented to run along an imaginary connection between the points or along the line.

The commands can be used to create simple dimensions that can even be expanded directly by defining additional distances into incremental, reference or NC dimensions.
The orientation of the remaining partial dimensions will be determined by that of the dimension first defined.

The procedure to be followed is more or less identical for the two commands:

- Definition of the first dimension by specifying
  – length of a line
  – distance separating two points
  – distance separating two parallel lines
- For simple dimensions only:
  Cancel by clicking right mouse button once
- For expanding the simple dimension to form a incremental, reference or NC dimension:
  Define the next partial dimension by specifying another point or by identifying another parallel line or by identifying a line running perpendicular to the dimension line.
  When a reference dimension is generated, the dimension lines are generated with the distance defined in the current dimension style using the menu command ***Settings***, ***Dimension***.

The special procedures described in the following sections only differ in the method used for defining the first (partial) dimension.

Please note:

If a distance dimension or a length dimension is used to specify the side view of a circle, you can use the ***Dimension, Add/Remove Additives*** command to insert a diameter or radius symbol before the dimension numerals.

Whenever an incremental, a reference, NC or coordinate dimension is being created, it must be remembered that whatever is drawn will be managed as an individual object.

When any of the partial dimensions is to be deleted or one is to be added, the commands to be used are ***Dimension, Insert Partial Dimension*** and ***Remove Partial Dimension***.

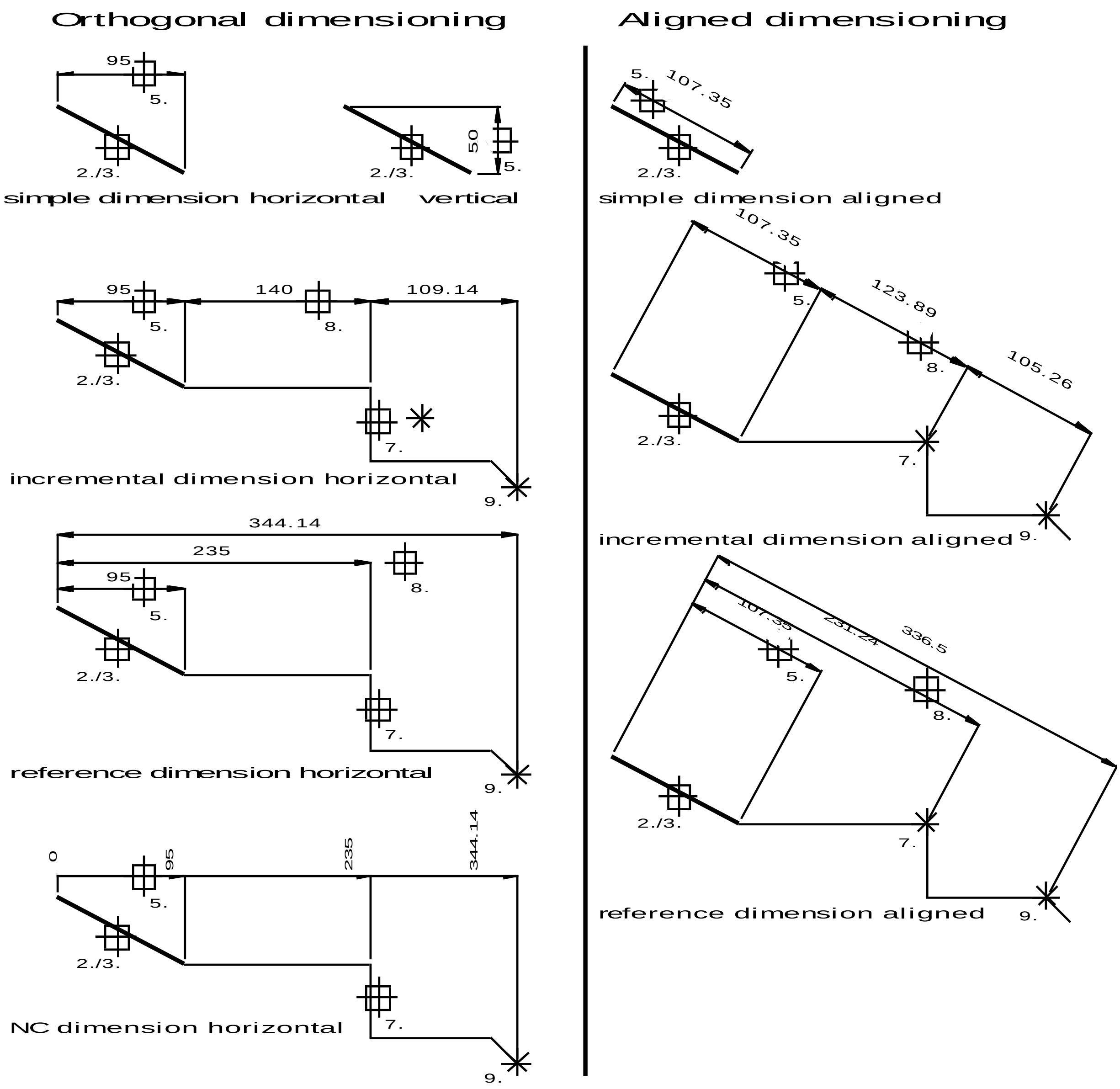
Dimensioning the length of lines

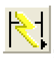
The procedure described below can be used for dimensioning the length of lines (including partial lines in polylines, rectangles and polygons).

The dimension is generated in the partial drawing which contains the line object identified first.

Depending on the command chosen, it is possible to dimension horizontally, vertically or along a line.

If required, the simple dimension can be enhanced for incremental, reference or NC dimensions.



1. ,  Activate the command ***Dimension, Multidimension Orthogonal***, if the horizontal or vertical spacing between the endpoints of a line are to be dimensioned and, possibly, to be expanded to a reference dimension or an incremental dimension.

 Activate the command ***Dimension, NC Dimension***, if the horizontal or vertical spacing of the endpoints of a line are to be dimensioned and, possibly, be extended to an NC dimension.

 Activate the command ***Dimension, Multidimension Aligned***, if the actual length of the line is to be dimensioned and, possibly, expanded to a reference dimension or to an incremental dimension.

2. If the 2D drawing window is active if necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be considered during the dimensioning procedure.
3. Identify the line.
4. Identify the line again. The dimension is generated according to the current style defined in ***Settings***, ***Dimension***.
5. Only ***Multidimension Orthogonal*** or ***NC Dimension***:
   Specify how the dimension line is to be oriented:
   The horizontal dimension is completed by moving the cursor upwards or downwards away form an imaginary rectangle that contains the diagonal forming the line.
   A vertical dimension is completed by moving the cursor to the right or the left.
6. Position the dimension line by clicking the left mouse button or by entering a value against ***Distance*** in the dialog box.

   The dialog box fields can be used to alter the ***Distance*** from the dimensioned object, the ***Text*** distance from the dimension line, the ***Tolerances*** and the ***Dimension Number***.
7. If for the current dimension style the option card ***General*** shows that the ***Settings, Dimension*** contains the active button ***dimension text positioning freely definable***, the dimension text can be dragged along the dimension line and then positioned (which is not meaningful when further partial dimensions are defined).
   If the text is positioned outside the extension lines, the dimension line will be lengthened accordingly on the same side as the text position. On the other side the line will be lengthened by the amount indicated as the ***overhang*** on the ***Dimension geometry*** option card.
8. If you want to create a simple dimension, click the right mouse button once. The next simple dimension can then be defined.

The command is completely ended by clicking the right mouse button again.

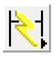
If the simple dimension is to be expanded to an incremental, a reference or NC dimension, instead of following the above procedure you must specify the next distance by defining a point. Alternatively, identify a line that runs at a right angle to the one identified first.

9. Only ***Multidimension***:
Specify whether you want to complete an incremental dimension or a reference dimension by positioning the cursor and then click the left mouse button:
If the cursor is already positioned within the dimension text or on an imaginary extension, an incremental dimension will be completed; in any other position the result will be a reference dimension.

10. If required, you can return to Step 7 to complete another partial dimension; otherwise, click the right mouse button.
Activate the command by clicking the right mouse button again to end the command completely.

Please note:

In the case of reference dimensions or NC dimensions the origin will be on the starting point for the line identified first. Depending on the direction in which you draw, you may also have to change the position by using ***Dimension, Define Reference Point***.
If you do not want to do this, complete the first of the partial dimensions by means of points or parallel lines.

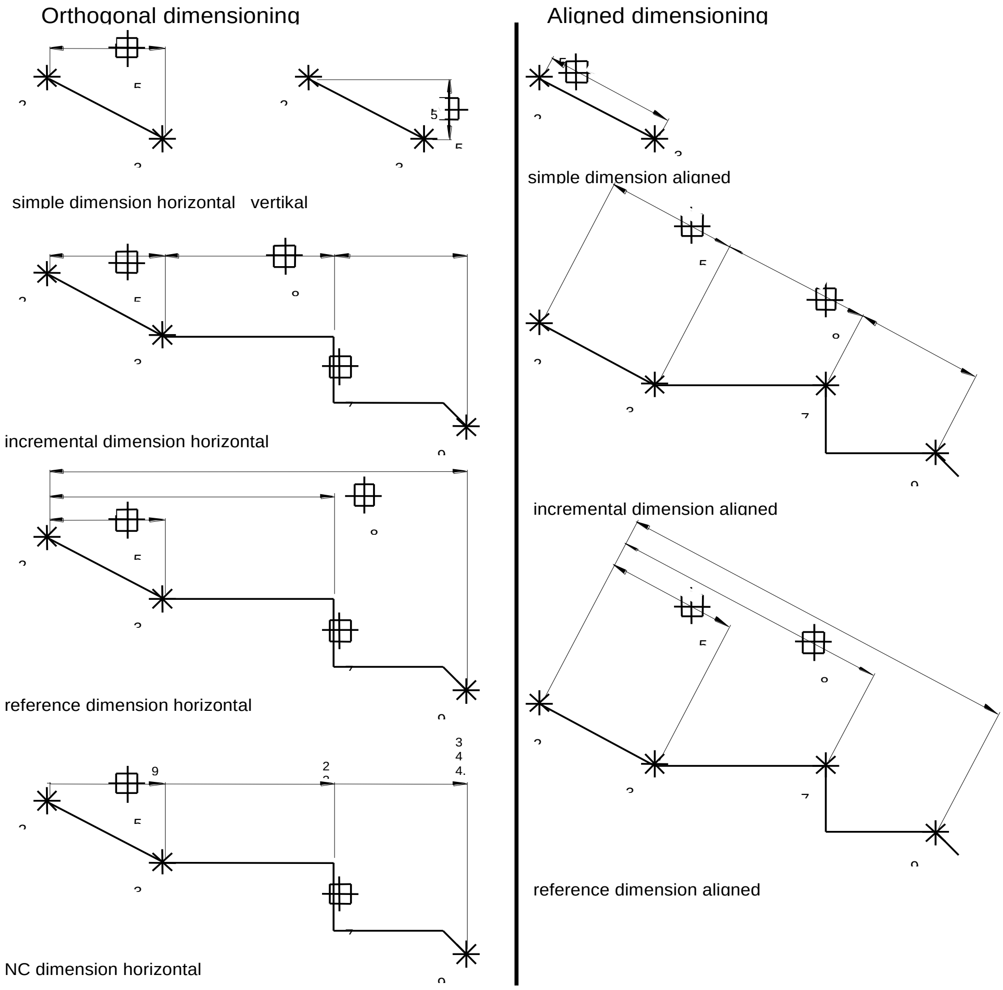
,  For the single dimensions ***Length of Lines*** and ***Distance of Points*** an inclination can be defined afterwards to the dimension auxiliary lines by using the command ***Dimension***, ***Edit Dimension Properties***

Dimensioning the distance between points

The procedure described immediately below can be used to dimension the distance between any two points.
The dimension will be created in the current partial drawing.

Depending on the command the dimension can be completed horizontally, vertically or can be placed along an imaginary connection between the points.

If required, the simple dimension can be extended to an incremental, reference or NC dimension.



1. , Use the command ***Dimension, Orthogonal Multi Dimension*** to dimension the horizontal or vertical distance of points and if applicable extend it to a string or reference dimension.

   Use the command ***Dimension, NC Dimension*** to generate a NC dimension.

   Activate the command ***Dimension, Multidimension Aligned***, if the actual length of the line is to be dimensioned and, possibly, expanded to a reference dimension or to an incremental dimension.

2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selection.

3. Define the first point.
   When the dimension is expanded to a reference dimension or to an NC dimension, it defines the reference point or the origin.

4. Define the second point.
   The dimension is generated according to the current style defined in ***Settings***, ***Dimension***.
5. Only ***Multidimension Orthogonal*** or ***NC Dimension***:
   Specify how the dimension line is to be oriented:
   The horizontal dimension is completed by moving the cursor upwards or downwards away form an imaginary rectangle that contains the diagonal forming the line.
   A vertical dimension is completed by moving the cursor to the right or the left.
6. Position the dimension line by clicking the left mouse button or by entering a value against ***Distance*** in the dialog box.

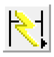
   The dialog box fields can be used to alter the ***Distance*** between dimensioned points, the ***Text*** distance from the dimension line, the ***Tolerances*** and the ***Dimension Number***.
7. If for the current dimension style the option card ***General*** shows that the ***Settings, Dimension*** contains the active button ***dimension text positioning freely definable***, the dimension text can be dragged along the dimension line and then positioned (but not worthwhile when further partial dimensions are defined).
   If the text is positioned outside the extension lines, the dimension line will be lengthened accordingly on the same side as the text position. On the other side the line will be lengthened by the amount indicated as the ***overhang*** on the ***Dimension geometry*** option card.
8. If you want to create a simple dimension, click the right mouse button once. The next simple dimension can then be defined.
   The command is completely ended by clicking the right mouse button again.

   If the simple dimension is to be expanded to an incremental, a reference or NC dimension, instead of following the above procedure you must specify the next distance by defining a point. Alternatively, identify a line that runs at a right angle to the one identified first.
9. Only ***Multidimension***:
   Specify whether you want to complete an incremental dimension or a reference dimension by positioning the cursor and then clicking the left mouse button:
   If the cursor is already positioned within the dimension text or on an imaginary extension, an incremental dimension will be

completed; in any other position the result will be a reference dimension.

10. If required, you can return to Step 7 to complete another partial dimension; otherwise, click the right mouse button.

Please note:

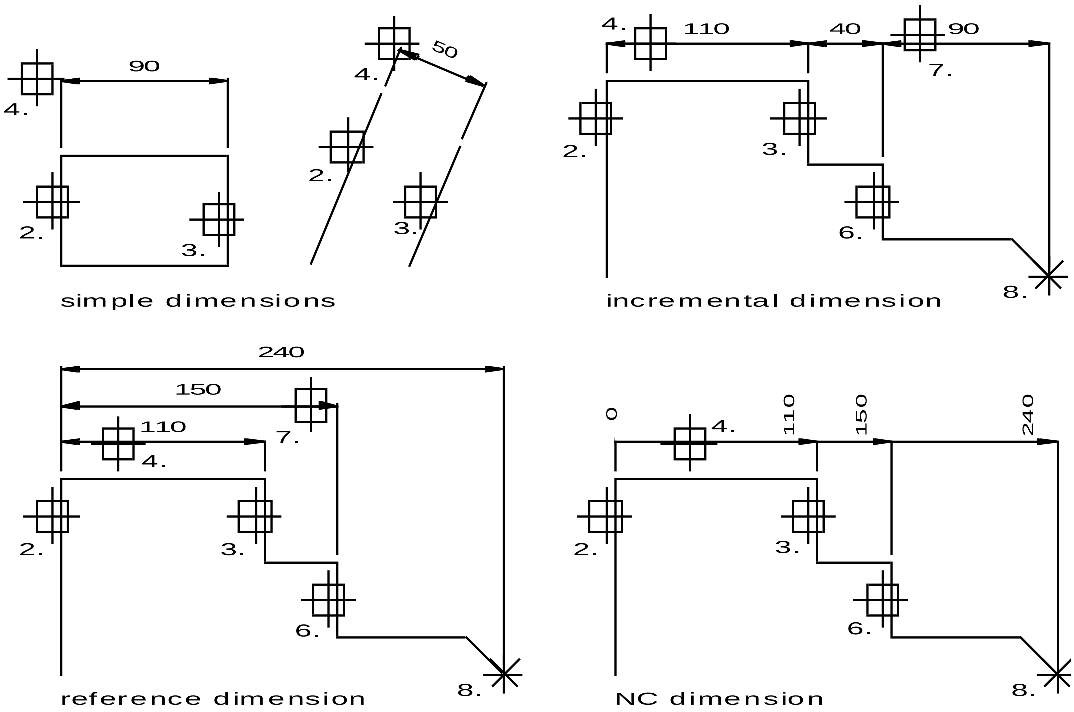
,  For the single dimensions ***Length of Lines*** and ***Distance of Points*** an inclination can be defined afterwards to the dimension auxiliary lines by using the command ***Dimension***, ***Edit Dimension Properties***

Dimensioning the distance between parallel lines

The procedure described immediately below can be used to dimension the distance between parallel lines (including partial lines in polylines, rectangles and polygons).
The dimension will be completed in the partial drawing containing the linear object that was identified first.

The dimension will be oriented to run perpendicular to the parallel lines.

If required, the simple dimension can be extended to an incremental dimension or to a reference dimension by defining additional points.



The dimensions listed here will be computed orthogonally in each case, i.e. horizontally or vertically. They can be defined as a simple dimension, incremental or a reference dimension:

1. ,  Activate the command ***Dimension, Multidimension Orthogonal*** or ***Align Multi Dimension*** if the spacing between the endpoints of a line are to be dimensioned and, possibly, to be expanded to a reference dimension or an incremental

    Do not call up the command ***Dimension, NC Dimension***, unless the lines are to run horizontally or vertically. The command can be used to dimension the horizontal or vertical spacing between the parallel lines and expand it to an NC dimension.

2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selection.

3. Identify the first line.
   When the dimension is expanded to a reference dimension or to an NC dimension, the endpoint determines the position the reference point or the origin.

4. Identify a line running parallel to the first line.
   The dimension is generated according to the current style defined in ***Settings***, ***Dimension***.

5. Position the dimension line by clicking the left mouse button or by entering a value against ***Distance*** in the dialog box.

   The dialog box fields can be used to alter the ***Distance*** between the objects that are dimensioned, the ***Text*** distance from the dimension line, the ***Tolerances*** and the ***Dimension Number***.

6. If for the current dimension style the option card ***General*** shows that the ***Settings, Dimension*** contains the active button ***dimension text positioning freely definable***, the dimension text can be dragged along the dimension line and then positioned (which is not meaningful with further partial dimensions).
   If the text is positioned outside the extension lines, the dimension line will be lengthened accordingly on the same side as the text position. On the other side the line will be lengthened by the amount indicated as the ***overhang*** on the ***Dimension geometry*** option card.

7. If you want to create a simple dimension, click the right mouse button once. The next simple dimension can then be defined.
   The command is completely ended by clicking the right mouse button again.

   If the simple dimension is to be expanded to an incremental dimension, a reference dimension or NC dimension, instead of following the above procedure you must specify the next distance by identifying another parallel line or by defining a point

8. Only ***Multidimension***:
Specify whether you want to create an incremental dimension or a reference dimension by positioning the cursor and then clicking the left mouse button:
If the cursor is already positioned within the dimension text or on an imaginary extension, an incremental dimension will be created; in any other position the result will be a reference dimension.

9. If required, define another partial dimension, or click the right mouse button.
Activate the command by clicking the right mouse button again to end the command completely.

Dimensioning the diameter of circles

The procedures described below are for dimensioning the diameter of circles.
The dimension will be completed in the partial drawing that contains the circle.

Alternatively to the herein described diameters and radii can also be dimensioned using the command ***Dimension, Diameter***. The description can be found in the chapter ***Dimension Diameter***.

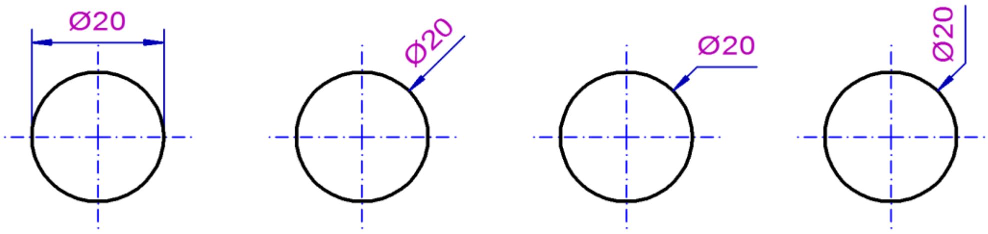
1. , , , Activate the command by clicking the icons ***Dimension, Multidimension Orthogonal*** or the icons ***Dimension, Multidimension Aligned*** or call up the command ***Dimension, NC Dimension***.

2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selection.

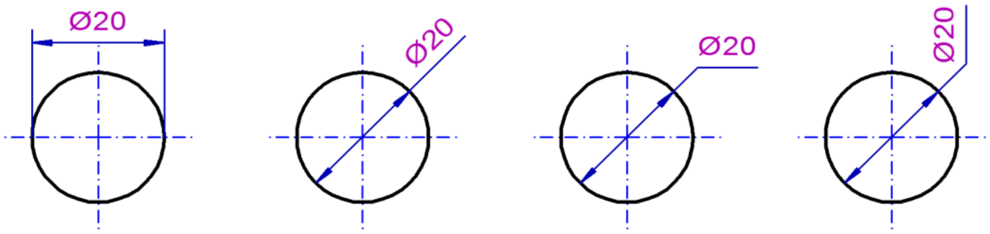
3. Identify the circumference.

4. Identify the circumference a second time.
The dimension is generated according to the current style defined in ***Settings***, ***Dimension***.

5. With the **Y** key you can toggle through the following options of the diameter dimensioning, if the option ***radius dimension line up to centre*** is deactivated in the ***Settings***:



If the option ***radius dimension line up to centre*** is activated, you can toggle between following options in the diameter dimensioning:



Please note:

In the options with the kinked dimension lines keep the ***Ctrl key*** pressed to fix the dimension line and position the dimension text on a free position.

Using the menu command ***Settings, Dimension*** the global setting to display the arc dimension line can be defined in the ***Dimension Geometry*** tab.

You can step between the different dimensioning options for radii by clicking the **X** key.

6. Use the cursor to specify the position of the dimension line, or make an entry against ***Distance*** in the dialog box.

If the cursor is inside the circle, the result will be an inner dimension; if the cursor is outside, the result will be an external dimension.

The orientation of the dimension line can be altered by rotating it about the centre point of the circle at 15° intervals. By pressing the shift key at the same time you can change the orientation yourself.

The dialog box fields can be used to alter the ***Distance*** to the dimensioned object, the ***Text*** distance from the dimension line, the ***Tolerances*** and the ***Dimension Number***.

7. If for the current dimension style the option card *General* shows that the ***Settings, Dimension*** contains the active button ***dimension text positioning freely definable***, the dimension text can be dragged along the dimension line and then positioned.
If the text is positioned outside the extension lines, the dimension line will be lengthened accordingly on the same side as the text

position. On the other side the line will be lengthened by the amount indicated as the ***overhang*** on the ***Dimension geometry*** option card.

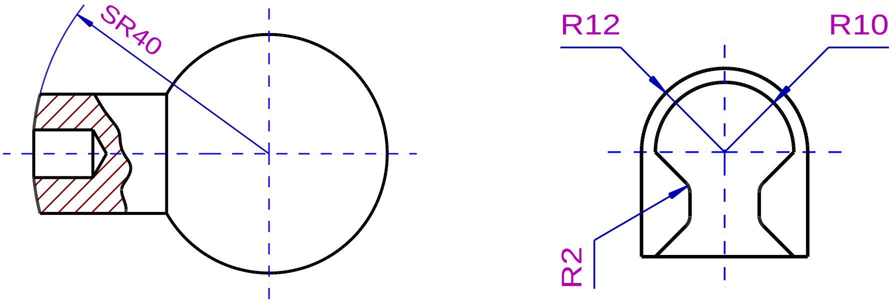
Please note:

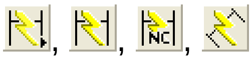
If you want to dimension the lines denoting a side view of circles by the addition of a diameter symbol, you must first dimension the length of the lines.
Then load the ***Dimension, Add/Remove Additives*** command and then select the ***Prefix*** (∅) in the dialog box to be used with the dimension numbers. Finally identify the dimension by individual identification or by identifying the dimension by dragging a rectangular window..

Dimensioning the radius of arcs

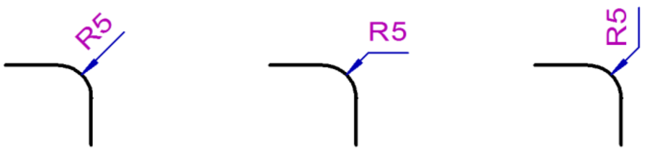
The procedures described below are for dimensioning the radius of circles.
The dimension will be completed in the partial drawing containing the arc.

If necessary, before dimensioning the radii, define in the general dimensioning parameters under ***Settings, Dimensioning*** for the current dimension style whether the radius dimension lines are to be drawn up to the midpoint.
Later drawing of the dimension lines up to the midpoint can be defined using the ***Dimension, Edit Dimension Properties*** command.



1.  Activate the command by clicking the icons ***Dimension, Multidimension Orthogonal*** or the icons ***Dimension, Multidimension Aligned*** or call up the command ***Dimension, NC Dimension***.

2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for specifying the dimension.
3. Identify the circumference.
4. Identify the circumference a second time.
The dimension is generated according to the current style defined in ***Settings***, ***Dimension***.
5. You can step between the different dimensioning options for radii by clicking the **X** key.:



Please note:

When placing with horizontal or vertical dimension line the dimension limit can be fixed by using the ***Ctrl key*** and then move the dimension text to the desired position.

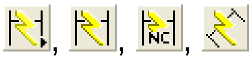
By clicking the **Y** key you can switch to the options for the diameter dimensioning.

6. Position the dimension line by clicking the left mouse button.
The orientation can be altered by rotating about the centre point of the circle at 15° intervals. By pressing the shift key at the same time you can change the orientation yourself.

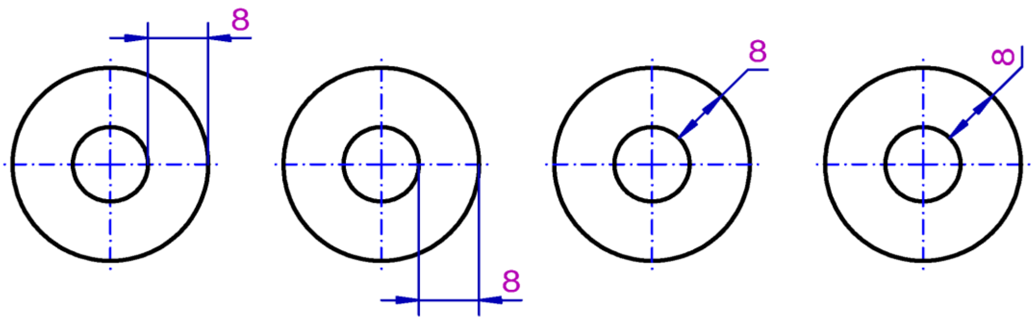
   The dialog box fields can be used to alter the *distance* between the objects that are dimensioned, the ***Text*** distance from the dimension line, the ***Tolerances*** and the ***Dimension Number***.
7. If for the current dimension style the option card ***General*** shows that the ***Settings, Dimension*** contains the active button ***dimension text positioning freely definable***, the dimension text can be dragged along the dimension line and then positioned.

Dimension distance of concentric Circles / Arcs

You can dimension the distance between concentric circles and arcs using the steps described below. Concentric circles are called when several circles have the same center, but have different radii.
The measure is generated in the partial drawing, in which the first identified circle is located.

1.  Start the command ***Dimension***, ***Orthogonal Multi Dimension***, the command ***Dimension***, ***NC Dimension*** or the command ***Dimension***, ***Aligned Multi Dimension***.
2. If the 2D drawing window is active, you can use the ***Select on ... sketchboard*** function at the top of the window to determine which partial drawings are to be taken into account when defining the dimension.
3. Identify the first circle / arc.
4. Identify the second circle / arc.

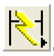
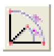
   The dimension is created according to the current dimension style defined in ***Settings***, ***Dimension***.
5. By clicking the ***X*** key you can switch between the following options of the dimension:



6. Define the position of the dimension line by cursor or by entering a value in the filed ***distance*** of the status dialog window.

   The alignment of the dimension line is set in steps of 15 °. If the shift key is pressed, the orientation can be freely defined.
   In the fields of the status window, you can change the text distance to the dimension line, the tolerances, or the dimension text, if necessary.
7. If the default dimension text position is freely definable for the current dimension style on the ***General*** tab of ***Settings***, ***Dimension***, you can now dynamically move the dimension text along the dimension line to the desired position.
   If the text is placed outside the extension lines, the dimension line on the side of the text position is extended by the necessary amount. On the other hand, the dimension line is extended by the value that is preset as the length overlap on the ***Dimension Geometry*** tab.

Angle Chain Dimension

,  The command ***Dimension***, ***Angle Chain*** is activated after these symbols (above) have been selected.

You can create angle dimensions as a single, continuous chain or reference dimension with this command. For chain and reference angle dimensions, each angle dimension is created as an independent individual measurement.

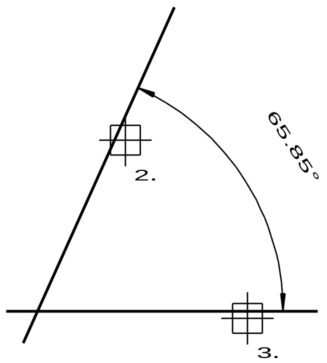
First you must identify a line object in the command. In this way, each of the angles will be defined using other non-parallel line objects. However, you must first identify a point corresponding to the procedure for the definition of an Angle Using 3 Points (apex – first arm – second arm).

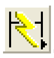
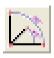
The definition of the different angle dimensions is described on the next pages.

Dimensioning angles between two lines

The procedures described below are for dimensioning the angle formed by two lines that are not parallel. The dimension is generated in the partial drawing which contains the line that was firstly identified.

If desired, the single angle dimension can be extended to a string or reference dimension by identifying objects with the same apex. Each angle dimension is created as an independent individual measurement.



1. ,  Activate the command ***Dimension, Angle Chain***.

2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for specifying the dimension.

3. Identify the first line.

4. Identify the second line, not parallel to the first line.
The angle will be calculated anti-clockwise from the first to the

second line, and the dimension is generated according to the current style defined in ***Settings***, ***Dimension***.

5. Move the cursor, whilst the dimension is dynamically on the cursor into the quadrant the dimension should be positioned.
Using the hotkey X it can be switched between the inside and the outside angle.

6. Use the cursor to specify the position of the dimension line, or make an entry against ***Distance*** in the dialog box.

The dialog box fields can be used to alter the ***Distance*** from apex, ***Text*** distance from dimension line, the ***Tolerances*** and the ***Dimension Number***.

Note:

If the option card ***General*** shows that the ***Settings, Dimension*** contains the active button ***dimension text positioning freely definable***, the dimension text can be dragged along the dimension line and then positioned.

7. If only the define single dimension should be generated press the right mouse button. Another single dimension can be generated afterwards. Another right mouse button click quits the command.

Should the single dimension be extended to a string or reference dimension the next angle will be defined by identifying another line with the same apex.

8. Only ***Angle Chain***
The cursor position and clicking the left mouse button defines wether a string or reference dimension is generated:
If the cursor is upon the dimension text or the same circle a string dimension is generated otherwise a reference dimension. The first identified line is the reference for the next string dimension and for all reference dimensions.

9. Generate an additional angle dimension by identifying another line with the same apex or press the right mouse button. Another right mouse button click quits the command.

Notes:

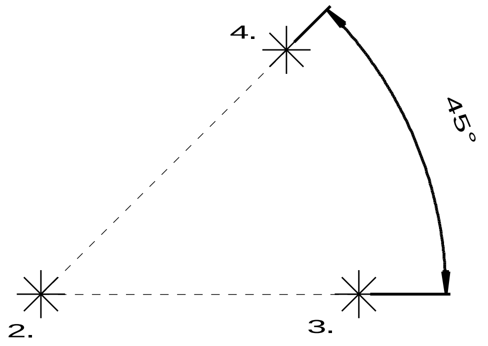
, , ,  The angle between two lines can also be dimensioned using the commands ***Dimension, Orthogonal Multi Dimension, NC Dimension*** or ***Align Multi Dimension***.

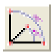
In string and reference angle dimensions every angle dimension is a single dimension and can be edited or deleted as such.

Dimensioning angles – 3 points

The procedures described below are for dimensioning the angle between 3 points.
The dimension will be completed in the current partial drawing.

If desired, the single angle dimension can be extended to a string or reference dimension by identifying objects with the same apex. Each angle dimension is created as an independent individual measurement.



1. ,  Activate the command ***Dimension, Angle Chain***.
2. If the 2D drawing window is active, use the ***Access to … Partial Drawing*** at the top of the window frame, if necessary, to determine which partial drawings are to be taken into consideration during the definition of the dimensions.
3. Activate a snap option using the point definition menu (middle mouse button) or by shortcut.
4. Identify a line object and define the apex.
5. Specify the endpoint of the first (imaginary) arm.
6. Specify the endpoint of the other (imaginary) arm of the angle that is to be dimensioned.
   The angle will be calculated anti-clockwise from the first to the second line, and the dimension is generated according to the current style defined in ***Settings***, ***Dimension***.
7. Move the cursor, whilst the dimension is dynamically on the cursor into the quadrant the dimension should be positioned.
   Using the hotkey X it can be switched between the inside and the outside angle.
8. Use the cursor to specify the position of the dimension line, or make an entry against ***Distance*** in the dialog box.

The dialog box fields can be used to alter the ***Distance*** from apex, ***Text*** distance from dimension line, the ***Tolerances*** and the ***Dimension Number***.

Please note:

If for the current dimension style the option card ***General*** shows that the ***Settings, Dimension*** contains the active button ***dimension text positioning freely definable***, the dimension text can be dragged along the dimension line and then positioned.

9. If only the define single dimension should be generated press the right mouse button. Another single dimension can be generated afterwards. Another right mouse button click quits the command.

Should the single dimension be extended to a string or reference dimension the next angle will be defined by identifying another line with the same apex.

10. Only ***Angle Chain***
The cursor position and clicking the left mouse button defines wether a string or reference dimension is generated:
If the cursor is upon the dimension text or the same circle a string dimension is generated otherwise a reference dimension. The first identified line is the reference for the next string dimension and for all refernce dimensions.

11. Generate an additional angle dimension by identifying another line with the same apex or press the right mouse button. Another right mouse button click quits the command.

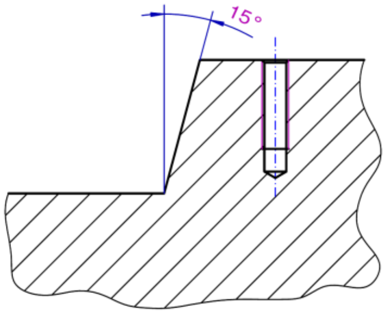
Please note:

The dimension that has been completed is object related, if the endpoints of the arms have been snapped on 2D line objects: if this is not the case, the dimension is point related.

In string and reference angle dimensions every angle dimension is a single dimension and can be edited or deleted as such.

Angle Dimension to Coordinate System

,  Using the command ***Angle to coordinate system*** the angle between two points or a line and an imaginary horizontal or vertical coordinate axes can be dimensioned.



Angle by 2 points

1. Define the apex by identify a point (=middle mouse button).
2. Define a second point for the first side.
3. The second side is defined by an imagenary horizontal or vertical coordinate axis. Place the dimension.

Angle by line

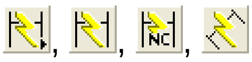
1. Identify the endpoint of a line. (endpoint of the line = apex; line = first side)
2. The second side is defined by an imagenary horizontal or vertical coordinate axis. Place the dimension.

Please note:

By pressing the ***X*** key it can be switched between the inner and outer angle before the dimension is placed.

Dimension threads

To dimension a thread generated with the commands from ***Draw 2D*** follow these steps:

1.  Load the command ***Dimension, Orthogonal Multi Dimensioning,*** the command ***Dimension, NC Dimensioning*** or the command ***Dimension, Aligned Multi Dimensioning***.
2. If the 2D drawing window is active, use the ***Access to ... partial drawing*** commands at the top of the window frame if necessary to determine which partial drawings are to be taken into consideration during the definition of the dimension.
3. Identify the thread arc in the top view or one of the thread lines in the side view.
4. In the top view, identify the thread arc again, in the side view the parallel thread lines.

4.1. For tapped holes in the top view: confirm the ***x*** or ***X*** key before positioning. This key command can be used to change between the radius and diameter dimensioning.

5. Position the dimension.

6. Select the command ***Dimension, Edit Dimension Properties***.

7. Identify the dimension.

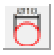
8. If necessary, select M as the ***Prefix*** or enter another name.

9. Confirm with ***OK***.

Please note:

If the dimension text is not positioned correctly, this can be changed using the commands ***Dimension, Move Dimension / Dimension Text***.

Dimensioning diameters

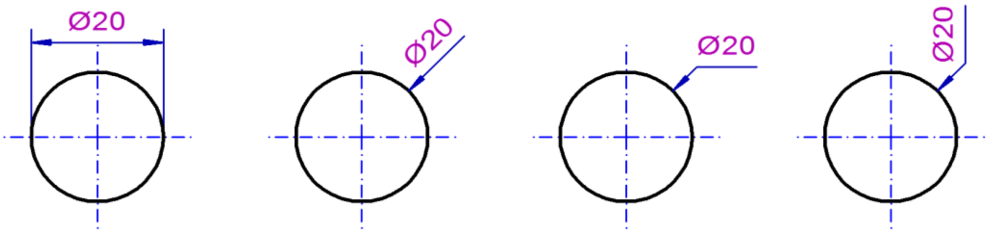
,  By clicking the displayed icons the command ***Dimension, Diameter Dimension*** is started.

Using this command diameter dimensions can be generated on points, lines, parallel line objects, circles and circular arcs.

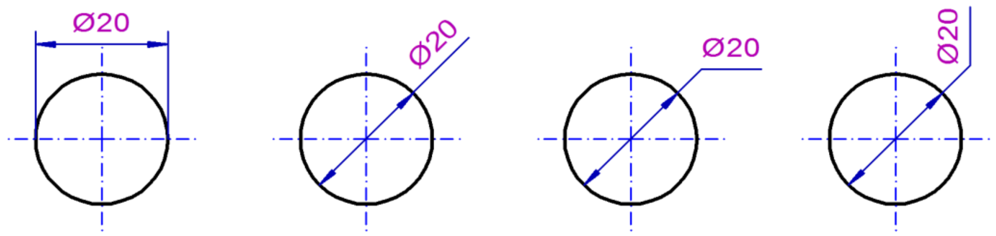
1. If the 2D drawing window is active, use the ***Access to ... partial drawing*** commands at the top of the window frame if necessary to determine which partial drawings are to be taken into consideration during the definition of the dimension.

2. Identify a circle, an arc, 2 parallel line objects, or identify a line twice (also possible in model views). Also, point snap is supported to dimension a point-to-point or object-to-point diameter.
The dimension is created according to the current dimension style defined in ***Settings***, ***Dimension***.

3. When dimensioning circles and arcs, press the ***Y*** key to toggle through the following options of the diameter dimensioning, if the option ***radius dimension line up to centre*** is deactivated in the ***Settings:***



If the option ***radius dimension line up to centre*** is activated, you can toggle between following options in the diameter dimensioning:



Please note:

In the options with the kinked dimension lines keep the ***Ctrl key*** pressed to fix the dimension line and position the dimension text on a free position.

You can step between the different dimensioning options for radii by clicking the **X** key.

4. Position the dimension line by clicking the left mouse button or by entering a value against ***Distance*** in the dialog box.
As long as the cursor is inside the circle or arc an inner dimension is created, if the cursor is moved outside the circle or arc an outside dimension in generated.
The drawing grid can be turned off by pressing the Shift key.

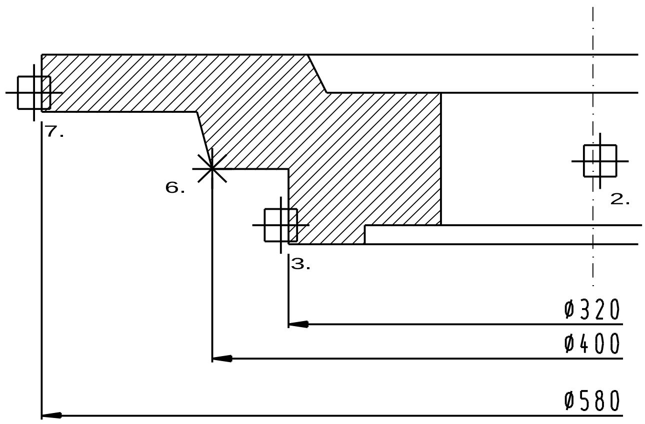
In the text boxes of the dialog box the parameters ***Distance***, ***Text Distance***, ***Tolerance*** or ***Dimension Number*** can be changed.

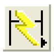
5. If for the current dimension style the option card ***General*** shows that the ***Settings, Dimension*** contains the active button ***dimension text positioning freely definable***, the dimension text can be dragged along the dimension line and then positioned.
If the text is positioned outside the extension lines, the dimension line will be lengthened accordingly on the same side as the text position. On the other side the line will be lengthened by the amount indicated as the ***overhang*** on the ***Dimension geometry*** option card.
With the pressed ***Shift key*** the dimension text can be positioned freely.

Dimensioning half section

The procedures described below are for dimensioning the half sections: Starting at the reference line the command will determine the spacing between parallel lines or points. The dimension text contains the value of the double spacing and the diameter symbol as the prefix.

The dimension will be completed in the partial drawing containing the reference line.



1. ,  Activate the command by clicking the icons ***Dimension, Half section***.
2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for specifying the dimension.
3. Identify a line or a centre line.
   The dimension line will be drawn perpendicular to this line.
4. Identify a line running parallel to the that will define the shortest distance that is to be dimensioned. The dimension will be completed in accordance with the ***Settings*** against ***Dimension***. The dimension text will show twice the value of the distance that has been computed, prefixed by the symbol ∅.
5. Position the dimension line with the cursor or by entering the appropriate value against ***Distance*** in the dialog box.

   The dialog box fields can be used to alter the *distance* to the object that is dimensioned, the ***Text*** distance from the dimension line, the ***Tolerances*** and the ***Dimension Number***.
6. If the option card ***General*** shows that the ***Settings, Dimension*** contains the active button ***dimension text positioning freely definable***, the dimension text can be dragged along the dimension line and then positioned.

   If you have to complete several dimensions, specify their distances in ascending order.
   Until you have completed the procedure, the dimension lines will

be positioned automatically in accordance with the value specified for the ***distance with reference dimensions***.

7. If the distance that is to be dimensioned to a point not on a parallel line, the Snap Menu must be used to specify the point in question.

Please note:

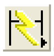
This command cannot be used to complete a reference dimension: the result will be a simple dimension, which means that each dimension can be processed and erased separately.

Dimensioning coordinates

The procedures described below will help you to dimension the centre of a circle/ an arc or any other points, i.e. the point can be denoted by a pair of coordinates taking the form (100.0 / 150.0) ***Coordinate Dimension*** can also be displayed in polar coordinates. The polar coordinates are displayed in the format ***(Distance, Angle)*** to the defined coordinate origin (25,5 / 50°).
The first dimension that is specified indicates the origin and the next the X and Y distances from this point.

The dimensions are completed in the partial drawing in which the first point was defined. If this point is snapped on an object, the dimensions will be completed in the partial drawing containing this object.

1. To create the dimension in polar coordinates, activate the option ***polar coordinates*** using the menu command ***Settings***, ***Dimension*** in the tab ***Dimension Geometry*** in the area ***coordinate dimension***.
,  To change the style of the dimension afterwards, use the command ***Dimension***, ***Process Dimension Properties***.

2. ,  Activate the command ***Dimension, Coordinate Dimension*** by clicking the icons.

3. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for specifying the dimension.

4. Identify a circle with the centre that is to be taken as the origin. You can specify a different point by using one of the Snap Menu commands.

5. Identify a circle in order to determine the X and Y distances of its centre from the origin.
You can specify a different point by using one of the Snap Menu commands.

The dimension text will be centred above the point that has been determined.

6. Complete the same procedure for any other dimensions.

Please note:

If for the current dimension style the option card ***General*** shows that the ***Settings, Dimension*** contains the active button ***dimension text positioning freely definable***, the dimension text can be dragged along the dimension line and then positioned.

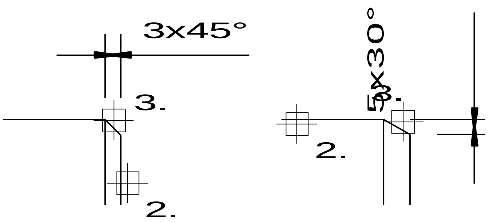
Coordinate dimensions with free defined position (using the shift key while placing) can be drawnwith a reference line between text and tzhe coordinate. This option can be activated using the menu command ***Settings, Dimension*** in the tab ***Dimension Geometry*** in the area ***Coordinate Dimension***. The option defined here effects all coordinate dimension with free text positon which follows the general settings.

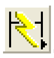
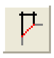
,  In addition the reference line can be activated to coordinate dimensions using the command ***Dimension, Edit Dimension Properties***.
Identify a coordinate dimension and deactivate the option ***according to defaults*** and activate the option ***coordinate dimension***, ***with reference line*** in the tab ***Dimension Geometry***.

The reference line is drawn with the current dimension parameters and gets the display properties from the current dimension layer.

Dimensioning chamfers

The procedure described immediately below can be used to dimension a chamfer:



1. ,  Activate the command by clicking the icons ***Dimension, Chamfer***.

2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for specifying the dimension.
3. Identify the reference line from which the chamfer angle is to be measured and the length calculated.
The dimension will be completed perpendicular to this line.
4. Identify the chamfer line.
You must ensure that this shares and endpoint with the reference line.
5. Use the cursor to specify the position of the dimension line, or make an entry against ***Distance*** in the dialog box.

   The dialog box fields can be used to alter the ***Distance*** from reference line*,* ***Text*** distance from dimension line, the ***Tolerances*** and the ***Dimension Number***.
6. If for the current dimension style the option card *General* shows that the ***Settings, Dimension*** contains the active button ***dimension text positioning freely definable***, the dimension text can be dragged along the dimension line and then positioned.

Dimensioning arc lengths

The procedures described below are for dimensioning the arc length of a circle or an arc.
The dimension will be completed in the partial drawing containing the circle.

1. ,  Activate the command by clicking the icons ***Dimension, Arc Length***.
2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for specifying the dimension.
3. Identify the arc.
4. Click the right mouse button to dimension the complete length of the arc or

   Point to point dimension

   a. Define the first point on the arc determine the arc length by two points.
   b. Press the ***X*** key to switch between the inner and outer arc length.

c. Define the second point.

The dimension is generated according to the current style defined in ***Settings***, ***Dimension***.

5. Use the cursor to specify the position of the dimension line, or make an entry against ***Distance*** in the dialog box.

The dialog box fields can be used to alter the ***Distance*** from reference line, ***Text*** distance from the object that is dimensioned, the ***Tolerances*** and the ***Dimension Number***.

6. If for the current dimension style the option card ***General*** shows that the ***Settings, Dimension*** contains the active button ***dimension text positioning freely definable***, the dimension text can be dragged along the dimension line and then positioned.

Define text reference

With this command, text that has already been positioned can be allocated a reference to any geometry or a point.

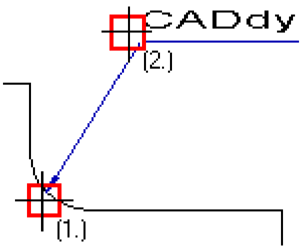
The display of the referenced text is depending on the style defined in the option card ***Referenced Text/Position Number*** using the menu command ***Settings***, ***Text***.

1. Activate the command. ,  or ,  , 

2. Determine the starting point of the reference line using the cursor or the Snap Menu options.

3. Identify the text to be referenced. (2.)
A text reference can also be alligned to multiple texts.
If , for example, a complete text block, which was defined by multiple texts in the drawing, should get one frame and one reference line, then all texts should be selected to align the reference with.

4. A reference line is generated between the defined point and the text. The connection to the text is depending on the settings in the ***Current style for referenced texts***.

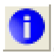
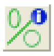


5. The allocation of the reference line to layers is defined in the optioin card ***Referenced Text/Position Number*** using the menu command ***Settings***, ***Text***. The size of the reference symbol is depending on the current dimension settings.

Please note:

Reference lines are associative to the referenced text and change when the text is changed.
For the display properties different styles can be defined using the option card ***Referenced Text/Position Number*** in the dialog box ***Settings***, ***Text***. Changes in this parameters can be assigned to all the referenced texts in the model using the option ***Change style of referenced texts for all existing***.

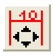
Parameters for referenced texts can also be saved in the menu command ***File***, ***Read/Write Parameters*** (*.mps) and transferred to other models. Basically all parameters for texts, dimensions, position numbers and referenced texts should be saved in Templates (*.TPL).

,  The display of single, referenced text can be edited using the command ***Information***, ***Edit Object***.

,  A referenced text can be disassembled using the command ***Process2D, Disassemble 2D Object***.

Move dimension

The procedure described immediately below can be used to alter the position of an existing dimension:

1. ,  Activate the command by clicking the icons ***Dimension, Move dimension/Dimension text***.

2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selecting.

3. Identify the dimension line or an construction line belonging to the dimension that you want to move.

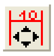
4. Use the cursor to specify the position of the dimension line, or make an entry against ***Distance*** in the dialog box or use a snap option on already existing geometry.

   In the case of a radius dimension the direction can be altered and in the case of diameter dimensions an alteration can be made to the direction and the position inside or outside the circle.

In the case of any other dimensions the change can be made between the dimension and the objects or points that have been dimensioned.

Move dimension text

The procedure described immediately below can be used to move the text relating to dimensions that already exist:

1. ,  Activate the command by clicking the icons ***Dimension, Move dimension/Dimension text***.
2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selecting.
3. Identify the dimension text in such a way that the cursor cross-hair lies inside the (imaginary) rectangle surrounding the dimension text.
4. The dimension text lies freely on the cursor and can be positioned as required.

   If the distance between text and dimension line is to be determined in one defined distance, enter the required value in the ***Dimension Text Distance*** text box of the Status dialog box.

   If the distance between text and dimension line is to be modified, enter the required value in the ***Text*** distance text box of the Status dialog box.

   If the text is to have a specific distance from the position to the midpoint of the dimension line, in the ***Move text*** box, enter the value for the distance. Positive values define a position on the right side, negative on the left side.

   If the text is positioned outside the extension lines, the dimension line is lengthened by the necessary dimension on the text position side. On the other side of the dimension, the dimension line is lengthened by the value determined in the ***overhang*** box in the ***Dimension Geometry*** option card.

   NC dimensioning
   If, whilst moving a dimension text of a NC dimensioning, you also wish to move the dimension construction line, activate this option as the general settings in the dimensioning parameters or only for special NC dimensionings using the ***Dimension, Edit Dimension Properties*** command from the toolbox icon bar.

Edit Dimension properties

By changing the attributes that have been used for dimensions already in position you can alter the dimension style, the format of the dimension elements as well as the dimension text.
The procedure described immediately below can be used to process attributes that already exist for dimensions:

1. ,  Activate the command by clicking either the icons ***Dimension, Edit Dimension Properties***. If dimensions are in the selection list they are offered.

   Alternatively, the command can be activated directly from the empty selection list and then accept the dimension to be processed in the action list.

   Next a dialog box will be presented containing the attributes that can be altered for all the dimensions that are selected independent from the current dimension style.
   If there are several dimensions or several dimensions with different formats (e.g. incremental dimension, radius dimension, angle dimension) in the selection, the only attributes that can be altered are those common to all the dimensions.

2. The alteration options are subject to the setting on the ***General*** option card for ***according to defaults***:
   If the setting is active, the majority of the dimension attributes will be taken over from the current dimension style.

   In this case the alterations listed immediately below can be made, depending on the dimension selected:

   Dimension style
   If different dimension styles are defined, a set of parameters can be choosen from the list of styles. The common settings are taken from this style.

   simple dimension
   dimension number, suffix, prefix, type and value of tolerances;
   with the radius and diameter dimensions a change can be made between the two;
   with distance dimensions a change can be made between horizontal, vertical and aligned dimension.

   Incremental/reference/NC dimension
   Type and value of tolerance entries, however whereby a tolerance entry has an affect on all dimensions in the dimension string.
   Define a tolerance on one of the individual texts in the dimension

string using the ***Dimension, Edit Dimension Text*** command. Change between incremental, reference and NC dimension.

3. If any other attributes relating to the dimension that has been selected need to be altered, you will first have to deactivate ***according to defaults***.
Then specify the attributes.

If the attributes are to be taken over from the ***Settings, Dimension***, you will have to activate ***according to defaults***.

4. You must confirm the changes by clicking either of the buttons ***Accept*** or ***OK***.

Please note:

The dimension text relating to incremental dimensions and to reference dimensions cannot be dealt with by means of this particular command. You will have to use ***Dimension, Edit Dimension text****.*

Only those parameters are changed in the selected dimensions which are altered in the dialog box. If dimensions for example have different tolerances, they are not changed except a new tolerance was entered or the option *Without* was choosen.

Dialog box – Edit dimension properties

The option cards in this dialog box can be used to alter the attributes relating to all the dimensions that are selected. In the case of incremental/reference/NC dimensions the alterations made will apply to the partial dimensions.

This can only be done in active fields with their contents displayed in black. Fields with their contents displayed in grey cannot be selected.

Only those parameters are changed in the selected dimensions which are altered in the dialog box. If dimensions for example have different tolerances, they are not changed except a new tolerance was entered or the option *Without* was choosen.

Option card - General

Parameter set
If different dimension styles are defined, a set of parameters can be choosen from the list of styles. The common settings are taken from this style.

according to defaults
The setting here has a direct influence on whether the attributes relating to selected dimensions are to comply with those in the current dimension style or whether they can be user-specified instead

When the setting is active, the majority of the dimension attributes will be taken over from the current dimension style.
If this setting is deactivated, the selected dimensions can be defined separately from the properties determined in the current dimension style.

The individual settings:

simple dimension
dimension number, suffix, prefix, type and numerical value of tolerances;
the distance dimensions change between horizontal, vertical and aligned dimension.

incremental/reference/NC dimension
Type and value of tolerance entries, however whereby a tolerance entry has an affect on all dimensions in the dimension string. Define a tolerance on one of the individual texts in the dimension string using the ***Dimension, Edit Dimension Text*** command.
Change between incremental, reference and NC dimension

dimension text
You can only alter the settings for the dimension text in relation to <u>one</u> simple dimension.
If the dimension text relating to partial dimensions in an incremental/reference/NC dimension are to be altered, you will have to use ***Dimension, Edit Dimension text****.*

value
 When you want to alter the value of a simple dimension so that it will deviate from the true value, click the button shown on the left so that you can type an entry.
If the filed contains a text that has been altered, you can restore the original dimension value by clicking the button.

Underlining will indicate a dimension text that has been altered.

position
An alteration can be made to the position of the dimension text in relation to the dimension line.

prefix
An alteration can be made to the prefix to be placed before the

dimension number. This can be done by selection or by making an entry. (# = Symbol for square)

suffix
An addition can be made either by alteration or by entry to the text that is to be placed after the dimension number or the tolerance.

distance to dimension line
This entry allows you to modify the distance separating the dimension text and the dimension line. Enter the required value in order to carry out the respective modification in all selected dimensions. If the ***fitted positioning*** is active, this value is not taken into account.

font
The settings for the selected dimension texts (***font***, ***height***, ***width***, ***spacing***, ***proportional***, ***block***) can be changed.

Underline modified text
As long as you have disassembled the dimension from the presettings, this command can be used to decide whether an edited dimension is to be underlined.

Check dimension
If this option is activated, all dimensions that you have tagged in this command will be identified as check dimensions.

With color
Similar to a highlighter, the dimension text is highlighted with a color. Select a desired color by clicking on the color button.

Preciseness of the display

unit
Provided that the selection contains only distance (or length) dimensions, the dimension number can be displayed in the metric system or as inches.
When the contents are only the angle dimensions, these can be displayed in degrees with a number of decimal places and in degrees, minutes and seconds.

decimal places
The number of decimal places can be altered for length and angle dimensions.

display trailing zeros
The setting will specify whether or not the decimal point is to be followed by trailing zeros.

round off architecture

When the dimensions have been completed using the dimensioning type Architecture 0,5/0,7, the rounding can be altered.

tolerance entries

The settings in this section can be changed without the user having to deactivate ***according to default***.

type

The type of tolerance can be altered.
Depending on the tolerance type that is selected, there will be either one field or two fields for entering tolerances.

Tolerance by value

Tolerances on dimensions are interpret as character string (texts) and displayed in the drawing. Therefore it is warranted that the defined tolerance value is displayed consistently, independent from the current model units.
In the case, tolerances should be recalculated when changing the model units (mm, cm, m, inch), the tolerance types *deviation*, *limit dimension* and *symmetric deviation* can now also be handled as numerical values in a dimension.
The display of the tolerance values in this case adjusts to the global or local settings made in the area *Dimension number*.

text size factor

A factor can be entered for scaling the tolerance text in relation to the sizes of the dimension text.

format

Depending on the type of dimensions that have been selected, the format can be altered.
A change is possible between diameter and radius dimension or between incremental/reference/NC dimension.
In the case of distance and length dimensions there is a choice between horizontal, vertical and aligned dimension.

<u>Option card - Dimension Geometry</u>

dimension lines

overhang

This entry will specify the amount by which a dimension line will be lengthened, when the dimension text and/or the dimension limiters extend beyond the extension lines.
The entry will only be of relevance when the limiter specified is the arrow-head.

reference dimension with constant distance
If the choice is restricted to reference dimensions, the setting can be altered.
If the setting is not active, partial dimensions in a reference dimension can be moved individually by using ***Dimension, Move dimension***. In any other case all the partial dimensions will be moved using the specified **distance with reference dimensions**.

Return with radius dimension
In the text box, define a value by which the radius dimension is to be drawn moved from the centre line. A positive value shorten the dimension line, a negative value lengthens it.

Radius dimension line to midpoint
If this option is activated, the dimension line of radii dimension is drawn up to the midpoint of the radius. This option can only be drawn if the ***according to settings*** switch is deactivated.

Draw dimension line
If this switch is deactivated, the dimension line is not drawn from the selected dimensions. This option can only be drawn if the ***according to settings*** switch is deactivated.

dimension auxiliary lines
distance to contour
If this setting is active, the entry will specify the distance between the object that has been dimensioned and the extension line. The **dimension line overhang** will then define the amount by which the extension line is to extend beyond the dimension line.

with fixed length
If this setting is active, the value entered will specify the total length of the extension line. The entry against **relation to dimension line** will specify the ratio of the part of the extension line further from the contour to that closer to the contour.

return with radius dimension
This value determines the shortening of the dimension line for radius dimensions within the framework of the command **dimension radius**. The defined return shortens the dimension line starting from the defined symmetry axis/the defined symmetry point. This option should be used if the symmetry axis/symmetry point of the radius lie outside the active drawing area.

This value is only taken into account if a radius dimension is processed.

Parallel arc dimension construction lines
If this option is activated, the dimension construction lines are drawn parallel for arc dimensionings.

Slope angle
For the single dimensions ***Length of Lines*** and ***Distance of Points*** an inclination can be defined afterwards to the dimension auxiliary lines.

dimension limits

limit type
A type of dimension limiter can be selected.

position
It the limiter that is chosen is the arrow-head, the position for it can be specified.

display options
In this option it can be defined if the dimension limits should only be displayed on the left, on the right or both or not.

NC Dimension

dimension text parallel to dimension line
If the setting is active, the dimension text for NC dimensions will run parallel to the dimension line. If the setting is not active, they will run perpendicular to it.

limit type origin
You can specify whether the origin for an NC dimension is to be denoted by a small circle or by a filled circle.

text displacement
When the dimension text orientation is perpendicular, the spacing from the dimension line is derived from the total of the extension line overhang beyond dimension line and the value to be entered her for the displacement.

Moved text with lead
If this option is activated, the position of the dimension text to the dimension line is recognised by a dimension reference line. The dimension line that underlines the dimension text, is known as ***lead***. This option can only be drawn if the ***according to settings*** switch is deactivated.

Length of lead
Use this value by entering a positive value, to define the length overhang of the lead to the dimension text, a negative value shortens the lead. If the value is 0, the length of the lead corresponds to the dimension text.

Text distance
The value in this field defined the distance between the dimension text and the dimension line.

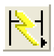
Coordinate Dimension

With Referenc Line
Coordinate dimensions with free defined position (using the shift key while placing) can be drawn with a reference line between text and the coordinate. The reference line is drawn with the current dimension parameters and gets the display properties from the current dimension layer.

Display
The display attributes can be specified here (colour, line type, and line width) for the partial objects in a dimension. This can only be done with this command for the objects.
If the changes are to be effected via the layers, you will have to use the ***Layer Explorer*** inside the ***Model Explorer***.

Edit dimension text

There are different ways of editing dimension text that have already been completed:

1. ,  Activate the command by clicking the icons ***Dimension, Edit Dimension text***.
2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selecting.
3. Identify the dimension text for the (partial) dimension that has a text for editing.
4. The dialog box will be presented with the current elements used in the dimension text that has been selected. The possibilities to be entered in every field are described in the following.
5. Specify the entries for each separate element:

   With color
   Similar to a highlighter, the dimension text is highlighted with a color. Select a desired color by clicking on the color button.

   The entry in the ***Prefix*** box will be placed in front of the dimension number, the entry in the ***Suffix*** box will be appended to the dimension number or, where applicable, after the tolerance text. (# = Symbol for square)

The fields for the ***tolerances*** can be used to define the type of tolerance. The respective entries for the tolerances can be made in the text boxes below.

Tolerances on dimensions are independent from the current model units. In the case, tolerances should be recalculated when changing the model units (mm, cm, m, inch), the tolerance types *deviation*, *limit dimension* and *symmetric deviation* can be handled as numerical values by set the option ***by value***.

If the ***dimension numbe****r* is to be altered, you cannot do so until the box above  can be accessed. Sometimes there will already be a changed text in the box and so you will have to click the button to obtain the original dimension number again.

An altered dimension text can be recognised by the underlining, as long as the option is activated in the general dimensioning parameter

6. Confirm the entries by clicking ***OK***, or end the procedure by clicking ***Cancel***

Alternatively, you can click the two icons ,  ***Dimension, Edit Dimension Properties*** to activate the commands to ***Edit Dimension text*** used with simple dimensions.

Examples:

Dimension text (dimension number 10) in the form 5x∅10
Prefix: 5x∅

Dimension text (dimension number 40) in the form 4x10=(40)
Prefix: 4x10=(
Suffix:)

Dimension text (dimension number 32) in the form □ **32**
Prefix: **#**

Please note:

If there are several dimension texts to be given a prefix such as a diameter symbol, or if it is to be deleted, use the command ***Dimension, Remove/Add Additives***.

If a dimension text that has been changed is not to be so indicated by underlining, erase the number denoting the actual dimension and then enter the number you want in the box for the prefix or suffix. Be careful to ensure that the text is constant and not associative like a genuine dimension text.

Add/Remove Additives

Whenever an existing dimension text is to have a prefix, such as the preceding diameter symbol Ø or a suffix and/or a tolerance added, altered or deleted, use the following method:

1. , Activate the command by clicking the icons ***Dimension, Add/Remove Additives***.
2. If the additive is going to be added, select it from the list in the dialog box, or type in the necessary entry. When choosing a tolerance select the mode before entering the values and activate the option ***As Value*** if the tolerance should be recalculated when changing the units (mm, cm, m, inch) (possible only for the tolerance types möglich für die Toleranzarten *Deviation*, *Limit dimension* and *Symmetric deviation)*.

   If the additives are going to be removed, choose the empty line in prefix, place the cursor in the empty field for suffix and/or choose the mode without for tolerances.
3. Accept the entry by clicking ***OK***, or end the procedure by clicking ***Cancel***.
4. Insert the dimension texts in the action list where they can be dealt with.

Please note:

A dimension text can be given only one prefix. Thus, if there is already a prefix, it will be deleted before the new one is inserted.

Any dimensions that are to be dealt with can be inserted in the selection list first. But it is very important to ensure that, in the case of string dimensions, NC dimensions and reference dimensions, all the partial dimensions have to be dealt with identically.

Adjustment of NC Dimensions

NC dimension text is automatically adjusted so that it can no longer conceal itself.
Via ***Settings, Dimension*** in the menu bar, activate the ***moved text with lead*** option on the ***Dimension Geometry*** tab. Define desired parameters for the ***length of the lead*** and the ***text distance***.

, Start the command using the buttons shown above and identify an NC Dimension. The dimension text is arranged according to the preset parameters so that it can no longer conceal itself.

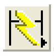
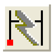
Note:

Every single dimension text can still be moved using the command ***Move Dimension / Dimension Text***. To freely move the dimension text press the Shift key in addition.

If the option ***moved text with lead*** in the menu ***Settings, Dimension*** is active, newly generated NC dimension are automatically positioned without interferences.

Define reference point

Use the procedure described below with the NC dimensions to change the origin:

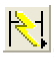
1. ,  Activate the command ***Dimension, Define reference point***.
2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selecting.
3. Identify the extension line for an NC dimension that is to denote the origin in future.

Please note:

You can also use this command to change the reference point for reference dimensions, if it has been defined incorrectly while you were dimensioning the length of a line.

Insert partial dimension

The following procedure can be used to add a partial dimension or several partial dimensions to any of the simple, incremental/reference/NC and coordinate dimensions that have been completed with ***Dimension, Multidimension Orthogonal*** and ***Multidimension Aligned*** *and* ***Coordinate Dimension***:

1. ,  The command is activated by clicking the icons ***Dimension, Insert Partial dimension***.
2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selecting.
3. Identify the dimension that is going to be expanded.
4. Specify the distance that is to be dimensioned by identifying a line parallel to the construction lines or by defining a point.
   Identify a circle in the case of coordinates dimensioning, or specify

a point. The dimension text will be positioned above the point that is determined in this way.

5. If the dimension that has been identified is a partial dimension in an incremental, reference or coordinate dimension, the new dimension will be added using the parameters that apply to the existing dimension.

If the dimension that has been identified is a simple length or distance dimension, use the position of the cursor to indicate whether the new dimension is to be an incremental dimension or a reference dimension. Confirm by clicking the left mouse button.
If the cursor is within the face of the dimension text, or an imaginary extension of it, an incremental dimension will be completed; in other cases it will be a reference dimension.

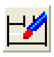
Please note:

If a partial dimension contains a dimension text, which was modified using the ***Move dimension text*** command, the dimension text, which was edited is currently moved to an adjoining partial dimension where necessary.

When you are going to expand a simple dimension into an NC dimension, first expand it to a reference dimension. Then use ***Dimension, Edit Dimension Properties*** to change the dimension format.

Remove partial dimension

Incremental/reference/NC and coordinate dimensions are always dealt with as individual objects. If partial dimensions are to be deleted without their being erased completely, the following procedure must be used:

1. ,  The command is activated by clicking ***Dimension, Remove Partial dimension***.

2. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selecting.

3. Identify the partial dimension that you want to delete.
In the case of "inside" partial dimensions with string dimensions and NC dimensions you must identify one of the extension lines so that the two adjacent dimensions can be combined into a single dimension.

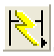
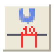
Please note:

If a partial dimension contains a dimension text, which was modified using the ***Move dimension text*** command, the dimension text, which was edited is currently moved to an adjoining partial dimension where necessary. **IXI**

Disassemble partial dimension from dimension string

The command ***Disassemble Partial Dimension From Dimension String*** can be used to disassemble individual dimension from a reference or string dimensioning.

The disassembled dimension is then an individual dimension and can be processed and positioned as one.
The rest of the dimensions of a reference dimensioning are retained in the previous form. A string dimensioning is divided into 2 separate string dimensionings if the disassembled dimension finds itself within the string. In order to disassemble a dimension from a dimension string, proceed as follows:

1. ,  Load the command ***Dimension, Disassemble Partial Dimension From Dimension String***.

2. If the 2D drawing window is active, use the ***Access to ... partial drawing*** commands at the top of the window frame if necessary to determine which partial drawings are to be taken into consideration during the selection.

3. Identify the partial dimension that is to be disassembled from the dimension string.

Please note:

When disassembling a partial dimension from a reference dimension the option constant distance is deactivated. It can be activated again using the command ***Dimension***, ***Edit Dimension Properties*** in the option card ***Dimension geometry***.

Disassemble dimension

Dimensions are managed as a combined object. This command can be used to disassemble dimensions into their individual component parts. The dimensions to be processed are removed from the model and replaced by 2D line elements, if necessary fill faces and text. In order to disassemble a dimension, proceed as follows:

1. ,  Load the command **Dimension, Disassemble Dimension**.

2. Identify the dimension to be disassembled.

Please note:

,  Dimensions can also be disassembled using the command ***Process2D, Disassemble 2D Objects***.

Fit dimensions table

A table of this kind cannot be created unless there a re already dimensions with tolerances of the *fit dimensions* type and is for tolerance classes.
The method for indicating tolerances for different dimension styles can be set by using the commands ***Settings, Dimension*** ; otherwise you can allocate existing dimensions with ***Dimension, Edit Dimension Properties***.

The fit dimension table will be saved as an object in the current group.

The display attributes of the partial elements (frame, fit dimensions, deviations) are to be specified by the layers defined with the help of ***Settings, Fit dimensions table***.

The text parameters for the fit dimension table are according to the current text style.

The entries in the table or the text parameters can be updated only by completing a new table.

Specifying the settings for the fit dimensions table

The commands to be used for specifying the defaults for the table are ***Settings, Fit dim table***.

table titles
The entries in this section must be the text for the table column headings for the fit dimensions and the deviations.

deviation dimensions
tolerance value
This option must be selected, if the upper and lower deviations are to be entered as the tolerances.
absolute value
This option must be selected, if the delimiter dimensions are to be entered.

display calculated deviation dimensions
The deviations will be computed in a database. If this is incomplete, or if a nominal value lies outside the permitted interval, the deviations will be computed.
When this option is active, the deviations that have been computed

will be presented inside a dialog box so that they can be edited. When the option is not active, the deviations that are computed will be taken over directly into the fit dimensions table.

layers

After clicking this icon you will be able to select the layers for the elements in the fit dimensions table: these layers will have a direct effect on the display attributes of the elements.
The layer reserved for the text in the fit dimension table is at the same time reserved for the table headings.

If the layers that you have entered do not exist, they will be inserted below the *Standard* layer when you insert the dimension table.

Insert fit dimensions table

The procedure for completing and inserting a fit dimensions table is described below:

1. , Activate the command by clicking the icons ***Dimension, Generate Fit Dimensions Table***.
All the partial drawings belonging to the current drawing will be searched for nominal dimensions and the associated symbols for tolerance classes.
2. The fit dimensions database will be used to compute the deviations for each nominal dimension.
If the default for the ***Display calculated deviation dimensions*** is the ***Settings, Fit dimensions*** is active, the deviations that are computed will be presented inside a dialog box for editing.
3. The fit dimensions table will be completed in accordance with the currently valid defaults.
If there is no such table in the drawing, it can be positioned by using its upper left corner.
If there is already a table, it ill be erased and replaced by the current one.

Please note:

After a dimension or tolerance has been changed in a drawing, or had an addition made, or has been deleted, you must remember to generate a new fit dimensions table.

If the layer structure dies not contain the layers preset for the fit dimensions tables, they will be inserted in the layer structure.

4.12 Drawing additions

 The commands for inserting and processing drawing additions can also be loaded using this icon in the 2D toolbar.

With the commands within here, identification for the surface quality and identification for the form and positional tolerances can be inserted and edited in the drawing.

In addition, you also have the opportunity to insert section identifiers in your drawing.

Settings for surface entries and form and positional tolerances

In each of the dialog boxes for the generation or processing of surface entries, form and positional tolerances, welded seams and edge symbols there is a ***Parameters*** command button.

In the dialog boxes for the ***Definition*** of surface entries and form and positional tolerances define the settings for the new objects in your drawing. The settings made in these dialog boxes are (like the presettings of the surface drafting symbol for surface roughness) saved in the UIBECKER.SYS file. The parameters can also be stored using the command Write Parameters in the ***File*** menu.
Existing surface drafting symbols for surface roughness and form and positional tolerances can be modified in your appearance drawing using the parameters in the dialog box to ***Process*** these objects.

If you activate the ***Parameters*** command button, the Parameters dialog box opens. In this dialog box, you can make the following settings:

- The according to dimension parameters button can be used to determine whether the drawing additions are defined in their appearance form according to the general dimension parameters. If this command button is deactivated, you can determine the settings that deviate from the dimensions.
- The switch according to dimensioning parameters can be used to determine whether the drafting symbol for surface roughness or the form and positional tolerance are defined in the appearance form according to the general dimensioning parameters. If this is deactivated, you can make differing parameters to the dimensioning.
- Layer allocation can also be modified if the according to dimensioning parameters switch is activated.

Please note:

The font always orientates itself on the current font for dimensions. This font can be defined in the menu ***Settings, Text***.

 Finish characteristics can also be disassembled using the command ***Process2D, Disassemble 2D Objects***.

Finish characteristics

In the 2D drawing area, entries concerning the finish characteristics can be defined, inserted in the drawing and processed.

The content of the entries for the finish characteristics can be defined in a dialog box after loading the command ***Define Finish Characteristics***.

They can then be positioned freely or with reference to existing lines.
If the finish characteristics refer to a line object, it is linked with this. This means that modifications to the position of the lines also affect the finish characteristics as well as the erasure of the line.
Freely positioned finish characteristics are independent of other objects.

The size of the finish characteristics is determined by the current values that are specified using ***Settings, Text*** in the ***current style for dimension texts***. The size and type of the reference arrows will also depend on the settings for the dimension. For the individual formatting of finish characteristics, select ***Drawing Additions***, ***Process Finish Characteristics*** and use the ***Parameter*** dialog box which then appears.

The ***Process Finish Characteristics*** command can be used to later modify the content and/or position of a finish characteristic.

In the dialog box which then appears for ***Parameter***, you can reformat the existing finish characteristics afterwards.

Partial drawing
Finish characteristics with reference to a linear object are generated in the partial drawing in which the object is and finish entries without a reference will be generated in the active partial drawing.

Group
Finish characteristics are saved in the **current group** as objects of type 2D standard part. This can be selected from the existing groups using group selection in the main toolbar.
Group allocation can later be modified in the ***Model Explorer***.

Colour, line type, line width

A finish characteristic will be linked to the currently valid **layer(s) for dimensions**. These can be specified in the ***Layer Explorer*** in the ***Model Explorer***.

All the elements in the finish characteristics will be allocated the display attributes for colour, line type and line width as specified for this layer already.

The display of line type and line width corresponds to the settings made using the menu command ***Settings, Options***.

The link with a layer as well as colour, line type and line width can later be modified using the menu command ***Process, Object Display***.

To ensure that whilst processing your drawing, only specific object types can be identified, use ***Settings, Object Filter*** to activate the snap possibilities for 2D standard parts and deactivate the other object types. It must be noted that finish characteristics cannot be copied.

Please note:

> Finish characteristics cannot be processed directly by using the commands in the Transformation menu. When they have been positioned with reference to a linear object, they will be transformed when the object itself is transformed.
> If finish characteristics are to be re-positioned, you must use the command ***Drawing Additions, Process Finish Characteristics***.
>
> ,  Finish characteristics can also be disassembled using the command ***Process2D, Disassemble 2D Objects***.

Define finish characteristics

  Alternatively, the command can also be loaded using this icon in the 2D toolbar.

After you have loaded the command, a dialog box is opened which can be used to determine all definitions for a finish characteristic and then select one of the positioning types for positioning in the drawing.

1. Activate one of the command buttons ***Pre-assigned 1*****, ., *****Pre-assigned 5***.
   When positioning the finish entries or when closing the dialog box they are given the defined values. These values can be activated directly when loading the command again using the respective command button.
   This means that four of the pre-allocations can be used

permanently for frequently required finish entries, one is given the currently defined values.

2. In the ***Symbol*** block, select the required settings or enter the values.
   In the ***Preview*** window, the appearance of the finish entries is displayed for checking purposes.

   Type of symbol
   In the list above this line, the type of basic symbol to be used can be selected.

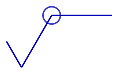
   Roughness top limit, lower limit
   Select one of the displayed roughness values from the list or enter a roughness degree in the text box, e.g. N1. Confirm the entry with Enter.

   Manufacturing process
   Select one of the processing procedures from the list or enter the required name.

   Reference line
   If necessary, enter the length of a reference line.

   Other roughness
   In both these lists, select the type of roughness dimension sizes and their value.
   Both text boxes can be edited, if these values should be in brackets, e.g. (Rz = 0,4).

   valid around the contour
   If all surfaces of the part should have the same surface properties, a circle is added to the surface symbol.

   Groove direction
   If necessary, select a groove direction. The symbol ! stands for ⊥ "vertical to projection plane".

   Processing addition
   If necessary, enter the required value for a processing addition.

3. Activate the command by clicking on one of the command buttons with the illustration of a positioning type in order to insert the finish entries in the drawing.

   When selecting the *OK* command button, the dialog box is closed and the values determined for the selected pre-allocation are saved.

If the dialog box is opened again, the last selected pre-allocation is activated again.
On finishing a work session with *BeckerCAD*, the determined settings are saved in the file UIBECKER.SYS.
The parameters can also be stored using the command Write Parameters in the ***File*** menu.
Please note:

, Finish characteristics can also be disassembled using the command ***Process2D, Disassemble 2D Objects***.

Insert finish entries without reference

Alternatively, the command can also be loaded using this icon in the 2D toolbar.

This command can be used to position finish entries freely in the drawing.

1. Determine the settings for the finish entries or select a set of settings using one of the command buttons ***Pre-assigned 1***, ., ***Pre-assigned 5***.
2. Select the illustrated command button.
3. Determine the position of the entry using the cursor, a command from the Snap Menu or by entering the coordinates in the Status dialog box.
4. Position a further finish entry in the same way or Cancel the command using the right mouse button.

Insert finish entry on line

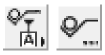
Alternatively, the command can also be loaded using this icon in the 2D toolbar.

This command can be used to position finish entries in the drawing on existing line objects and link them with these.

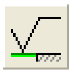
1. Determine the settings for the finish entries or select a set of settings using one of the command buttons ***Pre-assigned 1***, ., ***Pre-assigned 5***.
2. Select the illustrated command button.

3. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selecting.
4. Identify the line object at the position at which the finish entry is to be positioned.
It is aligned parallel to the line to which it refers and on the side of the line object from which you click the object.
5. Position a further finish entry in the same way or Cancel the command using the right mouse button.

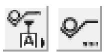
Insert finish entry with construction line

 Alternatively, the command can also be loaded using this icon in the 2D toolbar.

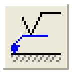
This command can be used to position finish entries in the drawing: They are positioned on construction lines which are drawn in the extension of the line object to which the entry refers.
Construction lines and finish entries are linked with the line object.

1. Determine the settings for the finish entries or select a set of settings using one of the command buttons ***Pre-assigned 1*, ., *Pre-assigned 5***.
2.  Select the icon shown here to position.
3. If the 2D drawing window is active, if necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selecting.
4. Identify the line object to which the finish entry is to refer. The position of the finish characteristic on the construction line depends on the side on which you identify the line object.
5. Define the length of the construction line dynamically using the cursor.
The entry is aligned parallel to the construction line.
6. Position a further finish entry in the same way or Cancel the command using the right mouse button.

Insert finish entry with individual reference

 Alternatively, the command can also be loaded using this icon in the 2D toolbar.

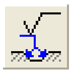
This command can be used to position finish entries in the drawing with reference lines to a line object.
Reference lines and finish entries are linked with the line object.

1. Determine the settings for the finish entries or select a set of settings using one of the command buttons ***Pre-assigned 1*, ., *Pre-assigned 5***.
2.  Select the icon shown here to position.
3. If the 2D drawing window is active, if necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selecting.
4. Identify the line object to which the finish entry refers, on the place where the dimension arrow-head is to be positioned.
5. Define the position of the finish entry dynamically using the cursor.
6. Position a further finish entry in the same way or Cancel the command using the right mouse button.

Insert finish entry with multiple reference

  After clicking on the displayed icons, the ***Drawing Additions, Define Finish Characteristics*** command is started.

This command can be used to position finish entries in the drawing. They are joined with the two lines to which the entries refer, by two reference line with dimension arrows.
Reference lines and finish entries are linked with the line objects.

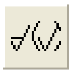
1. Determine the settings for the finish entries or select a set of settings using one of the command buttons ***Pre-assigned 1*, ., *Pre-assigned 5***.
2.  Select the icon shown here to position.
3. If the 2D drawing window is active, if necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selecting.
4. Identify the first line object to which the finish entry refers, on the place where the dimension arrow-head is to be positioned.
5. Identify further line objects to which the finish entry refers, on the place where the dimension arrow-head is to be positioned.

6. Press the right mouse button to quit identification of the reference objects and define the point on which the reference line is to be added to an endpoint.
7. Define the position of the finish entry dynamically using the cursor.
8. Position a further finish entry in the same way or cancel the command using the right mouse button.

Enhance finish entry by addition

  Alternatively, the command can also be loaded using this icon in the 2D toolbar.

This command can be used to enhance any finish entries without reference by one or two additional entries in brackets.

1. Determine the settings for the finish entries or select a set of settings using one of the command buttons ***Pre-assigned 1***, ., ***Pre-assigned 5***.
2.  Select the icon shown here to position.
3. If the 2D drawing window is active, if necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selecting.
4. Identify a finish entry which is to be positioned without reference. If a finish entry which already has an addition allocated to it is to be allocated a second, identify the first addition entry.

Process finish entry

  Alternatively, the command can also be loaded using this icon in the 2D toolbar.

This command can be used to process finish entries in the drawing. The command can also be used to re-position these objects.

1. If the 2D drawing window is active, if necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selecting.
2. Identify the finish entry to be modified. The dialog box for defining the finish entry settings is then opened. It contains the values for the identified entry.
3. Determine the new settings.

4. If the position of the entry is not to be modified, confirm the modifications by pressing the command button *Close*.

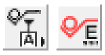
   If the position of the entry is to be modified, click on one of the command buttons for positioning. Then define the new position as for inserting finish entries.

Please note:

This command cannot be used for processing finish entries with additional entries. These must be erased and inserted again.

 Finish characteristics can also be disassembled using the command ***Process2D, Disassemble 2D Objects***.

Modify format of finish entries

 After clicking on the displayed icons, the ***Drawing Additions, Process Finish Characteristics*** is started. The parameter option in the dialog box that appears then opens a dialog to explicitly format finish characteristics.

The ***according to dimension parameters*** button determine whether the finish characteristics are formatted explicitly or in tendency towards the general presettings. In order to format finish characteristics explicitly and individually, deactivate the button.

If the button is deactivated, text parameters and parameters for the reference arrow can be defined using the respective options. Accept the modification with OK.

Please note:

The font always orientates itself on the current font for dimensions. This font can be defined in the menu ***Settings, Text***.

Layer allocation for finish characteristics can also be modified, if the ***according to dimension parameters*** button is activated.

Explicitly defined parameters can be replaced by the general settings from the dimensioning parameters by activating the ***according to dimension parameters*** button.

Form and positional tolerance

In the 2D drawing window, entries concerning form and positional tolerance can be defined, inserted in the drawing and processed.

The content of an entry for form and positional tolerance or a reference identification can be defined in a dialog box after loading the command ***Define Form/Pos Tolerance***.

Then positioning can take place with or without reference to existing line objects.
With reference to line objects means that positional modifications to the line also have an affect on finish entries, as does the erasure of the line.
Tolerance entries and reference identification which are positioned without reference, are independent of other objects.

The size of the annotation and of the tolerance border, of the tolerance information and of the reference identifier are determined by the current values specified in ***Settings, Text*** in ***current style for dimension texts***. The size and type of the reference arrows is taken over from the settings for dimensions.(Menu: ***Settings***). To define an individual style for the form and positional tolerances define the parameters in ***Drawing Additions, Define Form / Positional Tolerances*** using the button ***Display Parameters..***

The ***Process Form and Positional Tolerance*** command can be used to later modify the content and/or position of the form and positional tolerance.

In the dialog box which then appears for ***Parameter***, you can reformat the existing form and positional tolerances afterwards.

Partial drawing
Form and positional tolerances with reference to a linear object are generated in the partial drawing in which the object is and finish entries without a reference will be generated in the active partial drawing.

Group
Form and positional tolerances are saved in the **current group** as objects of type 2D standard part. This can be selected from the existing groups using group selection in the main toolbar.

Group allocation can later be modified using the ***Model Explorer***.

Colour, line type, line width
The partial objects in form and positional tolerance will inherit the display attributes from the current layer(s) specified for dimensioning. They can be changed by using the ***Layer Explorer*** in the ***Model Explorer.***
In addition, a layer with the name *FLSYM* will be inserted in the layer structure specifically for the symbol indicating the tolerated property and for the reference triangle. This layer determines their display

attributes. Create a layer ***FLSym*** in your standard template and define a line width for this layer to see the symbols clearly in every print. Otherwise the line width for the symbols can be defined separately in every model.

The display of line type and line width corresponds to the settings made using the menu command ***Settings, Options***.

The link with a layer as well as colour, line type and line width can later be modified using the menu command ***Process, Object Display***.

To ensure that whilst processing your drawing, only specific object types can be identified, use ***Settings, Object Filter*** to activate the snap possibilities for 2D standard parts and deactivate the other object types. It must be noted that shape a position tolerances cannot be copied.

Please note:

It is important to remember that tolerance information and the identification of references cannot be worked on directly from the transformation menu. If they have been positioned with a reference to a linear object, they will be transformed at the same time as the object itself.
If tolerance information and the identification of references are to be re-positioned, you can use the command ***Drawing Additions, Process Form/Pos Tolerance***.

 Form and positional tolerances can also be disassembled using the command ***Process2D, Disassemble 2D Objects***.

Settings for form and positional tolerance

According to the presetting, the appearance of ***Form and positional tolerances*** correspond to the appearance of general dimensions.

 After clicking on the displayed icons, the ***Drawing Additions, Form and Positional Tolerance*** command is started. The parameter option in the dialog box that appears then opens a dialog to explicitly format finish characteristics.

The ***according to dimension parameters*** button determines whether the form and positional tolerances are formatted explicitly or in tendency towards the general presettings. In order to format finish characteristics explicitly and individually, deactivate the button.

If the button is deactivated, text parameters and parameters for the reference arrow can be defined using the respective options.

Please note:

The font always orientates itself on the current font for dimensions.

This font can be defined in the menu ***Settings, Text***.
Layer allocation for form and positional tolerances can also be modified, if the ***according to dimension parameters*** button is activated.

, Form and positional tolerances can also be disassembled using the command ***Process2D, Disassemble 2D Objects***.

Define tolerance entry and reference identification

Alternatively, the command can also be loaded using this icon in the 2D toolbar.

After you have loaded the command, a dialog box is opened in which all definitions for form and positional tolerance or for a reference identifier can be determined and one of the positioning types can be selected for positioning in the drawing.

Define tolerance entry

1. In order to determine the format, i.e. which principle design the tolerance frame should have, activate one of the command buttons ***Fola 1, .., Fola 4***.

For the entry of the required values, there are three line pairs available. In this case, the text boxes in the top and bottom lines belong together.
In order to erase all entries which belong to one line, use the L button at the end of the line pair.

Depending on the previously selected type of tolerance frame, only one part of the text boxes in this dialog box can be sued for determining the values. The others are negative.

2. Select the required settings and enter the values - depending on the selected type of tolerance frame.
In the ***Preview*** window, the appearance of the tolerance entry is displayed for checking purposes.

Property
This list can be used to select the type of symbol used above the name of the tolerance attribute, e.g. parallelism or symmetry.

Tolerance value
Enter the numeric tolerance value in the central text box (with decimal point).
If the numeric value is preceded by a ∅ character, it can be selected from the previous list.

The symbol for the identification of a projected tolerance zone or the maximum material condition can be selected from the next list. If the tolerance is valid on a limited length, enter the value (with decimal point) in the last text box in this face.

Reference
Up to three names for references can be entered, e.g. A, A-B, after this one of the additional symbols for a projected tolerance zone or the maximum material condition can be selected.

Additional text
In the top text boxes for the additional text, individual or double line text can be entered, which will be inserted above the tolerance frame.

In the lower text boxes for additional text, individual or double line text can be entered, which will be inserted to the right of the tolerance frame.

Reference triangle
If the ***on lines*** command is used for positioning a tolerance entry, a reference line with reference triangle is drawn.
You can select whether the triangle symbol used is to be ***filled*** or ***empty***.

Reference object
This setting concerns the type of definition of the endpoint of a reference line on a line object and the relation of the tolerance entry after positioning:
with reference to line object
The endpoint of the reference line is snapped on the line object as the next point from the cursor position. The tolerance entry is linked with the line object.
without reference to line object
The endpoint of the reference line is determined by definition using the cursor or any command on the Snap Menu.
The tolerance entry is independent of other objects.

<u>**Define reference identifier**</u>

1. Enter the reference character in the ***Reference symbol*** block.
2. In addition, determine the following two settings:

   reference triangle
   You can select whether the triangle symbol used is to be ***filled*** or ***empty***.

reference object
This setting concerns the type of definition of the endpoint of a reference line on a line object and the relation of the tolerance entry after positioning:
with reference to line object
The endpoint of the reference line is snapped on the line object as the next point from the cursor position. The tolerance entry is linked with the line object.
without reference to line object
The endpoint of the reference line is determined by definition using the cursor or any command on the Snap Menu.
The tolerance entry is independent of other objects.

After quitting a *BeckerCAD* work session, the determined settings are saved in the file UIBECKER.SYS. The parameters can also be stored using the command Write Parameters in the ***File*** menu.

Please note:

,  Form and positional tolerances can also be disassembled using the command ***Process2D, Disassemble 2D Objects***.

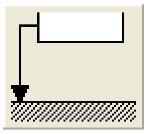
Insert tolerance entry on line

  Alternatively, the command can also be loaded using this icon in the 2D toolbar.

This command can be used to position tolerance entries which have a reference character in the 2D drawing window. The procedure depends on the selected setting in the ***Reference object*** box:

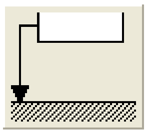
1. Using one of the ***Fola 1, .., Fola 4*** commands, select the required format of the tolerance frame and define the required value for the tolerance entry.

Reference object: with reference to line object

2. Select the ***with reference*** setting for ***Reference object***.
3.  Click this button.
4. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selection.
5. Identify the line object to be toleranced.

6. Use the cursor to dynamically drag the tolerance entry, the respective reference line and, if necessary, an added construction line to the required position.
   Before confirming the position using the left mouse button, the alignment of the tolerance frame can be modified from right to left and vice versa by confirming the *Tab key*.
7. Position an identical tolerance entry in the same way or Cancel using the right mouse button.

Reference object: without reference to line object

2. Select the ***without reference*** setting for ***Reference object***.
3.  Click this button.
4. Define the starting point of the reference line on the line object to be toleranced using a command from the Snap Menu.
5. Use the cursor to dynamically drag the tolerance entry, the respective reference line and, if necessary, an added construction line to the required position.
   Before confirming the position using the left mouse button, the alignment of the tolerance frame can be modified from right to left and vice versa by confirming the Tab key.
6. Position an identical tolerance entry in the same way or Cancel using the right mouse button.

Please note:

If a tolerance entry is positioned ***with reference to line object***, this is also transformed when transforming the toleranced object.

The symbols for the properties of form/positional tolerances, lie on the ***FLSym*** layer. This is set up automatically as soon as a form/positional tolerance has been inserted in the drawing. If you define this layer with properties in a template, all form/positional tolerances in the drawings have the allocated line width due to this template.

Insert tolerance entry on two lines

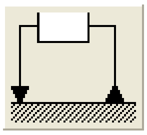
 Alternatively, the command can also be loaded using this icon in the 2D toolbar.

This command can be used to position tolerance entries with direct reference to another object in the 2D drawing window. The procedure depends on the selected setting in the ***Reference object*** box:

1. Using one of the ***Fola 1*, ., *Fola 4*** commands, select the required format of the tolerance frame and define the required value for the tolerance entry.

Reference object: with reference to line object

2. Select the ***with reference*** setting for ***Reference object***.

3.  Click this button.

4. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selection.

5. Identify the line object to be toleranced.

6. If the reference triangle is to be positioned to the left of the tolerance frame, turn it to the left by pressing the *Tab key*.

7. Use the cursor to drag the tolerance entry dynamically, the respective reference line and, if necessary, an added construction line to the required position. Confirm using the left mouse button.

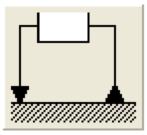
8. Identify the reference object, i.e. the line object on which the reference triangle is to be positioned.
The alignment of the triangle is adapted to that of the line.

9. Define the position of the reference line dynamically with the cursor. If required, an additional construction line is inserted.

10. Position an identical tolerance entry in the same way or Cancel using the right mouse button.

Reference object: without reference to line object

2. Select the ***without reference*** setting for ***Reference object***.

3.  Click this button.

4. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selection.

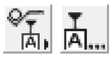
5. Define the starting point of the reference line on the line object to be toleranced using a command from the Snap Menu.

6. If the reference triangle is to be positioned to the left of the tolerance frame, turn it to the left by pressing the *Tab key*.
7. Use the cursor to dynamically drag the tolerance entry, the respective reference line and, if necessary, an added construction line to the required position or define using a command from the Snap Menu.
8. If necessary, specify the partial drawing to be used for selecting the reference object.
9. Define the position of the reference triangle on the reference object.
   The triangle is drawn in its original position.
10. Position an identical tolerance entry in the same way or Cancel using the right mouse button.

Please note:

If a tolerance entry is positioned ***with reference*** to two lines, it is also transformed if one of these lines is transformed. With this, the reference to the other line is cancelled.

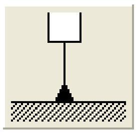
Insert reference identifier

 Alternatively, the command can also be loaded using this icon in the 2D toolbar.

This command can be used to position reference identifiers to tolerance entries in the 2D drawing window. The procedure is dependent on the selected setting in the ***Reference object*** box:

1. Enter the reference character.

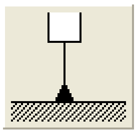
Reference object: with reference to line object

2. Select the ***with reference*** setting for ***Reference object***.
3.  Click this button.
4. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selection.
5. Identify the reference object.
6. Use the cursor to dynamically drag the reference identifier, the respective reference line and, if necessary, an added construction line to the required position. Confirm using the left mouse button.

7. Position a reference identifier in the same way or Cancel using the right mouse button.

Reference object: without reference to line object

2. Select the ***without reference*** setting for ***Reference object***.

3.  Click this button.

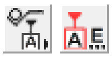
4. Define the position of the reference triangle using the cursor or a command from the Snap Menu.

5. Define the position of the reference identifier and the respective reference line dynamically using the cursor or with a command from the Snap Menu.
The reference triangle is aligned so that the top shows the direction of the reference line.

Please note:

If a reference identifier is positioned ***with reference*** to a line, it is also transformed if the line is transformed.

Process tolerance entry or reference identifier

 Alternatively, the command can also be loaded using this icon in the 2D toolbar.

This command can be used to process existing form and positional tolerances or reference identifiers in the drawing. The same command can also be used to re-position these objects.

1. If necessary, use the ***Access to ... partial drawing*** command at the top of the window to determine which partial drawings are to be available for selection.

2. Identify the finish entry to be modified.
The dialog box for the definition of the finish entry settings is then opened. It contains the values of the identified entry.

3. Determine the new settings.

4. If the position of the entry is not to be modified, confirm the modifications by clicking on the command button ***Close***.

If the position of the entry is to be modified, click on one of the positioning command buttons. Then define the new position in the same way as for inserting a finish entry.

Please note:

, Form and positional tolerances can also be disassembled using the command ***Process2D, Disassemble 2D Objects***.

Indicate a sectional display

, A sectional display of a 2D drawing can be identified using the ***Drawing Additions, Draw Sectional Display*** command.

Firstly, if necessary, define the required settings for the format of the sectional display and the intersection line using ***Settings, 3D/2D Commands*** and the ***Section view*** option card.

1. Once the command is started, begin with the intersection line on an object on which the sectional display is to begin. The intersection line is automatically carried out in 15° angle increments.
2. Draw the full and incremental section until the object comes round again.
3. Quit the definition of the sectional display by pressing the right mouse button.
4. If necessary, modify the suggested value for the format of the intersection line in the text boxes of the Status dialog box.
5. Define the position of the view direction arrows dynamically with the cursor and confirm using the left mouse button.

Please be aware, that view direction arrows can only be created parallel. The sectional display can only be used for full cuts and step cuts under 90°.

Please note:

, The sectional display can be edited using the command ***Process 2D***, ***Process Sectional View/Section Run***.

, Sectional displays can also be disassembled using the command ***Process2D, Disassemble 2D Objects***.

Process sectional view/section run

, The command ***Drawing Additions***, ***Process Sectional View/Section Run*** is started by clicking the displayed icons..

If an existing sectional display and/or the respective sectional identifier is to be processed, proceed as follows:

1. Activate the command ***Drawing Additions, Process Sectional View/Section Run***.
2. Identify the section view that is to be displayed according to these values, on one f its contour lines or identify the respective section identifier.
3. In the dialog box that appears, define the required values for the sectional display.
   In the option card ***Display*** changes can be made for the display of the section view. Deactivate the option ***Display according to global settings***, when object depending changes are made to the section view

Please note:

,  Sectional displays can also be disassembled using the command ***Process2D, Disassemble 2D Objects***.
The actual section-run identifier is retained by doing so, in order that any respective sectional view is not deleted.

4.13 Information concerning 2D objects

 This command can also be activated using this icon.

The commands in this toolbox can be used to display information concerning the 2D objects and carry out alterations to the parameters valid for these objects.

When measuring distances in the 3D window, the construction aids line is only created, if both identified points are in the current work plane.

Determine Solid from Model View

,  After clicking the displayed icons, the command ***Information***, ***Determine Solid from Model View*** is started.

It allows you to identify the associated body in the 3D area by identifying a line in a model view or a sectional view.

To do this, identify a line in a model or sectional view. If the 3D window is opened, it is activated and the corresponding body is highlighted and focused in the 3D window; According to the setting under ***Settings***, ***Options*** on the ***User Interface*** tab.

You can also use the following key functions within the command:

Ctrl-key

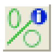
Press and hold the ***Ctrl*** key while identifying a line to reverse the current ***Centre / Zoom Centre*** setting in the ***Options*** dialog.

Shift-key

If you press and hold the ***Shift-key***, while identifying a line, all other solids in the 3D area are hidden.

 ,  Press the ***Start Hide Action*** or the ***Start Hide Others Action*** icon at the top of the 3D window to redisplay the solids.

Edit object

  After clicking on these icons, the command ***Information, Edit Object*** is started.

The command is for displaying the parameters valid for the 2D and 3D objects and then to edit them.

Identify the object.
Then a dialog box appears displaying the current data for the object.

line object

The information displayed for these objects in the dialog box is as follows:

The **area** text box shows the description of the current partial drawing or the current workplane, the **group** text box shows the group to which the object belongs. The content of this field cannot be edited.
The **layer** text box contains the name of the layer with which the object is linked.

After you have clicked  you will see a dialog box containing the display attributes of the object. You can change them.

All the other fields contain – depending on the type of object that is selected – the geometry values of the object. The main values, such as the coordinates for the starting point for a line, can be edited; others, such as the angle and length, will be determined by this entry.

Specifically for ***Symbols***, additional information is allocated concerning the reference point of the symbols, the defined rotation angle and the scaling of symbols in X and Y.

hatching

The ***settings for hatch*** or ***face*** dialog box is loaded. If necessary, it can be advantageous to deactivate the faces using ***Settings /***

Object Filter, in order to be able to edit the hatching using this command.

dimension

This will call up the command ***Dimension, Edit Dimension Properties***.

text

This will call up the command ***Text, Edit Text***.

addition

This will call up the command ***Addition, Process Finish Characteristics***; ***Process Form/Positional Tolerance*** or ***Detail View Parameter***

2D standard part

This will call up the command ***Process 2D Objects, Edit 2D Standard part***.

2D-shaft

This will call up the menu command ***Insert, Shaft****.*

solids

This will call up the menu command ***Process, Object Display***.

Model views

The ***2D object editor (model view)*** dialog box is loaded. Object dependend display properties can be changed here, for example to show or not show hidden lines.

section view

This will call up the command ***Process 2D Objects, Process Section View***.
The option card ***Format*** is used to change parameters for sectional displays. In the option card ***Display*** the display properties for section views can be changed.

detailed views

When identifying the frame of the original or the detail, the dialoguw window ***Detail Parameter*** is offered. The ***Scale Label***, the ***Reference Symbol*** and the ***Drawing Options*** can be changed here.

Information - 2D face

The objects that are of the type 2D face can have their area content, circumference, centres of gravity and moments calculated and then presented inside a dialog box; the results can be inserted in the drawing or saved in a file as well.

You can generate a 2D face with the commands ***Face Definition, Hatch face***, ***Fill Face*** or ***Define Face***.

Specifying settings for calculating 2D faces

The ***Settings, 2D Face Calculation*** command can be used to specify the settings required by the ***Information, 2D Face Calculation*** option.

In the upper section the ***units*** can be specified for calculating and for completing the output of the values.

decimal places

This entry will indicate the number of decimal places for rounding the values that are displayed.

preciseness of approximation

This value will have a direct effect on the precision applied in calculating faces enclosed by splines or elliptical arcs:
Such shapes will be approximated by lines. The value for the approximation will indicate the maximum permissible distance between the true contour of the line and each partial line. The lower this number, the greater will be the precision and the time required to complete such a calculation. A realistic value is, for example, 0.001.

Calculate 2D faces

The procedure for calculating the geometric values associated with 2D faces is described below:

Selection – Action

1. Accept the 2D faces, but no others, into your selection list.
2. Activate the command by clicking the icons ***Information, 2D Face Calculation***.

Action - Selection

1. If the selection list is empty, activate the command by clicking the icons ***Information, 2D Face Calculation***.
2. Identify an individual 2D face, or select several at once by holding down Ctrl at the same time.
3. The values for the 2D faces selected above will be presented in the ***2D Face Calculation*** dialog box.

Please note:

If no 2D face yet exists, a face can be generated in the ***Calculate***

2D Face command by keeping the mouse button pressed to drag open a rectangle over the geometry and clicking in the face to be calculated (or via collection with the Ctrl key for a number of faces). An object of type 2D-face is generated which acts as the basis for the face calculation.

Dialog box - 2D Face Calculation

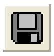
The sizes that have been calculated for the chosen 2D faces will be presented inside this dialog box.

In addition, you will have the following options:

Insert results in drawing

 After you have clicked the icon shown here, you will be able to insert the list containing the numerical values in the drawing as an individual object.
Please note:
If you work on this object with one of the commands ***Process 2D Objects, Disassemble 2D Standard Part*** or ***Disassemble 2D Objects***, you can then work on each of the text line in it separately.

Save results in file

 After you have clicked the icon shown here, you will be able to specify a directory and a file in the dialog box that is presented. Unless you specify a different one, the extension appended automatically will be GEO.

<u>Option card - General</u>

Show centre of gravity

 After you have clicked this icon you will see the common centre of gravity displayed inside the 2D faces chosen. It will be indicated by a marker.
Press any key to remove the marker again and to return to the dialog box.

Draw axis of inertia

 After you have clicked this icon, you will add the axis of inertia to the drawing. It will appear as the centre line.

<u>Option card – Relative to Coordinates</u>

Moments related to any coordinates

Type the coordinates for the point in the two lines XB and YB. It is this point to which the moments will relate.

After you have confirmed the coordinates by pressing Enter, the values will be shown on the option card.

Determine and insert face content

, After selecting the displayed icons, the ***Information, Determine and Insert Face Content*** command is started.

In the 2D window or in the current workplane of the 3D window, this command determines the face contents within a closed polyline to be designed and then offers this content as text which can then be positioned. The text is saved on the current workplane and is displayed with the properties defined for this plane. The text display is carried out according to the current text style. The number of decimal places and the unit can be defined using ***Settings, Calculate 2D Face***.
The closed polyline can be defined be making entries in the Status dialog box or by using the Snap Menu.

1. Generate a closed polyline from at least two partial lines by making entries in the Status dialog box or by using the Snap Menu or by using the Snap Menu to move about snap points of existing geometry.
   The start point of the first partial line is connected like a rubberband to the endpoint of the last partial line. A closed polyline is therefore generated.
2. Finish the generation of further partial lines by pressing the right mouse button.

The 2D face is then calculated within the closed polyline and offered as text for positioning.

Information - angle

, After clicking on these icons, the command ***Information, Angle 3 Points*** is started.

This command determines the angle between three points and displays the value in the Status dialog box.

1. Define the summit of the angle.
2. Define the endpoint of the first side of the angle.
3. Define the endpoint of the second side of the angle.
   The angle is determined in an anticlockwise direction from the first to the second side.

Information - Determine Distance

, After clicking on these icons, the command ***Information, Determine Distance*** is started.

The command determines the distance between objects and / or points and displays the value in the Status dialog box.

1. Identify a 2D line object or define a point.
2. Identify another 2D line object or define another point.
   The distance will be displayed in the status dialogue window as value and in the graphic as line.
   When pressing the ***SHIFT key*** during the identification a help line is created on the current layer for help lines.

Please note:

If you determine the distance between 2 points in 2D, the ***dX***- and ***dY*** Coordinates are displayed as well, in 3D also the ***dZ*** Coordinate. In 3D the coordinates are related to the global Coordinate system.

When determine the distance in 3D, the help line which is created when pressing the ***SHIFT key*** will only be created, when both identified points are on the current work plane.

When identifying two 2D line objects (Line, Circle, Ellipse, Spline) always the minimum distance will be determined.

, , Alternatively, the individual functions ***Distance Point - Point***, ***Distance Point - Line*** and ***Distance Line - Line*** can be activated using a user defined icon bar.

Information - point-point distance

This command can only be activated using a user defined icon bar.

This command determines the distance between two points also the ***dX*** and ***dY*** coordinates and displays the value in the Status dialog box.

If the distance of two points on 3D solids is measured, also the ***dZ*** coordinate is displayed. In 3D space the coordinates are calculated based on the global coordinate system.

1. Define both points.
   The distance will be indicated temporarily by a line.

Please note:

The temporary displayed line showing the distance can be created

as ***Construction Aids Line***, if the ***Shift*** key is pressed while selecting the second point or line.
When measuring distances in the 3D window, the construction aids line is only created, if both identified points are in the current work plane.

Information - point-line distance

This command can only be activated using a user defined icon bar.

This command determines the minimum distance between a point and a 2D line object (line, circle, ellipse, spline) and displays the value in the Status dialog box.

1. Define the point.

2. Identify the required 2D line object.
The distance will be indicated temporarily by a line.

Please note:

The temporary displayed line showing the distance can be created as ***Construction Aids Line***, if the ***Shift*** key is pressed while selecting the second point or line.
When measuring distances in the 3D window, the construction aids line is only created, if both identified points are in the current work plane.

Information - line-line distance

This command can only be activated using a user defined icon bar.

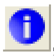
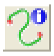
This command determines the minimum distance between two 2D line objects (line, circle, ellipse, spline) and displays the value in the Status dialog box.

1. Identify two line objects.
The distance will be indicated temporarily by a line.

Please note:

The temporary displayed line showing the distance can be created as ***Construction Aids Line***, if the ***Shift*** key is pressed while selecting the second point or line.
When measuring distances in the 3D window, the construction aids line is only created, if both identified points are in the current work plane.

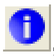
Information continuous line length

,  This command can also be activated using this icon.

This command determines the length of a line consisting of line objects (line, circle, ellipse, spline) and displays the result in the Status dialog box.

1. If the objects forming the continuous line are not in the selection list, take them over into the action list.
2. Define the starting point and end point on this continuous line for computing the length.
   The value that is determined will be displayed in the Status dialog box.

Check and clean a model

,  After clicking the displayed icons the command ***Information, Check / Clean Model*** is started.

This command checks the complete Model and offers a dialogue window with the results. If objects were found with the listed properties, the control button is activated. If all the objects of this type found in the model are to be deleted from it, tag the object type in its check box and then click the ***Remove*** command button.

Line length is 0
If the current model contains objects with a length below the computational accuracy, for example due to the import of objects, there may occur problems when editing such objects.

Remove empty groups
In addition you can use this command to delete empty groups from the model. The groups that contain neither subgroups nor objects will be deleted.

Faces with area 0
2D faces, which got lost when editing their surrounding contour elements, are displayed here and can be erased from the model.

Solids without geometric part
Solids, which have no geometric part any more (volume = 0), are displayed here and can be erased from the model.

Remove empty partial drawings
This command can also be used to remove empty partial drawings. All

empty partial drawings in all drawings in the current model are removed except one partial drawing and the Standard Sheet.

Empty Workplane
Empty workplanes in the 3D window -except of the three standard workplanes- are found and the number is listed then. To delete this empty workplanes, activate the control button and click the command button ***Remove***.

Dimension with lost associativity
Dimensions in model views, with lost references, due to changes in the 3D model, will be removed.

Break Out views without break line
The display of a break out view can be lost, if the depth is not dimensioned sufficiently or if the design has changed significantly. In this case the break out view can be deleted with this option and can be redesigned again.

4.14 Control display of 2D objects

The following options are available in *BeckerCAD* for determining the display attributes (colour, line type and line width) of 2D objects:

- layer specific display attributes
- object specific display attributes

In most cases the ideal solution is to use layers, because there will be a number of advantages:

- Layers containing the display attributes can be saved in templates so that they can always be used for creating a new model.
- The three display attributes will not have to be specified separately. They will be set together as soon as a layer is chosen.
- Layers can be used to control whether the objects linked by them are to be visible or hidden (e.g. for printing).
- Layers can be used to control whether the objects linked by them are "active" i.e. can be selected with the cursor (e.g. to prevent accidental manipulation or deletion).
- The settings made for the layers will be valid for all the drawings and their partial drawings as well as for all the construction planes in the 3D display window.

If, however, objects are allocated object-related attributes, they will take precedence over the attributes in the layer. The object related properties are stored with the object. Even when in between they were changed to layer related.

Please note:

If you use ***Settings, Drawing*** to examine the ***Status*** of partial drawings, the only objects to be displayed will be those in the active partial drawing in the colours associated with the objects or the layers. The objects in the other partial drawings will be displayed in the colour that has been specified for a particular status (***read only, without access***).

If your system has an Open GL graphics card available and it supports this option, 2D lines which are drawn in the 3D-view window in an even solid surface, are displayed so that they are not covered by these solid surfaces. This option can be activated using the menu command ***Settings, Options***.

Determine user-defined colours

2D and 3D objects can be allocated specific colours. They can be selected from a specified Windows colour palette.

In addition to the system colours there are 16 others that can be user-defined.

Use the following procedure to specify these colours:

1. Select the colour selection for solids in the 3D symbol bar, for example.
In addition, you can call up the colour selection from any of the dialog boxes.

Next, the Windows dialog box for colour selection will be displayed.

2. Click any of the fields beneath the heading ***Custom colours***.

3. If you want to modify one of the *basic colours* and take it over as your user-defined colour, click it.

Alternatively, place the cross-hair in the colour palette on the right of the dialog box.

4. Change the intensity and the red/green/blue component of the colour you have chosen by moving the slide control or by entering a number.

5. Confirm the setting by clicking ***Add to custom colours****.*

6. Repeat steps 3. – 5. for all the user-defined colours that you want and then exit the dialog box by clicking OK.

Please note:

If you save the user-defined colours in a template, you will be able to use them for any other model completed on the basis of the template.

If you leave the program after defining your colours, the settings will be saved in the file UIBECKER.SYS.

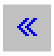
Determine user-defined line types

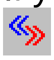
The standard number of line types available in *BeckerCAD* is 10.
You can add user-defined line types to these to display line objects: to do so you will have to determine their patterns.

You can use the individual objects dash, space, dot, circle, text or symbol for this.

Use this procedure:

1. Select the menu command ***Define, User-Defined Line Types***.

2.  Click the button ***Insert line type***. The list of line types will now contain a new type with the description ***Standard***.
   If you repeat this step, new line types will be added to the end of the list with the descriptions ***Standard_1***, ***Standard_2***, etc.

3. If you want to rename of a line type, select it in the list first.
    Then click the button ***Rename line type***, type the ***New name***, and confirm by clicking ***OK***.

4. Next select the new line type from the list, according to which of them is to be allocated a pattern.

5. After you have selected the new line type the first of the individual objects presented will always be the dash with the length 10 and the width 1.
   Type the length and width of the dash, or select a space in the list and type its length.
   You can type only whole numbers.

   Dashes and spaces form the "skeleton" of the line pattern: They are used for lengthening the line pattern.
   The positioning of the dots, circles, text and symbols is always with reference to the endpoint of the preceding dash or space.
   By using this method you can arrange several of these partial objects on a dash or in a space.

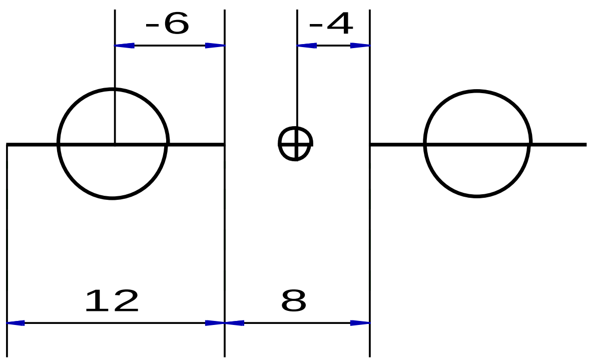
6. Click to add a new individual object to the one displayed in the ***current individual object*** field, or click to insert it in front of the current individual object.

   The two buttons and are for scanning forwards and backwards through all the individual objects that have been defined.

7. Define the parameters for the new object according to its type:

dash	length, width
space	length
dot	relative distance
circle	relative distance, radius, width
Text	relative distance, total width, height, letter distance
symbol	relative distance, library name, symbol name, scaling factor

   The value for the relative distance is with reference to the endpoint of the preceding dash or space, e.g.:



   If a value of > 0 is entered for the width, the partial elements, which have this width, are displayed. If the value 0 is entered, they are displayed with a width, which is set in relation to the layers or objects.

   When you are using symbols click the button in the symbols section to display a symbol library where you can select the symbol.

8. Define the factor for the ***scale*** in the section ***Line description*** so that the complete pattern can be extended or reduced.

9. You can activate the option ***fitted*** as required.
In this way you will be able to scale the lines automatically so that complete pattern of the line type will always be displayed on the line objects.

10. If you want to define more types of line, repeat steps 2. - 9. or 4. - 9. as often as required.

11. Conclude the line type definition procedure by clicking the ***Accept*** button.

If there is the same individual object at either end of the line pattern, such as a dash, a warning message will be displayed.
In such a case select the "extra" individual object as the current one and erase it by clicking .

Please note:

By saving the user-defined line types in a template you will be able to use them in other models based on the template.
If you leave the program after defining the line types, the settings will be saved in the file UIBECKER.SYS.

Save user-defined line types in file

User-defined line types are saved in templates and models.

In addition, you can save a list of line types in a line types file with the extension CFL.

Use this procedure:

1. Select the menu command ***Define, User-Defined Line Types***.

2. Refer to the section dealing with the determining user-defined line types to define the lie types.

3. Click the button in the section ***File selection***.

4. This will open a dialog box for you to enter the name and the folder, if required, for the line types file. Then confirm by clicking ***Save***.

Load user-defined line types from file

Once you have saved user-defined line types in a file, you can add them to the line types that you are currently defining.

Use this procedure:

1. Select the menu command ***Define, User-Defined Line Types***.

2. Click the in the ***File selection*** section.
3. Use the file selection to specify the file that you want to load and then click ***Open***.
4. The line type file will be loaded.
   If it does contain a line type that has a description identical with existing line types, you can decide whether or not to overwrite.

Please note:

CFL files can also be loaded using Drag & Drop.

Process user-defined line types

Use the following procedure when you want to modify a user-defined line type:

1. Select the menu command ***Define, User-Defined Line Types***.
2. Select the line types in the list, according to what you want to modify.
3. If you want to change the line type description, click the button ***Rename line type***, type the ***New name***, and then click *OK*.

   If you want to change the pattern for the line type, select it by clicking either or until you find the individual object.
   Use the procedures described in the section ***Determine User-Defined Line Types*** to change the parameters for existing partial objects, to insert new partial objects or to delete partial objects already in use.
4. Conclude the procedure by clicking ***Accept***.

   If the identical partial object is at both ends of the line pattern, such as a dash, a warning message will appear.
   In this case select the "extra" individual object as the current one and then delete it by clicking .

Erase user defined line types

Use the following procedure to delete a user-defined line type:

1. Select the menu command ***Define, User-Defined Line Types***.
2. Select the new line types in the list, according to what you want to delete.
3. Click to delete the line type.

4. Conclude the procedure for modifying line types by clicking ***Accept***.

Define control display attributes using layers

All 2D objects are drawn "on" layers. They are actually linked with the layers and their attributes.

Layers have the following influence options on 2D objects:

Display attributes of 2D objects

colour
line type
line width

The colour for the objects will probably help you to check on screen, whereas line types and the line width are vital for plotting drawings correctly.
You will be able to work more systematically, if you create template directories that contain a layer structure with fixed attributes and that have to be utilised by all members of staff.

In a similar way you can use the layers to control the visibility of 2D objects and the access to such objects:

Visibility of 2D objects
visible or hidden

Hidden 2D objects are not displayed on the screen nor taken into account when printing.
This setting will also have a direct bearing on the display of any solids.

Access options on 2D objects
active or inactive

2D objects which are inactive cannot be selected or identified with the cursor. In this way it is possible to control access to objects and protect them from modifications.
This setting will also have an effect on access to solids.

The general procedure for utilising display attributes based on layers

If the display attributes for the 2D objects is to be based on layers, use the following method:

1. Open a template in which you can save the layer structure.

2. Use the command ***View***, ***Model Explorer*** and change to the option card ***Layer Explorer.***.
   Generate a layer for all the required combinations of line type, line width and (if necessary) colour: then give the layers a description and the attributes that are required.

   Define a special layer for dimensions, hatches, centre lines etc. by selecting a layer and click on the related command button.

3. Select one of the layers that have been defined using the main symbol bar layer selection. This layer will be the ***current drawing layer***. If necessary, also activate the coloured layer symbol next to the display attributes to specify that these are to be allocated to the objects.

   All the 2D objects that are generated afterwards with the commands in the ***Draw 2D*** toolbar, will be linked to the current drawing layer.

   Certain 2D objects, such as centre lines, hatching, dimensions, will not be linked to the current drawing layer.
   These objects will, in each case, be linked to those layers that you have specified for them in the Layer Manager.

4. Save the template file with the layer structure that you have set up.

, The Layer Manager can also be used with ***Layer Scan*** to examine the contents of the layers. You can also use ***Cut*** and ***Paste*** to link all the objects in one layer with a second and to change such attributes as visibility and access.

You can obtain information on the layer allocation of individual objects as well or change them by using the menu command ***Process, Object Display***.

If you want to select or deselect objects on the basis of their layer allocation, use the various commands available in the toolbar, such as ***Select***.

Layer Manager in the Model Explorer

In the model explorer, to be opened using the menu command ***View, Model Explorer***, there is the option card ***Layer Explorer***.
Choosing this option card the following icons appear:

Layer Scan On / Layer Scan Off

Using the displayed icon the layer scan is turned on and off. In activated mode only the objects on the tagged layer are displayed.

Important: The scan mode is only valid for the tree structure not for the list view.

New Layer tagged / New Layer not tagged

If generating a new layer this new layer is tagged when this icon is active. Generating another new layer with the shortcuts ***Ctrl-N*** or ***Ctrl-n*** will sort this layer inside the first new layer and automatically offers the command to ***Rename***.

If this icon is not activated new layers are on the same level without offering the command to ***Rename***.

Localize Object in Model Tree

Start the command by clicking the displayed icon and identify an object in the current graphic window. The layer, related to the identified object, is marked (blue background) in the Structure Tree.

The other command buttons are used to define ***Layer as…*** for special objects.

Tree view active / List view active

If the ***Tree view*** is active then in the detail view only those layers are displayed which are one step beyond the selected layer.

If the ***List view*** is active all layers beyond the selected layer are displayed in the detail view.

Layer: Notes in the Model Explorer

The message box in the ***Model Explorer*** is not only used to output error messages, it can be used to store own notes with the model.

Using the command ***Clear message box*** system internal tipps and warnings, are deleted. User notes will no be deleted.

To delete all the texts in the message box, he new command ***Erase All*** is used. Single texts can also be handled with standard text procedures, e.g. Mark text and delete it using the delete key.

If texts where deleted by accident, an undo is available with the key combinations ***Ctrl+z*** or ***Ctrl+Shift+z*** , while the cursor is in the message box.

Context menu in the Layer Explorer

The layer context menu is used to process layers. It is opened by placing the cursor somewhere in the left side of the Layer Explorer and clicking the right mouse button.

Layer structure

The left section of the dialog box contains the structure of the layers as a hierarchy.

Below the basic layer with the description *Standard* there are the descriptions of the separate layers or. where applicable, branches of the layers. The symbols alongside the layer descriptions denote the object type for which these layers are used.

The layer branches are denoted by the symbols ⊞ ⊟ placed alongside. You can open a branch by clicking ⊞ and close it by clicking ⊟. If the layer is tagged, you can use the two keys + or – instead.

Commands in the context menu

Using the context menu in the Layer Explorer variuos settings can be made and also layer van be defined or erased. The following commands can be used in the context menu.

New: Generate layer (Strg+n)

Before layers can be used for drawing 2D objects, they must firstly be generated and defined their attributes.

Layers with their own names and attributes should be defined in a template. If saved in a template, all models generated on the basis of this template are available.

When starting *BeckerCAD* the template STANDARD.TPL is loaded. Some standard layers with their own display attributes have already been defined in this template. These can be used as stated, modified slightly to meet your requirements or changed completely.

The layer structure in *BeckerCAD* can be set up as a directory tree with different hierarchical levels.
For later processing of layers and the attributes allocated to them, this structure should be formed simply. The hierarchy of this structure should be kept with as few levels as possible.

1. Select the menu command ***View, Model Explorer***.
Choose the option card ***Layer Explorer***. Below which you want to generate a new layer. Then click the right mouse button.

If the layer structure is to remain simple, select the basic layer.

2. Select the command ***New..., New Layer (Ctrl+n)*** in the context menu.

   If layers are to be generated for dimensions, for which dimension text, dimension lines and dimension delimiters are to have different display attributes, select the *Dimension* option. Otherwise, select the command ***New dimensioning layer (Shift+Ctrl+n)***.
   If you do not need to make this distinction, you must select the command ***New Layer*** as well for dimensions.

   The new layer is added to the end of the layer list. Independent of the selected option, the new layer is added to the end of the list. It receives the name ***New_layer*** from the program. If a number of layers are generated one after the other, further layers receive the names ***New_layer0, New_layer1***, etc.

   When the command  ***Tag new layer*** is active, the description of the layer can be entered immediately.

   If the ***New Dimension Layer*** option has been selected, four layers for dimension text, dimension lines, dimension delimiters and tolerance entries are generated on the next hierarchical level.

Erase: Erase layer

When you want to erase a layer or all the objects in a layer, or want to complete both these actions, use the following method:

1. Select the menu command ***View, Model Explorer***.
   Choose the option card ***Layer Explorer***
2. Move the cursor to the layer structure and then tag the layer to be dealt with. Then click the right mouse button.
3. Next, choose one of the following commands:

   Erase, Erase layer only (Del.)
   This action will delete the layer. The objects linked to this layer will now be linked to the higher up in the layer structure hierarchy.

   Erase, Erase geometry only (Ctrl+Del.)
   This action will delete all the objects that are linked with the layer.

   Erase, Erase layer and geometry (Shift+Ctrl+Del.)
   This action will delete the layer and all the objects that are linked with it.

   Delete, Delete empty layers (Alt+Del.)
   All layers are deleted, which do not contain any object. To delete

all empty layers in the model place the cursor on the base layer ***Standard*** and choose the command ***Delete, Delete all empty layers*** from the context menu.

Please note:

The basic layer cannot be deleted.

Multiple layers can be deleted by tagging them in the list view and use the ***Del*** button.

,  If you want to delete objects linked to one layer or to several layers, you can do so with the menu command ***Select, Select Using Layer*** and then execute the command ***Erase Object/Selection***.

Using the command **Delete empty layers** no layers are deleted which are defined for special objects.

Properties: Define display attributes of layer (Shift+F2)

Objects (2D objects) linked with a layer are displayed with the colour, line type and line width which have been defined for the current layer.

This does not apply to objects with display attributes specified as related to objects.

Line type and line width are ignored with the Truetype fonts.

Display attributes can be specified for one layer or several layers or also changed by using the following method:

1. Select the menu command ***View, Model Explorer***.
Choose the option card ***Layer Explorer***

2. **Processing single layers:**
Tag the layer and click the right mouse button. Select the command Properties ***(Shift+F2)*** in the context menu or double click the layer in the list view.

Processing several layers
When you want to allocate the identical attributes to several layers, press down *Ctrl* or *Shift* and tag them in the right section of the ***Layer Explorer***.
Click the right mouse button and select ***Attributes*** in the context menu.

3. Select the ***colour*** and ***line type***, and enter a value for the ***width*** in the box for the line width.

When you select a user-defined line type, click the line in the line types list for user-defined lines, and then select the line type in the dialog box.

4. Confirm the modifications by clicking on the ***Accept*** command button.
If there is already a link between the layer(s) and 2D objects with layer-related attributes, they will be re-drawn using the display attributes that have been specified.

5. They attributes for other layers can be specified by repeating Step 2 to Step 4.

Please note:

If the display attributes are to be modified - independent of the layer, use the menu command ***Process, Object Display***.

The display of the Windows system line types can be influenced, using the menu command ***Settings, Options*** and entering a factor in the ***scale line pattern with factor***, by which the pattern length is to be enlarged or reduced. The defined value is stored separately for the current graphic window.

Use the menu command ***Settings, Options*** to determine whether lines are to be displayed with the allocated dimensions in relation to the dimensions of the drawing sheet or with a standard line width.

In the 3D-Window user-defined line types are displayed as a continuous line.

Process: Rename layer (F2)

Layers can be given any clear name.

1. Select the menu command ***View, Model Explorer***.
Choose the option card ***Layer Explorer***.

2. In the list of previously defined layers, click on the name which is to be modified.

3. Enter the key ***F2*** or or select the option ***Process***, ***Rename*** in the context menu. A text box will appear so that you can type the name.
Complete the procedure by pressing *Enter*.

Please note:

If drawings are to be converted in to DXF or DWG format, the layer names should only contain the characters A-Z, 0-9, -, _. It must also be noted that the DXF/DWG format does not support the

BeckerCAD differentiation between upper case and lower case characters.

Process: Search layer (F3)

A layer can be searched in the Layer Explorer.

1. Select the menu command ***View, Model Explorer***.
Choose the option card ***Layer Explorer***.

2. Click in the Layer Explorer and enter the key ***F3*** or select ***Process***, ***Find*** in the context menu.

3. Enter the search string in the dialog box. The search is started clicking the command button ***Find next***. Every click on the command button will show the next search result.

Match whole text only
There must be no further letters in front of and behind the search text.

Match case
The search is case-sensitive.

Match regular expression
With regular expressions, any complex search queries are possible. With special characters, search terms can be put together so that not only a single term is found, but also precisely defined combinations of terms.
Examples for the search with regular expressions can be found at the end of this chapter.

Wrap around
The search is continued at the beginning of the list up to the selected element.

Siblings only
The search is restricted to the current structure level within the selected structure tree.

Examples for regular expressions

For regular expression, the search text is expanded using simple controls or control elements. Here are some examples, which can only give a small overview of the entire spectrum of the search with regular expressions.

Character	Meanings	Example
.	Represents exactly one character	.ower = tower, lower
\w	An alphanumeric character	aAbBcC …
\d	A digit	0123456789
\d+	A digit sequence	
*	No or several repititions	The precending search expression can not occur once, or occur once or several times
+	At least one repitition	The precending search expression can occur once or several times
[ ]	Character class Matches all characters within the brackets. Areas can be specified with the hyphen	[abc] finds "a", "b" and "c". [a-z] finds a lower character. [0-9A-Z] finds a digit and an upper character
( )	Describes a sequence of characters	Grouping of characters and search operators

Process: Replace (F4)

The descriptions of layers can be replaced.

1. Select the menu command ***View, Model Explorer***.
   Choose the option card ***Layer Explorer***.
2. Click in the Layer Explorer and enter the key ***F4*** or select the command ***Process***, ***Replace*** in the context menu.
3. Enter in the text box ***Search for:*** the text to be found. Enter in the text box ***Replace by:*** the new text. Click on the command button ***Search***, ***Replace*** or ***Replace all***.
   The search is restricted to the beginning of the word. Wildcards are not allowed.

Process: Update (F5)

The display of the Layer Explorer can be updated.

1. Select the menu command ***View, Model Explorer***.
   Choose the option card ***Layer Explorer***.
2. Click in the Layer Explorer and enter the key ***F5*** or select the command ***Process***, ***Update*** in the context menu.

Cut and Paste: Tag all (Strg+A)

All layers in the Layer Explorer can be tagged in the list view for example to move the structure, to edit or delete all layers.

- by defining a rectangle over all layers with the pressed mouse button.
- by the key combination ***Crtl+A*** in the list view.
- by selecting the option ***Cut and Paste, Tag all***.

Cut and Paste: Cut (Ctrl+X) / Copy (Ctrl+C) / Paste (Ctrl+V)

To move or copy layers in the Layer Explorer follow these steps:

1. Select the menu command ***View, Model Explorer***.
Choose the option card ***Layer Explorer***.

2. Tag in the list view the layer(s) to be moved or copied.

3. To move the layers enter ***Ctrl+X*** or select ***Cut*** in the context menu.
To copy the layers enter ***Ctrl+C*** or select ***Copy*** in the context menu.

4. Tag the layer in the Layer Explorer where the layers should be pasted.

5. Enter ***Ctrl+V*** or ***Paste*** in the context menu.

Please note:

Layers can also be moved / copied using the standard Windows Drag&Drop command.

Cut and Paste: Move geometry between layers (Shift+Ctrl+V)

To move geometries from one layer to another follow these steps:

1. Select the menu command ***View, Model Explorer***.
Choose the option card ***Layer Explorer***.

2. Tag in the list view the layer(s), which contains the elements to be moved.

3. enter ***Ctrl+X*** or select ***Cut and Paste, Cut*** in the context menu.

4. Tag the layer the elements should be inserted to.

5. Enter ***Shift+Ctrl+V*** or select the option ***Cut&Paste*** or ***Geometry***, ***Insert Geometry***.

The objects gets the display properties from the new layer, except when they have object related display properties.
If the new layer has the property not visible, the objects are not displayed when inserted.

Geometry: Move selected geometry to layer

Selected objects canbe moved to a layer. Hereby selected elements on different layers are moved to one layer.

1. Select 2D or 3D objects.

2. Select the menu command ***View, Model Explorer***.
Choose the option card ***Layer Explorer***.

3. Tag the layer the selected objects should be moved to.

4. Enter ***Strg+M*** or select ***Geometry***, ***Insert Geometry*** in the context menu.

Please note:

Selected geometry can also be moved to another layer using the menu command ***Process, Object Display***.

 Click the displayed icon and select the layer the elements should be moved to.

Geometry: Erase geometry on a layer (Strg+Entf)

To erase the complete geometry on a layer follow these steps:

1. Select the menu command ***View, Model Explorer***.
Choose the option card ***Layer Explorer***.

2. Tag the layer on which the objects should be deleted.

3. Enter ***Ctrl+Del*** or select the command ***Geometry, Only Erase Geometrie***.

Geometry: Select using layer (Shift+Strg+A)

To select all objects on a layer follow the next steps:

1. Choose from the menu ***View, Model Explorer***.
Change in the ***Model Explorer*** to the tab ***Layer Explorer***.

2. Select the layer assigned to the objects to be selected.

3. Press ***Shift+Strg+A*** or choose from the context menu on the selected layer ***Geometry, Selected layer geometry***.

Note:

,  The content of layers can also be selected using the command ***Select, Select using layer***.

 To deselect the objects the command ***Reselect Object*** can be used.

Display of the list view in the Layer Explorer

It can be defined whether in the list view all layers should be displayed or only assigned layers or only empty layers.

1. Select the menu command ***View, Model Explorer***.
Choose the option card ***Layer Explorer***.

2. Select the layer in the structure tree which is on top of the layers to be displayed as empty or used layers.

3. / Select between Tree view and List view. If the ***Tree view*** is active then in the detail view only those layers are displayed which are one step beyond the selected layer.
If the ***List view*** is active all layers beyond the selected layer are displayed in the detail view.

Note:
To switch the display between all *empty layers* and all *used layers* select the base layer Standard and choose the ***List view***.

4. Using the context menu in the list view select the option ***Display, ...***

5. Select between the following possibilities:

Show all layers (Ctrl+A)
All layers are displayed which are below the tagged layer in the Layer Explorer.

Show used layers only (Ctrl+B)
All layers assigned to geometry are displayed which are below the tagged layer in the Layer Explorer.

Show empty layers only (Shift+L)
All empty layers are displayed which are below the tagged layer in the Layer Explorer.

Layer:Sort Layer Explorer

Using the ***Format, Sort*** option in the Layer Explorer opens another menu, which contains many different sort criteria.

Open the context menu in the layer structure tree or the detail view. Choose the option ***Format, Sort*** and then:

- **Sorting On/Off (F6)**
Turns the sorting on or off.

- **Toggle sort direction(F7)**
  After pressing the function key ***F7*** or after selecting this command the structure tree or the detail view is sorted ascending or descending if the sort is turned on.
- **Sort case sensitive (F8)**
  After pressing the function key ***F8*** or after selecting this command the sorting is case sensitive or not if the sort is turned on***.***
- **Sort logical (Strg+F8)**
  With this command the sorting follows logical criterias. An example for a logical sorting is the sorting of single and double digits numbers. Usually a zero must be added in front of the single digit numbers to get the correct result. With the logical sort this numbers are sorted correctly.

Settings for displaying the Layer Explorer

Using the context menu ***Format*** different settings can be made for the display of the Layer Explorer. The options are depending if the context menu is called in the layer tree or in he list view.

- **Open / close branch (F11)**

  This command can be used to open or close the selected branch in the tree structure including all the subordinate structures.
- **Open all / close all (Shift+F11)**
  This command can be used to open or close all branches of the entire layer area right up to the lowest structure.
- **Tile horizontal / vertical (Strg+L)**
  This command defines if the structure tree and the list view are tiled horizontally or vertivcally.
- **Toggle message splitter (Shift+Strg+L)**
  This option defines the positon of the message box to be beside or below the layer structure tree.
- **Extended text display (Strg+S)**
  This command can bre used to define if the column headlines are shortened or completely written.
- **Display of columns in the list view**
  This command is used to hide or display columns in the list view. Click in the context menu on the column to be displayed or hided. The setting is stored when leaving the program.

  The sequence of the columns can be changed by clicking on a column and move it to the new position with the pressed mouse

button.
The width of a column can be changed by clicking between two column and moving the cursor with the pressed mouse button.

Define visibility of layers

Layers can have the attribute visible or hidden. This will also have a bearing both on the 2D objects.

Objects and solids which are linked to layers which have the attribute as hidden, are not displayed on the screen and are not printed.

In one way, parts of the drawing which are not required at the current time can be suppressed, e.g. hatchings, and when the drawing is redrawn, the process is faster.

In another way, the parts of the drawing to be printed can be controlled using visible or hidden.

You can specify or change the visibility of any layer(s) by using the following method:

1. Select the menu command ***View, Model Explorer***.
Choose the option card ***Layer Explorer***.

2. **Processing single layers**
Double click the icon  on the layer to be hided.
When double clicking the icon  the objects on the layer are visible again – but not active also.

or

Double click the name of the layer in the list view that you want to deal with to open the dialog box for processing attributes.
You can also click the right mouse button to get the context menu and then select the command ***Attributes***.

Processing several layers
When you want to make several layers visible or hidden, press Ctrl or Shift and tag them in the right-hand section of the dialog box.
Click the right mouse button on one of the tagged layers and select ***Attributes*** in the context menu.

3. Deactivate or activate the ***visible*** setting.

4. Confirm the modification by clicking on the ***Accept*** command button.
If the setting is deactivated, the objects of this layer will be erased

from the screen.
If the setting is activated, the objects are displayed.

Layers can also be faded in or out using the context menu in the layer structure tree in the Model Explorer.
When opening the context menu by clicking the right mouse button on a layer, the following settings can be made in the option ***Visibility/Pickability***.

Context menu

in the structure tree	in the detailed view	key command
Toggle visibility	Toggle visibility	O
Toggle pickability	Toggle pickability	P
Toggle visibility and apply to subtree		Strg+Alt+O
Toggle pickability and apply to subtree		Strg+Alt+P
Enable visibility of tree	Enable visibility	Strg+O
Disable visibility of tree	Disable visibility	Shift+Strg+O
Enable pickability of tree	Enable pickability	Strg+P
Disable pickability of tree	Inaktiv	Shift+Strg+P

Please note:

Layers for which the visible setting has been deactivated, will be allocated the inactive attribute (***active*** setting deactivated).

If the layer is made ***visible*** again, the inactive setting is retained.

If a layer which has previously been defined as hidden is to be visible and active at the same time, the ***active*** check box should be activated.

Using the menu ***File, Read/Write Parameters*** the properties for existing layers can be read from an MPS file. Precondition for that is that the layers have identical names.

Define access options of layers

Layers can be allocated the *active* or *inactive* attribute. This will also have a bearing both on the 2D objects as well as the solids.

2D objects or solids which are linked to layers which have the inactive attribute, cannot be selected using the cursor.

In this way, it is possible to limit the selection of objects. In the same way, objects can be protected against modifications.

The access options relating to any layer(s) can be specified or changed by using the following method:

1. Select the menu command ***View, Model Explorer***.
Choose the option card ***Layer Explorer***.

2. **Processing single layers**
Double click the icon ✓ on the layer to be deactivated. When double clicking the icon ✗ the layer is active again.

or

Double click the name of the layer that you want to deal with to open the dialog box for processing attributes.
You can also click the right mouse button to get the context menu and then select the command ***Attributes***.

Processing several layers
When you want to allocate several layers the identical access options, press Ctrl or Shift and tag them in the right-hand section of the dialog box.
Click the right mouse button and select ***Attributes*** in the context menu.

3. Deactivate or activate the ***active*** setting.

4. Confirm the modifications by clicking on the ***Accept*** command button.
Layers which were previously hidden, will also be allocated the ***visible*** attribute when allocating the ***active*** setting.

Layers can also be activated or deactivated using the context menu in the layer structure tree in the Model Explorer.
When opening the context menu by clicking the right mouse button on a layer, the following settings can be made in the option ***Visibility/Pickability***.

Context menu

in the structure tree	in the detailed view	key command
Toggle visibility	Toggle visibility	O
Toggle pickability	Toggle pickability	P
Toggle visibility and apply to subtree		Strg+Alt+O

Toggle pickability and apply to subtree		Strg+Alt+P
Enable visibility of tree	Enable visibility	Strg+O
Disable visibility of tree	Disable visibility	Shift+Strg+O
Enable pickability of tree	Enable pickability	Strg+P
Disable pickability of tree	Inaktiv	Shift+Strg+P

Note:

Using the menu ***File, Read/Write Parameters*** the properties for existing layers can be read from an MPS file. Precondition for that is that the layers have identical names.

Determine layers for special objects

All 2D objects which have been generated using the CAD Menu, ***Draw 2D*** menu, are linked with the current drawing layer.
In order to link these geometry elements with different layers (e.g. because they are to have different line widths and line types when plotting the drawing), one of the previously defined layers must be determined as the *current layer*.

For other 2D objects such as text, dimensions, hatchings, etc. such differentiation is not required as such or at all. One layer can be defined for such objects, with which the objects are automatically linked when generated.
Therefore, hatch lines are not linked to the current layer when generated, but are linked to the hatch layer, text is linked to the text layer, etc.

1. Select the menu command ***View, Model Explorer***.
Choose the option card ***Layer Explorer***.
2. Tag the layer that is to be reserved for special objects.
3. Select from the the grey command buttons in the top which special objects should be assigned to this layer.

   Current drawing layer

   Texts

   Construction aids (green symbol)

   Hatching

Dimensions

Centre lines

Hidden lines (2D), i.e. standard parts and disassembled model views

Hidden edges (3D), i.e. model views

Standard parts thread lines

Tangential Edges for example in model views

Please note:

A layer, chosen for multiple object types, is marked with the displayed icon.
If the layer, chosen for dimensioning, does not consist of 4 sub layers also this layer is marked with the displayed icon.

All layers, not chosen for any object type, are marked with the displayed icon.

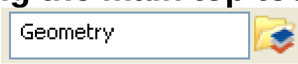
If e.g. text is to be linked to various layers in order to plot in various colours or line widths, generate the required layers and allocate them with the required names, e.g. *text_05* and *text_07* etc.
Before you begin to add the text, choose one of the layers in the ***Layer Explorer*** dialog box and then specify it as the ***Text Layer***.

Select current drawing layer

Before using the command in the ***Draw 2D*** menu, the layer should be determined with which the 2D objects to be generated will be linked.

The following options are available:

Using the main top toolbar



1. In the main top toolbar, click on the list box for layer selection.
2. In the dialog box which appears, click on the required layer. Confirm with ***OK***.

   The selected layer becomes the current drawing layer and the display attributes for this layer are then displayed in the colour selection icon, line type list box and line width text box in the main top toolbar:

This icon to the right of the above mentioned icons or text boxes show the symbol *layer specific*.

Using the main top toolbar

1. In the main top toolbar, click on the icon to the right of the displayed layer selection text box.
2. Identify a 2D object.
   The layer on which this 2D object lies becomes the current drawing layer and its description is then displayed in the layer selection text box. The fields for layers election, line type selection and line width definition show the display attributes that are linked with this layer, if the layer reference is switched on.

The icon to the right of the named fields each show the ***layer specific*** icon, if the icon is colour highlighted.

Using menu commands

1. Select the menu command ***View, Model Explorer***.
   Choose the option card ***Layer Explorer***.
2. Move the cursor into the layer structure and then tag the layer that is to be reserved as the drawing layer.
3. Select the command button **GEOMETRY** at the top of the Layer Explorer. The selected layer is determined with the icon .

Please note:

The current drawing layer is also used for solids and the frames of pixel drawings which have been inserted in the drawing using the menu command ***Extras, Pixel Drawing, Insert***.

Display layer content

If the 2D objects are to be displayed which are linked to individual layers, therefore by "paging" through the drawing layers - so to speak, proceed as follows:

1. Select the menu command ***View, Model Explorer***.
   Choose the option card ***Layer Explorer***.
2. Click the command button ***Scan layer off.***
3. In the layer structure click the name of the layer containing 2D objects you want to see, or use the cursor control keys.

   The only objects that will be displayed are those linked to the layer that is currently tagged.

Closed node points 

can be opened using the arrow right key during the scan. Opened node points can be closed using the arrow left key.
The command ***Layer Scan*** stays active, also when closing the Model Explorer.

Please note:

You can use ***Display, In Use*** when in the right-hand section of the dialog box to display and check just the layers linked to objects.

Modify layer link of 2D objects

If, whilst drawing, you have failed to select the "correct" layer for the 2D objects to be generated, the current link can be modified by using one of the methods described below:

Free selection

When you use this method, you can select any objects and link them with a new layer.

1. Load the menu command ***Process, Object Display***.

2. If there are any objects in the selection list, the action relates to these objects.
If the selection list is empty, transfer the 2D objects into the action list so that they can be dealt with.

3. If the selected objects are linked with different layers, the *layer* text box has the entry <?>.
If the objects are linked with the same layer, the respective name is displayed.
Open the layer selection using the icon to the right of the *layer* text box.

4. In the list of layers, click on the name which the 2D objects are to be linked with. Confirm with ***OK***.

5. Confirm with ***OK***.

Selecting a layer

With this procedure you can link all the objects in a layer with a new layer.

1. Select the menu command ***View, Model Explorer***.
Choose the option card ***Layer Explorer***.

2. Move the cursor into the layer structure and then tag the layer that is to be linked a new one and enter ***Ctrl+X*** or select the command ***Cut&Paste***, ***Paste*** in the context menu.

3. Move the cursor to target layer in the layer structure. Then click the right mouse button.
4. Enter ***Ctrl+V*** or select the command ***Cut&Paste***, ***Paste*** in the context menu.

Move layers

Layers can be set up structured in the Layer Manager. The dimensioning layer for example, is automatically generated with sublayers. If you later wish to modify the structure of the layer tree, you can move layers in the structure. Proceed as follows:

1. Select the menu command ***View, Model Explorer***.
   Choose the option card ***Layer Explorer***.
2. Position the cursor in the left half of the Layer Manager on the required layer in order to move a layer or, in the right half of the layer structure, tag a number of layers that you wish to move in the structure and confirm the right mouse button on one of the tagged layers.
3. Click on one of the tagged layers and keep the mouse button pressed.
4. Move the layer(s) in the structure tree on the layer, where it should be inserted.

Define layer change for special objects

All generated 2D objects are linked with respective type specific layers. In order to link these geometry elements with different layers using the ***Draw*** CAD Menu commands (e.g. because they are to have different line widths and line types when plotting the drawing), one of the previously defined layers must be determined as the ***current layer***.
For other 2D objects such as text, dimensions, hatchings, etc. one layer can be defined for such objects, with which the objects are automatically linked when generated. To link an object type to a defined layer the ***Layer Explorer*** inside the ***Model Explorer*** can be used or special commands in the ***toolbar***.

Change layer type

The ***Change layer type*** command allows for the type specific change of layers for the individual allocation of 2D objects.

1. Select the menu command ***View, Toolbar***.

2. In the toolbox list, tag a toolbox in which the command is to be integrated or define a new toolbox. *(also refer to: User defined symbol bar)*
3. In the command tree, open the command area ***General CAD Functionality*** and the commands are below this – ***Draw 2D linear object***.
4. Tag the command ***Change layer type*** and then select the command button ***<<Add>>.***
5. In the command area in the *Argument* text box, enter the name of the required type and the required layer separated by spaces.

   E.g. to define the layer with the name ***Text*** as the current layer for ***Text 0.5*** , enter:

   "Text 0.5" TEXT

   According to the layer types allocated in the program, the following declarations are available for selection:

GEOMETRY		current drawing layer
DIM		dimensions
HATCH		hatching
TEXT		text
HELP		construction aids
CENTER		centre lines
WINDING		thread lines
HIDDEN		hidden lines (2D)
GRF2SOLID3_HIDDEN		hidden edges (3D)

Please note:

The layer description should generally be carried out in speech marks, otherwise layers whose names contain spaces cannot be recognised correctly. It is imperative that the correct spelling is retained for the layer name (capital/small letters). If no type declaration is given, therefore only the layer description is entered as the argument, only the current drawing layer will be changed when executing the command.

6. Allocate a separate icon to the command, by entering the name of an ICO file in the ***Symbol*** text box or by selecting an ICO file using the command button ***Open ICO file***.

Please note:

When entering a name for the symbol, there must be an ICO file of the same name in the folder currently defined for icon files (refer to ***Settings***; ***Folder***)

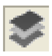
7. Save the modified configuration of your symbol bar using the command ***Save Toolbar Configuration*** in the ***Configuration*** section.
8. After taking over the modified symbol bar, the command ***Change Layer*** **>> define Text0.1 as current text layer** is available.

Define object specific display attributes of 2D objects

2D objects can be allocated an individual colour, line type and line width - independent of the display attributes of the layer.

Determine colour

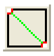
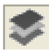
Using the ***colour selection*** icon in the main top toolbar, select the colour in which the 2D objects next generated are to be displayed.

 By selecting a colour, the colour allocated to the current layer becomes inactive. The icon which is displayed to the right of the colour icon in the main top toolbar, shows that the colour is *object specific*.

If the colour of the objects to be generated using the commands for ***Draw 2D*** is to be taken over from the current drawing layer again, click on the icon to the right of the colour selection icon. This icon then changes to the symbol *layer specific:* 

Please note:
If display properties like colour, line type or line thickness are assigned to 2D objects, those display properties are saved with the objects. Even when the display properties are changed to be assigned to the layer properties they are not lost and are reassigned when changed to assignment by object again.

Example:

1. 
   Chosse a different colour and line type than the actual one.
2. ,  Then draw a rectangle for example.
3.  Afterwards assign the layer properties using the menu command ***Process, Object Display*** by clicking on the grey icon which is then shown in colour. The layer properties now are assigned to the rectangle.
4. Then click again on the coloured icon and the rectangle is displayed in the previously assigned object display properties.

This also is valid for the display properties of the model views:

,  If display properties, different from the standard values, are assigned to model views using the command ***Information, Edit Object*** they are also saved with the model views even when activating the ***View according to global settings***.

Determine line type

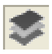
Using the ***line type selection*** icon in the main top toolbar, select the line type in which the 2D objects next generated are to be displayed.

When you select a user-defined line type, click the line in the line types list for user-defined lines, and then select the line type in the dialog box.

Please note:

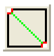
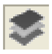
The display of the Windows system line types can be influenced, using the menu command ***Settings, Options*** and entering a factor in the ***scale line pattern with … factor***, by which the pattern length is to be enlarged or reduced.

In the 3D-Window user-defined line types are displayed as a continuous line.

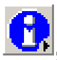
 By selecting a line type, the line type allocated to the current layer becomes inactive. The icon which is displayed to the right of the line type icon in the main top toolbar, shows that the line type is *object specific*.
If the line type of the objects to be generated using the commands for ***Draw 2D*** is to be taken over from the current drawing layer again, click on the icon to the right of the colour selection icon. This icon then changes to the symbol *layer specific:* 

Please note:
If display properties like colour, line type or line thickness are assigned to 2D objects, those display properties are saved with the objects. Even when the display properties are changed to be assigned to the layer properties they are not lost and are reassigned when changed to assignment by object again.

Example:

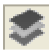
1. 
   Choose a different colour and line type than the actual one.
2. ,  Then draw a rectangle fo example.
3.  Afterwards assign the layer properties using the menu command ***Process, Object Display*** by clicking on the grey icon which is then shown in colour. The layer properties now are assigned to the rectangle.
4. Then click again on the coloured icon and the rectangle is displayed in the previously assigned object display properties.

This also is valid for the display properties of the model views:

,  If display properties, different from the standard values, are assigned to model views using the command ***Information, Edit Object*** they are also saved with the model views even when activating the ***View according to global settings***.

Determine line width

Using the ***line width selection*** icon in the main top toolbar, enter the line width in which the 2D line objects next generated are to be printed.

 By entering a line width, the line width allocated to the current layer becomes inactive. The icon which is displayed to the right of the line width text box in the main top toolbar, shows that the line width is *object specific*.

Please note:
Use the menu command ***Settings, Options*** to determine whether lines are to be displayed with the allocated dimensions in relation to the dimensions of the drawing sheet or with a standard line width.

If the line width of the objects to be generated using the commands for *Draw 2D* is to be taken over from the current drawing layer again, click on the icon to the right of the line width text box. This icon then changes to the symbol *layer specific:*

If display properties like colour, line type or line thickness are assigned to 2D objects, those display properties are saved with the objects. Even when the display properties are changed to be assigned to the layer properties they are not lost and are reassigned when changed to assignment by object again.

Example:

1. 

   Chosse a different colour and line type than the actual one.
2. , Then draw a rectangle fo example.
3. Afterwards assign the layer properties using the menu command ***Process, Object Display*** by clicking on the grey icon which is then shown in colour. The layer properties now are assigned to the rectangle.
4. Then click again on the coloured icon and the rectangle is displayed in the previously assigned object display properties.

This also is valid for the display properties of the model views:

, If display properties, different from the standard values, are assigned to model views using the command ***Information, Edit Object*** they are also saved with the model views even when activating the ***View according to global settings***.

Process display of 2D-objects

If, whilst drawing, you have failed to select the "correct" display attributes for the 2D objects to be generated - independent of whether these are layer specific or object specific - they can be modified as follows:

With selection list

1. Accept the 2D objects to be processed in the selection list.
2. Load the menu command ***Process, Object Display***.

Without selection list

1. Load the menu command ***Process, Object Display***.

2. Accept the 2D objects to be processed in the action list.

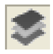
The rest of the procedure is the same in both cases:

The ***Object Display*** dialog box is displayed.

3. If the action or selection list contains solids, change to the ***2D objects*** option card.

If the selected 2D objects have different colours, line types or line widths, the respective text boxes contain the entry <?>.

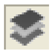
If all of them have the same display attributes of the layer with which they are linked, the symbol for *layer specific* is displayed to the right of the respective text boxes: 

If at least one of the selected 2D objects has display attributes which are object specific, the symbol for *object specific* is displayed: 

The following options are available for the modification of the display:

Allocate object specific display attributed:

Determine the colour, line type and/or line width.

All selected 2D objects are allocated *object specific*  with the determined display attributes.

Allocate layer specific display attributes:

If, after colour, line type and/or line width, the  symbol is activated, the selected 2D objects are displayed according to the display attributed of "their" layer.

Link objects with other layer

By selecting a layer, all selected 2D objects are linked with another layer. Their display depends on whether they are defined layer specific or object specific.

Modify drawing order

The drawing order (-100 ... 127) is used for defining the priority of a 2D object during drawing. In this procedure objects with a higher priority are above those with a lower one.

When you are drawing, "normal" objects will be allocated a drawing order of 0.
Clipped faces will have 60 as their drawing order, objects not to be hidden during their definition will have the drawing order 120.
Dimensioning is also given the drawing order 120.

Please note:

When you have called up ***Process 2D Objects, Hide Lines/Partial Lines***, the lines that are selected or partial lines will be allocated the specified display attributes.

The display attributes for dimensioning can only be changed with the command ***Dimension, Edit Dimension Properties***. Alterations to line types and line widths have no effect on the annotations and dimension text.

If display properties like colour, line type or line thickness are assigned to 2D objects, those display properties are saved now with the objects. Even when the display properties are changed to be assigned to the layer properties they are not lost and are reassigned when changed to assignment by object again.

In the 3D-Window user-defined line types are displayed as a continuous line.

4.15 Using groups

Groups are hierarchically arranged structural entities within a model. Although they are used above all to save 2D objects, you can also save 3D objects in them. In other words, groups can be used to structure the objects found in a model in the most logical way.

If you do not want to use groups for this purpose, you can use them to facilitate your work on the objects. All the objects belonging to a specific group can be selected and deselected together.

If you do not want to use groups for this purpose, you can use them to facilitate your work on the objects. All the objects belonging to a specific group can be selected and deselected together and groups, like layers, can be switched to active or not active, visible or hidden.

Groups are managed in the ***Model tree*** of the ***Model Explorer*** to be accessed via the menu ***View, Model Explorer***. To select objects or groups the selection modes ***Standard selection*** and ***Group selection (Shift key)*** in the main icon bar can be chosen.

Using the ***Standard selection*** by cursor in a drawing or in the 3D window and pressing the ***Shift key*** the objects in a group and all subgroups are selected; by pressing the ***Alt key*** the group and all subgroups are selected.

Using the ***Group selection*** mode by cursor and pressing the ***Shift key*** a context menu is offered to select:

- the objects in the group and all subgroups

- the group and all subgroups
- the objects in higher groups and all subgroups
- the groups and all subgroups in higher structure levels

The selection is displayed in colour.

, ,  This context menu also appears in the ***Group selection*** mode in the commands ***Select, Select Group*** respectively ***Deselect Group.***

2D and 3D objects are managed in different group structures:

Groups for 2D objects

In 2D work the 2D objects are generated in a **partial drawing**, and in 3D work in a **workplane**.
Whenever you want to use a group structure for these objects, you can create one subsidiary to the partial drawing, i.e. below the current workplane.

If 2D objects that are needed in other drawings as well are saved in a **symbol**, the objects will be compressed into a group and saved in this symbol. If this symbol is disassembled after being pasted in a drawing, the group in it will become "visible" again and embedded in the existing group structure.

Groups for 3D objects

3D objects are managed using the ***Model Explorer*** under the ***3D Solid*** structure.
If you also wish to set u 3D objects in a group structure, this can be defined under this structure node.

Generate group

2D objects

In the case of 2D objects the basis for a group structure is the active partial drawing in the current drawing. In the 3D window it is the current workplane.

When setting up a new group, proceed as follows:

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.
2. If necessary, open the structure branch using the ***Context Menu, Open Branch*** or ***Open All*** or using the ***F11*** key and tag the

workplane, the partial drawing or the group under which the new group is to be generated.

3. Define the new group using the key combination ***Ctrl+N*** or using the context menu ***New, Group***.
   The generated group is generated in the structure level under the active group, partial drawing or workplane and firstly is allocated the name ***New Group***.

4. If the ***Determine new group as current group*** setting is active, the new group automatically becomes the active group and can be directly renamed. Quit the procedure with the Enter key.
   If the command is deactivated the previous group, partial drawing or workplane remains active. In this case, you can rename the new group(s) as described in the following.

5. Tag the group you wish to rename. Press the ***F2*** key (or context menu ***Rename***) and enter the required name. Quit the entry with the Enter key.

3D objects

In the case of 3D objects the basis for a group structure is the structure node ***3D Solid***. Beneath it you can create groups in the same way as described in the previous section.

Please note:

Using the menu command ***Process***, ***Objects in group*** objects can be summarized in the drawing or in the 3D window without defining groups in the ***Model Explorer*** first.

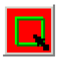
Objects in group

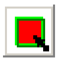
Objects can be summarized in a group with a mouse click using the menu command ***Process, Objects in group.*** Action / Selection or Selection / Action can be used for this. The command is also in the context menu ***Process***, which can be accessed via the right mouse button if no command is active. The elements which are selected already or which are selected then are summarized in a group with the name ***New Group*** within the active partial drawing / workplane and group.
Elements being in other groups or partial drawings are moved into this new group.

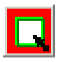
Action - Selection

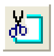
1. If the 2D window is active use the command ***Active sketchboard*** to select the partial drawing(s) containing the elements to be summarized in a group.

2. Choosing elements by selection window activate the correct selection mode:

 **Selection mode: All**
All 2D line objects touched or inside a selection by window are completely selected.

 **Selection mode: Inside**
Only those 2D line objects are selected which are completely inside the selection window.

 **Selection mode: Outside**
Only those 2D line objects are selected which are completely outside of the selection window.
Important:
Using this definition all the elements outside of the window are selected and summarized in a group, even those elements which maybe are outside of the visible area of your model.

 **Selection mode: Clip**

2D line objects are clipped at the frame of the selection window. The Clip frame is shown in a green rectangle.

3. Start the command ***Objects in Group*** using the following possibilities:

- the menu command Process, Objects in Group
- the context menu Process, Objects in Group
- the keys ***Alt b p***

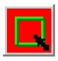
4. Choose the objects to be summarized in a group. Use the possibilities to select by window or using the ***CTRL-key***.
After the selection of the last element the new group will be generated inside the actual partial drawing / workplane or the active group with the name ***New Group***.

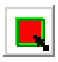
5. The command stays active.
Repeat the last step to summarize more objects in a group or stop the command using the ESC key or the right mouse button.

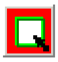
Selection - Action

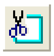
1. If the 2D window is active use the command ***Active sketchboard*** to select the partial drawing(s) containing the elements to be summarized in a group.

2. Choosing elements by selection window activate the correct selection mode:

 **Selection mode: All**
All 2D line objects touched or inside a selection by window are completely selected.

 **Selection mode: Inside**
Only those 2D line objects are selected which are completely inside the selection window.

 **Selection mode: Outside**
Only those 2D line objects are selected which are completely outside of the selection window.
Important:
Using this definition all the elements outside of the window are selected and summarized in a group, even those elements which maybe are outside of the visible area of your model.

 **Selection mode: Clip**

2D line objects are clipped at the frame of the selection window and summarized in a group. The Clip frame is shown in a green rectangle.

3. Start the command to select objects, e.g.:

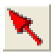
-  Reselect Object
-   Select, Select Object

4. Identify the objects to be summarized in a group

5. Start the command ***Objects in Group*** using the following possibilities:

- the menu command Process, Objects in Group
- the context menu Process, Objects in Group
- the keys ***Alt b p***

The selected elements will be summarized in a new group. This new group is generated inside the actual partial drawing / workplane or the actual group with the name ***New Group***.

6.  Deselect the objects using the command ***Reselect Objects***.

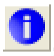
Please note:

When 2D and 3D elements both are selected in a 3D window the elements are summarized in the respective 2D and 3D groups.

The newly defined groups can be renamed, moved, disassembled or deleted in the ***Model Explorer***.

,  In addition, the contents of the groups can be shown using the command ***Scan-mode on***.

Using the context menu ***Visibility / Pickability*** in the ***Model Explorer*** the contents of groups can be dectivated and / or hidden.

,  If empty groups are left due to restructuring they can be deleted using the command ***Information, Check / Clean Model***.

Move into Group

Using the menu command ***Process***, ***Move into Group***, you can move objects including their group structure within the structure tree (model explorer) to a new or previously selected target group. You select the objects in a 2D drawing or the 3D window.

If a group contains more than one object, the selection of a single object is sufficient to move the entire group including the subgroups in the structure.
When several objects are selected from different groups, all superordinate groups are moved to the common node point if no further group is connected to this node. Otherwise, the node is copied. (Example).

Action - Selection

1. If the 2D drawing window is active, you can use the ***Select on ... sketchboard*** function at the top of the window to determine which partial drawings are to be taken into account when selecting the objects.
2. Activate a group to which the selected objects and groups should be moved.
   A new group is created, if no group is activated.
3. Start the menu command ***Process***, ***Move into Group***.
4. Identify the objects in the graphic that are to be moved, including the associated groups, within the model explorer's structure by

dragging a window or by collection by holding down the ***Ctrl*** key. The selection of the last object must be carried out without pressing the ***Ctrl*** key.

Selection - Action

1. If the 2D drawing window is active, you can use the ***Select on ... sketchboard*** function at the top of the window to determine which partial drawings are to be taken into account when selecting the objects..

2. Activate a group to which the selected objects and groups should be moved.
A new group is created, if no group is activated.

3. Start a command to select objects; for example one of the following commands:

- ***Reselect Object***
- , ***Select, Invert Selection***

4. Identify the objects that are to be moved along with the associated groups in the structural level.

5. Start the menu command ***Process, Move into Group***.

6. Deselect the objects using the command ***Reselect Objects***.

Please note :

If you have selected 2D and 3D objects in the 3D area, the selection list is split up and subdivided into 2D or 3D groups.

, ***Scan Groups On / Off***
Using the buttons shown, you can browse through the group structure in the ***Model Explorer*** to display the contents of the respective groups.

Use the command ***Objects in Group*** if the selected objects are to be grouped in a new group; Even though they may be removed from another group.

Lowlight non active Groups

2D and 3D objects can be structured hierarchically, for example to hide and show them or to select them in one step. Newly created objects are stored in the current group.

To be sure to have the desired group active when creating new objects, all the objects in non active groups can be displayed in another colour in the 2D window or can be displayed transparent in 3D space.

The ***Lowlight colour*** for 2D objects can be defined using the menu command ***Settings, Options*** in the tab ***General Settings***. The ***Transparency for Lowlight*** of Solids can be defined using the menu command ***Settings, 3D Display*** in the tab ***Solid Display***. If the edge display is activated for solids, the edges are also displayed in the lowlight colour for 2D objects.

To highlight the geometry in active groups, open the ***Model Explorer*** using the menu command ***View***. In the Model Explorer the following buttons can be found:

If the button is displayed as shown, the command is ***deactivated***. All objects in active and non active groups are displayed according to the standard settings. When the button is selected, the button display is changed as described below:

If the button is displayed as shown, the command is ***activated***. In this case only the objects in the active group are displayed according to the standard settings. All other objects are lowlighted. When changing the active group, e.g. in the Model Explorer, the display of the objects in active and non active groups will change accordingly.

Determine group as current group

Before a group can be used for the embedding of 2D or 3D objects, it must firstly be selected as the current group.

2D objects

You can use one of the methods described below to select the current group for 2D objects:

Using the *Model Explorer (1)*

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model*** tab.
2. Activate the option ***Activate selected components***.
3. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
4. By identifying a group, this will become the current group.

Using the *Model Explorer (2)*

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model*** tab..
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. Tag the required group in the structure.
4. On the tagged group, press the ***F12*** key or select the ***Active Group*** option from the context menu.

Using group selection

1. By clicking on the field in the 3D window, open the ***group selection*** dialog box.
2. In the dialog box which appears for group selection, click on the required group name and close the dialog box with *Close*.

Using object identification

1. ***Activate Group***. Click on this icon next to the 2D group selection in the main window.
2. Identify a 2D object.
   The group in which this object is will be the current 2D group.

<u>3D objects</u>

You can use one of the methods described below to select the current group for 3D objects:

Using the Model Explorer *(1)*

1. Using the ***View*** menu, open the ***Model Explorer***.
2. Activate the option ***Activate selected components***.
3. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
4. By identifying a group, this will become the current group.

Using the Model Explorer *(2)*

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. Tag the required group in the structure.
4. On the tagged group, press the ***F12*** key or select the ***Active Group*** option from the context menu.

Using group selection

1. By clicking on the field in the 3D window, open the ***group selection*** dialog box.
2. In the dialog box which appears for group selection, click on the required group name and close the dialog box with *Close*.

Using object identification

1. ***Activate Group***. Click on this icon next to the 3D group selection in the 3D window.
2. Identify a 3D object.
   The group in which this object is will be the current group.

Scan groups

You can display objects only in a group and its subgroups so that you can scan the groups by using the following procedure:

1. Using the ***View*** menu, open the ***Model Explorer***.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. Activate the option ***Scan Groups on.***
4. By clicking in the structure tree, select the group whose objects are to be displayed. Scan further through the structure with the up and down arrow keys on the keyboard. Right and left opens or closes a node point in the structure tree.

Define visible/hidden group

In order to make a group invisible, proceed as follows:

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. By clicking or using the cursor keys, select the group that is to be defined as hidden.
4. Confirm the hotkey ***O*** or select the ***Visibility/Pickability, Visible/Invisible*** command from the context menu.
   Hidden groups are automatically inactive and are identified by the icon.

Please note:

The visible yes/no attribute is inherited by all subgroups and objects.

In order to make the group and its subgroups visible again, repeat the procedure.

Hidden groups are automatically inactive. If you activate the group (Ctrl+P), the group is visible again.
However, if the group is switched to visible (Ctrl+O), it remains inactive.

Define active/inactive group

In order to define a group as inactive, proceed as follows:

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. By clicking or using the cursor keys, select the group that is to be defined as inactive.
4. Confirm the key combination ***P*** or select the ***Visibility/Pickability, Active/Inactive*** command from the context menu.
   Objects in groups that you have deactivated using this command, remain visible, but can however no longer be picked or processed. Inactive groups are given the  symbol.

Please note:

For inactive groups, their subgroups and objects contained within them are excluded fro every type of selection or identification with the cursor.

In order to make the group and its subgroups active again, repeat the procedure.

Rename group

In order to rename a group, use one of the following options:

1. Using the ***View*** menu, open the ***Model Explorer***.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. By clicking or using the cursor keys, select the group that is to be renamed.

4. Confirm the ***F2*** key or select the ***Rename*** option from the context menu.
5. Enter a new name. Quit the entry with the Enter key.

Move group

If a group and all the subgroups and objects (group branch) are to be moved in another group, proceed as follows:

Drag&Drop

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. By clicking or using the cursor keys, select the group that is to be moved excluding its subgroups.
4. Keep the mouse button pressed and drag the group to the respective folder.
5. Then release the mouse button.
   The group in the hierarchy will be set up one level under the selected folder.

Cut&Paste

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. By clicking or using the cursor keys, select the group that is to be moved with its subgroups.
4. Confirm the key combination ***Ctrl+X*** or select the ***Cut*** option from the context menu.
5. Tag the folder under which you wish to insert the group(structure).
6. Confirm the key combination ***Ctrl+V*** or select the ***Paste*** option from the context menu.

Please note:

If you wish to move or copy the entire group structure of a partial drawing and the 2D objects contained within this into another partial drawing, keep the mouse button pressed to move the topmost group of the structure within the ***Model Explorer*** into the target partial drawing (Drag&Drop). If you wish to take over a copy into the other partial drawing, keep the ***Ctrl key*** pressed. The objects are inserted in the new partial drawing according to their absolute coordinates.

Disassemble group

If a group is no longer required but the subgroups and 2D objects are to be retained, disassemble the group. In this way, the subgroups and objects move one hierarchical level up in the group tree.

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. By clicking or using the cursor keys, select the group that is to be disassembled.
4. Confirm the key combination ***Ctrl+Del.*** or select the ***Disassemble*** option from the context menu***.***

Erase group

This command can be used to erase a group branch, i.e. the current group with all subgroups and objects contained in it.

Using the Model Explorer, group branches can be erased, i.e. the current groups with all the subordinate groups and the objects within them.

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. By clicking or using the cursor keys, select the group that is to be deleted.
4. Press the ***Del.*** key or select the ***Erase*** option from the context menu***.***
5. Confirm the erase procedure with ***Yes.***

<u>Alternative Method to delete Groups in the Graphic.</u>

 After clicking the displayed icon the command ***Erase Group*** is started.

This command can only be executed in a user defined icon bar or in user defined menus.

Start the command by clicking the displayed icon and identify an object of the desired group.

Please note:

If you want to erase all the empty groups in the model, that is all the groups not containing subgroups or objects, use the command ***Information, Check and Clean Model***.

Move objects into another group

In order to move objects after they have been generated into other groups (within the same partial drawing, workplane or 3D model), proceed as follows:

For 2D objects in the drawing

1. Select the 2D objects that are to be moved into another group.
2. Select the menu command ***Process, Insert Selection In***.
3. Using group selection, determine the group in which the selected objects are to be taken over.

For 2D and 3D objects using the Model Explorer

1. Select the 2D or 3D objects in the drawing or 3D model that are to be moved into another group.
2. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.
3. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
4. By clicking, select the group, the partial drawing, the workplane or the 3D model in which the objects are to be moved.
5. Confirm the key combination ***Shift+Alt+S*** or select the ***Select / Selected Objects*** option from the context menu.

Please note:

Each object can only belong to one group. Therefore, objects that are taken over in a group that previously belonged to another group, must be removed from this other group.

Objects can also be moved between groups within the ***Model Explorer*** using ***Drag&Drop*** or ***Cut&Paste*** (within the same partial drawing, workplane or 3D model).

Objects can also be copied between the groups. If you keep the ***Ctrl key*** pressed during the move procedure, the selected objects are copied into the target group. The position is taken over starting from the absolute coordinate system.

Select objects of a group

If all objects in a group and the subordinate groups are to be taken over in the selection list, proceed as follows:

Using the toolbar

1. In the toolbar, select the command button ***Reselect Object*** .
2. Keep the ***Shift key*** pressed whilst you identify a 2D or 3D object.

   In the ***Standard selection*** mode all objects in the group are selected. To select all objects in the group in the ***Group selection*** mode choose ***Elements in group*** in the context menu.

Using the cursor

1. Select the menu command ***Process, Select Object***.
2. Keep the ***Shift key*** pressed whilst you identify a 2D or 3D object.

   In the ***Standard selection*** mode all objects in the group are selected. To select all objects in the group in the ***Group selection*** mode choose ***Elements in group*** in the context menu.

Using the Model Explorer

1. Using the ***View*** menu, open the ***Model Explorer***.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. Keep the ***Shift key*** pressed whilst selecting the required group in the structure.

   or

   In the structure, identify the required group and confirm the key combination ***Shift+S*** or, from the context menu, select the option ***Select, Select All Geometric Objects***.

Please note:

When you are using this procedure, all that will be selected are the objects in the group. The structural element ‘group’ itself will not be selected.
If you do want to select the group, you must use the menu command ***Select Group***.

If this is required, use the ***Select / Select Object (Alt+S)*** menu command from the context menu.

Deselect objects of a group

If all objects in a group ad the subordinate groups are to be removed from the selection list, proceed as follows:

Using the toolbar.

1. From the toolbar, select the ***Reselect object***  command button that has become red after selection.

All objects contained in the selection list are then selected.

Using the cursor

1. Select the menu command ***Process, Deselect All***.

Using the Model Explorer

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.

2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***

3. In the structure, identify the required group and confirm the key combination ***Shift+D*** or, from the context menu, select the option ***Select, Deselect All Geometric Objects***.

Select objects of the active group (excluding the subgroups)

If you only wish to select the objects of a group without the subordinate subgroups, the following options are available to you:

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.

2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***

3. By clicking, select the group, partial drawing, the workplane or the 3D model whose objects are to be selected.

4. Confirm the key combination ***Ctrl+S*** or select the ***Select / Select Geometric Objects*** option from the context menu.

Deselect objects of the active group (excluding the subgroups)

If you wish to remove objects of a group from he selection list without the subordinate subgroups, the following options are available to you:

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. By clicking, select the group, partial drawing, the workplane or the 3D model whose objects are to be deselected.
4. Confirm the key combination ***Ctrl+D*** or select the ***Select / Deselect Geometric Objects*** option from the context menu.

Select a group

If, whilst selecting with the cursor, you press the Shift key, in the ***standard selection*** or in the ***group selection*** (to be defined in the main toolbar) and using the option ***single objects in:..*** all objects in a group and all subgroups are selected.
This is also the case when using the context menu commands ***Select, Select All Geometric Objects*** that are offered in the ***Model Explorer***. For this, only the objects are selected, not however the structure element group.
This can be compared to tagging all files in a folder, whereby the folder itself is not tagged.

For the **Copy** and **Erase** commands you can also select the group itself and all the objects contained within it. In the Group selection mode not only the objects in the group are deleted but also the group. When copying not the objects are copied into the current group but the complete group is copied into a new group.

With the cursor

1.  Select the menu command ***Process, Select Objects,*** or use the command ***Reselect*** in the 2D or 3D toolbar.
2. Keep the ***Alt key*** pressed whilst identifying a 2D or 3D object. If the ***Group selection*** mode is active, structured groups can be selected from a context menu. For that keep the ***Shift key*** pressed.

   If the ***Group Selection*** is active in the main icon bar, by pressing the ***Shift*** key a popup appears to select the appripriate group. In this case keep the ***Shift*** key pressed whilst identifying an object. Move over the popup with the cursor and the related groups will be highlighted in the graphic. The different colours also indicates whether you are selecting a group or the objects in a group.

or:

1.   By clicking on the displayed icons in the symbol toolbar, select the ***Select, Select Group*** command.
2. Identify a 2D or 3D object of the group to be selected.
   If the ***Group selection*** mode is active select the desired group from the context menu.

Using the Model Explorer

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. Keep the ***Alt-key*** pressed and click on the group with the left mouse button. The group including sub groups and all included objects are selected.

   or

   tag the group to be selected and follow the further instructions.
4. Confirm the key combination ***Alt+S*** or select the ***Select / Select Object*** option from the context menu.

After you have selected a group, the objects in it will be displayed in the colour specified in ***Settings, Options*** as the ***Selection colour for groups***.

Please note:

Selected groups and the objects contained within them can then be processed using the commands ***Transform, Move (with copy)***, ***Rotate (with copy)***, ***Mirror (with copy)***, ***Scale (with copy)*** and ***Delete Object/Selection***.

Deselect a group

If you have selected a group, you can use one of following methods to remove it from the selection.

Using the toolbar.

1. From the toolbar, select the ***Reselect object***  command button that has become red after selection

   All objects contained in the selection list are then deselected.

With the cursor

1. ,  Select the menu command ***Process, Deselect Objects***.

   Keep the ***Alt key*** pressed and identify an object.

or:

1.   Using the displayed icons in the toolbar, select the command ***Select, Deselect Group***.
2. Identify an object in the group that is to be deselected.

Using the Model Explorer

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. By clicking, choose the group, the subgroups contained in it (and all objects) that is to be deselected.
4. Confirm the key combination ***Alt+D*** or select the ***Select / Deselect Object*** option from the context menu.

Select entire partial drawing

If you wish to move or copy the entire partial drawing i.e. including group structure and all objects contained within it, into another partial drawing, you can use Drag&Drop within the ***Model Explorer*** to move or copy or firstly take it over into the selection list. For this use the commands in the context menu ***Cut, Copy*** and ***Paste***.
In addition the structure of a partial drawing can be included into the selection list as follows:

1. Using the ***View*** menu, open the ***Model Explorer*** and activate the ***Model Explorer*** tab.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. By clicking, choose the partial drawing (2D drawing) or the workplane (3D model), the subgroups within them (and all objects) to be selected.

4. Confirm the key combination ***Ctrl+Alt+S*** or select the ***Select / Select All Contents***.
5. Finally, use the toolbars ***Transform / Move (if necessary with copy)*** in order to move the objects including the entire structure to another partial drawing / workplane.

4.16 Use symbols

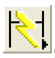
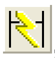
Symbols can be used to include identical tools, etc. which occur frequently into drawings. Once these symbols have been generated, they can then be inserted in the drawing as often as required.

Symbols can be used to save 2D objects and load them later as often as necessary in different positions and sizes in the active partial drawing of your current drawing.

Symbols are saved in the library files with the extension SYL, in which specific directory structures can be defined.

The selection for inserting symbols into a drawing can be made using the symbol name or by using graphic symbol selection has been set up with the illustrations for the symbols.

Symbols inserted in a drawing are managed as objects. The objects within the symbols retain their own layer or object specific display attributes. If needed also the object related properties can be used for the display properties of a symbol.
It is possible to snap points on the objects within symbols, limited to ***Endpoint***, ***Centre Point, Midpoint***, ***Quadrant Point***, ***Intersection 2 Lines, Text Reference*** and ***Snap Point***.
The Auto snap also snaps the following points on 2D line objects inside Symbols, without disassembling them before: ***End point***, ***Mid Point***, ***Centre Point***, ***Quadrant Point***.

, ,  Using the dimension commands ***Multi Dimension orthogonal*** and ***Multi Dimension Aligned*** line objects in Symbols can be identified, e.g. to dimension the length of a line, the distance between parallel lines, the angle between non parallel lines or the diameter of arcs and circles.
Dimensions created to lines in symbols in the same partial drawing, or to points of this elements are **object related**. They behave associative when changing the position or size of their related objects and they are deleted when the related symbol will be disassembled, updated, exchanged by another symbol or erased.

Symbols contained in a drawing are saved in a model, i.e. they can be used even without the symbol library.

If a symbol is disassembled, the objects within in it can be accessed again freely.
The group and layer structure within a symbol is integrated in the current structure of the model if the symbol is disassembled.

In a model, existing symbols can be archived at any time in a symbol library, i.e. transferred from the model into the library.

If a model contains symbols which have been modified and been saved under the same name in a symbol library, the original symbols are not automatically replaced by the modified symbols. In order to carry this out, use the opportunity to update the symbols in the model.

Apply Display Settings from Symbols

Also saved in a symbol, in addition to the objects saved (lines, circles, text, etc.), are the layer and display settings of these objects.

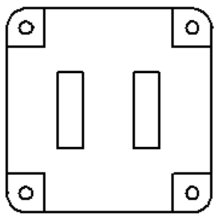
If you add the symbol to a drawing / to a model, the objects are then displayed in the symbol with their original layer and display settings.

The objects in the symbol may also take over the properties of the layer upon which the symbol was inserted, or the display settings which you have explicitly assigned to the symbol (layer-related or object-related display).

The settings in the menu ***Settings, Symbol …*** were expanded to the extent that you may now take over individual display settings from the symbol. These are: the settings for the object colour, the line type, the line width, the drawing order, visible and / or active.

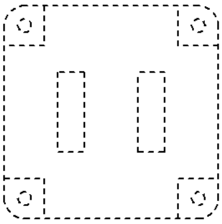
In addition to this, you may also retrospectively change the display settings of a symbol via the menu ***Process, Object Display*** as explained in the following example:

Example:
You want to change the line types of all objects in a symbol without having to disassemble the symbol. Proceed as follows:



Initial situation

1. In the menu bar, activate the command ***Process, Object Display***.
2. Identify the symbol.
3. Change the line type to dashed ------- . in the ***Object Display*** dialog.
4. Activate the option ***line type*** in the lower section of the dialog.
5. Close the dialog by pressing the ***OK*** button and end the command with the right mouse button.



Result

Generate symbol library

The requirement for saving symbols are symbol libraries. These are files with the extension SYL, in which individual symbols can be managed within its own directory structure.

When generating a new symbol library, proceed as follows:

1. Select the menu command ***Define, Symbols***.

2. In the dialog box, select the menu command ***Library, Generate***.

3. If the symbol library is to be saved in the current folder for symbol directories, enter the name of the new symbol library in the ***current library*** text box. The extension SYL is predetermined. It is added automatically.

If the library is to be saved in a different folder, before entering the file name, enter the required drive and folder.

Please note:

After generating a symbol library, this is added to the library list. At the same time, it becomes the current library.

Library files are always searched for in the preset ***symbol library folder*** for all commands which concern the library file itself or the symbols.

This folder can be determined using the menu command ***Settings, Folder***.

In the Symbol manager the drop down list for the ***current library*** is also an input field. When entering a library name and confirm with

Enter this library is loaded if it is in the current library directory. Otherwise a new library is created.

Load symbol library

When saving or loading symbols, one of many symbol libraries can be used.

These must previously be loaded in the working memory.

In order to ass an existing symbol library, proceed as follows:

1. Select the menu command ***Define, Symbols***.
2. Within the dialog box, select the menu command ***Library, Add***.
3. If the library file being searched for exists in the currently set folder for symbols, enter the name in the ***current library*** text box or select from the list displayed.

   If the file is in another folder, before entering the file name, determine the drive and folder in which the file can be found.

   After you have loaded a symbol library, it is added to the library list. At the same time it becomes the current library.

Please note:

You can also load a symbol library by using Drag & Drop.

In the Symbol manager the drop down list for the ***current library*** is also an input field. When entering a library name and confirm with ***Enter*** this library is loaded if it is in the current library directory. Otherwise a new library is created.

The list of symbol libraries that are loaded will be saved both in the templates and in the models.

Determine current symbol library

When saving symbols as well as by loading symbols, the current symbol library is used as the target and source library.

The current symbol library can be determined as follows:

1. Select the menu command ***Define, Symbols***.
2. Select the required symbol library in the ***current library*** list box.

   If the library list does not contain any entries, within the dialog box select either the menu command ***Library, Generate*** or ***Library, Add*** or enter a library name and confirm with ***Enter.*** This library is loaded if it is in the current library directory. Otherwise a new library is created.

Reorganise symbol library

When symbols or directories are erased from a symbol library, a tag will indicate that they have been erased, but they will not be actually deleted completely. To remove them from the library and to reduce the size of the file, you must reorganise the symbol library:

1. Select the menu command ***Define, Symbols***.

2. If no library is to be loaded, use the dialog box menu command ***Library, Add*** to select the library which is to be reorganised.

Otherwise select the library in the ***current library*** list.

3. Within the dialog box, start the menu command ***Library, Reorganise***.

Please note:

When this command is executed, the reorganised symbol library will be written on to the HD. For this reason you cannot execute the command unless you have write access to the drive.

Repair symbol library

If a symbol library is damaged or contains damaged symbols, you must repair the library.

1. Select the menu command ***Define, Symbols***.

2. Start the menu ***Library, Repair*** inside the dialog box and select the library file which is to be repaired.

After you have activated this command, an automatic backup copy of the symbol library will be made using BAK as the extension. Afterwards the library file and all the symbols it contains will be checked and, if possible, repaired.

Please note:

When this command is executed, the backup file and the repaired symbol library will be written on to the HD. For this reason you cannot execute the command unless you have write access to the drive.

Close symbol library

Each loaded symbol library requires memory. Therefore, close any symbol libraries which are not (permanently) required:

1. Select the menu command ***Define, Symbols***.
2. Select the symbol library which is to be removed from the ***current library*** list.
3. Within the dialog box start the menu command ***Library, Remove***.

Create library directory

Symbols could simply be saved under their name in a symbol library. In this case, all symbols would be saved in the main library directory.

With a large number of symbols such a procedure would make it difficult to find specific individual symbols.
By setting up a directory structure, symbols can be saved in symbol library separately according to special faces or order criteria.

Set up a library directory as follows:

1. Select the menu command ***Define, Symbols***.
2. Choose a library from the list in ***current library***.
   If no library is loaded, load one using the dialog box menu command ***Library, Generate*** or ***Add***, in which the directory is to be set up.

   Otherwise select from the ***current library*** list or enter a library name and confirm with ***Enter.*** This library is loaded if it is in the current library directory. Otherwise a new library is created.
3. By double clicking on the icon in the block under ***directory structure***, open the directory tree and subdirectories if any exist.
4. Select the directory under which a new subdirectory is to be set up by clicking on the respective one.
5. Within the dialog box, start the menu command ***Library, Create Directory***.
6. In the *new* directory dialog box, enter the name for the directory and confirm with Enter or ***OK***.

   Whole sequences of directories can be entered. The slash "/" is used as the separator for this.

Example:

The selected directory is e.g. the main directory, identified by /. By entering

`Screws/DIN933`

the directory /Screws/DIN933 is set up.

Please note:

After you have created this directory, you must generate at least one symbol in it: the reason is that empty directories will be deleted automatically as soon as you close the dialog box.

Rename library directory

An **empty** library directory can be given a new name as follows:

1. Select the menu command ***Define, Symbols***.
2. If no library is loaded use the dialog box menu command ***Library, Add*** to load the library which is to be renamed.

   Otherwise select from the ***current library*** list.
3. By double clicking on the icon in the block under ***directory structure***, open the directory tree and subdirectories if any exist.
4. Select the directory to be renamed by clicking on the respective one.
5. Within the dialog box, start the menu command ***Library, Rename Directory***.
6. Enter the new name for the directory in the ***to*** annotate box. Confirm with Enter or ***OK***.

Remove library directory

Library directory can only be erased if they are empty, therefore once they do not contain symbols or subdirectories.

In order to remove an **empty** library directory, proceed as follows:

1. Select the menu command ***Define, Symbols***.
2. If no library is loaded use the dialog box menu command ***Library, Add*** to load the library.

   Otherwise select from the ***current library*** list.
3. By double clicking on the icon in the box under ***directory structure***, open the directory tree and subdirectories if any exist.
4. Select the directory to be removed by clicking on the respective one.

5. Within the dialog box, start the menu command ***Library, Delete Directory***.
6. Confirm with Yes if the selected directory is to be removed.

Save symbol

The 2D objects in the current partial drawing can be saved in a symbol.

You can use one of the following methods to save a symbol.

By selecting the objects that are to be saved

The 2D objects that are to be saved in a symbol can be chosen by using the standard method for selecting.
They are inserted in the selection list before the command is executed or whilst the command is being executed inserted in the action list.

Please note:

If you want to save the group structure as defined so far and any attributes specified for the group as part of the symbol, you must use the method described below. This is the method to be used for **standard sheet symbols** and **2D standard parts**.

By selecting the group that is to be saved

You must select the 2D objects that are to be saved by selecting the group: this is the only method. The group that is selected and the objects and subgroups it contains will be saved in the symbol.

You must use this selection method, if the symbol is for saving not only the 2D objects but also attributes that have been allocated to the highest group within the group structure. This applies, for example, to standard sheet symbols.

<u>**By selecting the objects that are to be saved**</u>

1. Transfer the 2D objects that you want to save to the selection list. The only objects to be saved will be those in the current partial drawing.
   If you do not define a selection list, you can move the objects into the action list later.
2. Select the menu command ***Define, Symbols.***
   ,  Alternatively, you can click the two icons shown here to load the command ***Process 2D, Generate Symbol*** before continuing with Step 4.
3. Start the command ***Symbol, Generate*** in the dialog box.

4. In the line ***in library*** you will see the name of the most current symbol library now suggested as the target library.
   If you want to use a different library, select one from the list or load another by clicking .
   In additon the drop down list for the ***current library*** is also an input field. When entering a library name and confirm with ***Enter*** this library is loaded if it is in the current library directory. Otherwise a new library is created.
5. By double clicking the icon in the box alongside ***in directory*** open the directory tree and, if required, any subdirectories. Select the directory to be used to save the symbol by clicking it.
6. Type the name of the symbol in the ***symbol name*** box.
7. If you want to define two ***additional reference points*** for the symbol in addition to the insertion point, activate this setting. With symbols that are inserted you can use these points as Geo points, i.e. the symbols can be processed with the command ***Transform, Show Geo Points***. If you define the symbol as a standard sheet, the sheet format is defined using the additional reference point.
8. Select a creation mode:

   Save
   A copy of the 2D elements are stored in the current library as a symbol.

   Save and erase
   The geometry is erased from the drawing and stored as symbol in the current library.

   Save and replace
   The 2D elements are stored in the current library as a symbol. At the same time the 2D elements in the drawing are exchanged by the symbol.
9. Confirm the settings by clicking ***OK***.
10. If the selection list is empty, transfer the objects that are to be saved into the action list.
11. Define the first reference point, i.e. the point with which the symbol is to be inserted in drawings later.
12. Whatever the setting made in Step 6, specify two further reference points, if you are going to need them.

Afterwards the symbol will be saved in the symbol library. The symbol contains a **group** with the same name. The group contains all the 2D objects that have been saved.

<u>**By selecting the group that is to be saved**</u>

1. Select the menu command ***Define, Symbols***.

2. Start the menu command ***Symbol, Generate*** in the dialog box.
,  Alternatively, you can click the icons shown here to start the command ***Process 2D, Generate Symbol*** and then continue from Step 4.

3. Start the command ***Symbol, Generate*** in the dialog box. In the line ***in library*** you will see the name of the most recently selected symbol library now suggested as the target library.
If you want to use a different library, select one from the list or load another by clicking .

4. By double clicking the  icon in the box alongside ***in directory*** open the directory tree and, if required, any subdirectories.
Select the directory to be used to save the symbol by clicking it.
In additon the drop down list for the ***current library*** is also an input field. When entering a library name and confirm with ***Enter*** this library is loaded if it is in the current library directory. Otherwise a new library is created.

5. Type the name of the symbol in the ***symbol name*** box.

6. If you want to define two ***additional reference points*** for the symbol in addition to the insertion point, activate this setting.
With symbols that are inserted you can use these points as Geo points, i.e. the symbols can be processed with the command ***Transform, Show Geo Points***. If you define the symbol as a standard sheet, the sheet format is defined using the additional reference point.

7. Activate the setting ***save defined group as symbol***.

8. Select a creation mode:

Save
A copy of the 2D elements are stored in the current library as a symbol.

Save and erase
The geometry is erased from the drawing and stored as symbol in the current library.

Save and replace

The 2D elements are stored in the current library as a symbol. At the same time the 2D elements in the drawing are exchanged by the symbol.

9. Confirm the settings by clicking ***OK***.
10. The dialog box for selecting groups will appear.
    Tag the group that is to be saved a s a symbol and the confirm this step by clicking ***OK***.
11. Define the first insertion point, i.e. the point that is to be used later to insert the symbol in drawings.
    In a standard sheet symbol it is usually the Origin 2D (X=0; Y=0)
12. Whatever the setting made in Step 7, specify two further reference points, if you are going to need them.
    In a standard sheet symbol the first reference point is right below (e.g. A3: X=420; Y=0), the second reference point is left upper (e.g. A3: X=0; Y=297). Via this diagonal points the page format is stored with a standard sheet symbol.

Afterwards the symbol will be saved in the symbol library. The symbol contains a **group** with the same name. The group contains all the 2D objects that have been saved.

Please note: :

When entering the symbol name, upper case, lower case and digits are permitted. Spaces are automatically replaced by underlines. **But if** you are exporting data into the DXF or DWG format, the symbol name must only contain the characters A-Z, 0-9, -, _.

Groups and objects as well as layers, are saved in a symbol with which the objects are linked, as well as the information as to whether the display attributes are to be object or layer specific.

If a symbol is scaled whilst being positioned, the geometry of all the objects it contains will be enlarged or reduced for the display. This applies in particular to standard parts and to dimensions; for this reason these will not have their correct measurements/dimension numbers.
For this reason 2D standard parts should never be saved in symbols that are going to be scaled when positioned.
Dimensions in scaled symbols will, on the other hand, be corrected automatically, if the symbols are disassembled.

Archive symbol in library

Assume that symbols are to be used which are contained in model files, which are not or no longer in their symbol library.

In this case, it is possible to accept the symbols contained in the model into a symbol library.

1. In the 2D window, activate the partial drawing which contains the symbols to be archived.
2. Select the menu command ***Define, Symbols***.
3. Choose a library from the list in ***current library***.
If no library is loaded use the dialog box menu command ***Library, Generate*** or ***Add*** to load the library which is to be archived.
Otherwise select from the ***current library*** list or enter a library name and confirm with ***Enter.*** This library is loaded if it is in the current library directory. Otherwise a new library is created.
4. Within the dialog box, start the menu command ***Symbol, Archive***.
5. Identify an individual symbol or accept a number of symbols in the action list.
6. In the dialog box which then appears, the names of the source library, directories and symbols are displayed.

   Modify the entries for ***target directory*** and ***symbol name***, if another structure is to be used and another symbol name given. Folder names, and directories which are on different hierarchical levels should be separated by a slash (/).
7. Use one of the following options:

   OK, if the symbol is to be archived with these entries.
   Cancel if the currently displayed symbol is not to be saved.
   Exit if this command is to be aborted.

If a number of symbols have been selected, repeat steps 5 and 6.
If a symbol is to be saved which exists with the identical directory and name entries, a security query is displayed.

In this case, use one of the following commands:

Yes, if the symbol in the library is to be overwritten.

No, if it is not to be overwritten. A name can then be entered

Cancel, if the symbol is not to be saved.

Please note:

If, **before** starting this menu command, a selection list has been defined, all the symbols within it can be archived one after the other.

Determine settings for inserting symbols

Independent of whether symbols have been inserted in the drawing using the menu command ***Define, Symbols*** (with named selection) or using the menu command ***Insert, Symbol*** (with graphic selection):

It is possible to position symbols according to the respective settings which have been determined using the menu command ***Settings, Symbol***.

insertion point
If this check box is active, the symbol is automatically set at the position defined using the preset ***scale factor*** and ***rotation angle***.
These values can be entered as numeric values or - after clicking on the command button  - determined using the cursor or the Snap Menu commands.

If the check box is not active, the symbol can be positioned dynamically using the cursor or the position can be defined by entering coordinates in the Status dialog box.

scale factor
If this check box is active, during insertion, the symbol is enlarged or reduced according to the entered factor to the base point.
If no fixed ***insertion point*** is entered, the scale factor can be modified in the Status dialog box whilst positioning.

rotation angle
If this check box is active, during insertion, the symbol is rotated according to the entered rotation angle about the base point.
If no fixed ***insertion point*** is entered, the rotation angle can be modified in the Status dialog box whilst positioning.

Disassemble when positioning
If this setting is active, when inserting a symbol, the symbol will be disassembled into its individual objects. Any symbols contained within this symbol (nested symbols) remain as symbols. The group structure that may exist in a symbol is allocated to the group in which the symbol is embedded.

Disassemble completely
After you have activated the ***disassemble when positioning*** button, you can further determine whether nested symbols are to

be disassembled into the lowest level. All symbols contained within the symbol are also disassembled with this.

Insert symbols with attribute values
This block can be used to determine, whether and which of the attribute values in a symbol are to be displayed for editing whilst inserting the symbol.

This is only valid for the attribute values which are allocated to the “main group“, i.e. the group which is selected when saving the symbol. Attributes which are allocated to individual objects or subordinate groups are not taken into account.

do not edit attribute values
The attribute values are not offered for editing after inserting the symbol.

only edit variable attribute values
After positioning the symbol, the attributes whose value begins with ? or an empty text, are offered for editing.

edit all attribute values
After inserting the symbol, all the attribute values allocated to the symbols in the “main group“ are offered for editing.

If the attribute values are not edited directly after inserting the symbol, this can be done at any time using the menu command ***Extras, Attribute, Process***.

The attribute values allocated to subordinate groups or individual objects within a symbol can only be edited, if the symbol is previously disassembled using the command Process 2D, Disassemble 2D Object or ***Process 2D, Disassemble Symbol***. Only then are e.g. numeric attribute values such as ***length*** and ***contents*** updated.

Apply Symbol Display Settings
Also saved in a symbol, in addition to the objects saved (lines, circles, text, etc.), are the layer and display settings of these objects.

If you add the symbol to a drawing / to a model, the objects are then displayed in the symbol with their original layer and display settings.

The objects in the symbol may also take over the properties of the layer upon which the symbol was inserted, or the display settings which you have explicitly assigned to the symbol (layer-related or object-related display).

The settings in the menu ***Settings, Symbol …*** were expanded to the extent that you may now take over individual display settings from the symbol. These are: the settings for the object colour, the line type, the line width, the drawing order, visible and / or active.

Select and insert symbol using name

One method of selecting a symbol from a symbol library and to put it in a drawing is to use the following method:

1. Select the menu command ***Define, Symbols***.

2. Choose a library from the list in ***current library***.
If no library is to be loaded, load the library using the dialog box menu command ***Library, Add*** from which the symbol is to be selected.

Otherwise select the library in the ***current library*** list or enter a library name and confirm with ***Enter.*** This library is loaded if it is in the current library directory. Otherwise a new library is created.

3. By double clicking on the icon in the block under ***directory structure***, open the directory tree and subdirectories if any exist.

4. Select the directory in which the symbol is to be saved by clicking on the respective one.

5. Tag the name of the symbol in the list of symbols.

6. In the dialog box, start the menu command ***Symbol, Insert*** or double click on the symbol name.

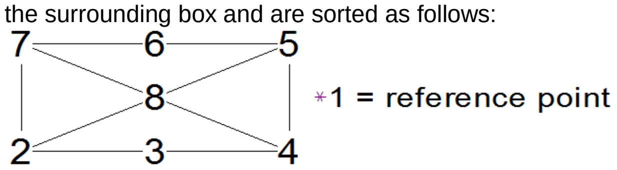
7. Insert the symbol according to the settings determined using the menu command ***Settings, Symbols***.
By typing the letter ***B***, or ***b*** you can switch to and from the box display and the complete display of the symbol's contents.

If necessary, use one of the following additional options before positioning the symbol in the target point:

Keyboard entry

X	The objects found on the cursor are mirrored on the X-axis.
Y	The objects found on the cursor are mirrored on the Y-axis.
Left arrow key ←	The objects found on the cursor are rotated in an anticlockwise direction by the angle defined in the Status dialog box.
Ctrl + Left arrow key ←	The objects found on the cursor are rotated in an anticlockwise direction by 90°
Right arrow key →	The objects found on the cursor are rotated in a clockwise direction by the angle defined in the Status dialog box.

Ctrl + Right arrow key →	The objects found on the cursor are rotated in a clockwise direction by 90°
Up arrow key ↑	The objects found on the cursor are scaled larger by the factor defined in the Status dialog box.
Down arrow key ↓	The objects found on the cursor are scaled smaller by the factor defined in the Status dialog box.
Ctrl+R	The objects found on the cursor are rotated dynamically about the cursor point.
Ctrl+S	The objects found on the cursor are scaled dynamically.
> **<**	switch between 7 additional insert points forward and backwards. The insert points will be calculated according to the surrounding box and are sorted as follows:



Please note:

When inserting a symbol, the display attributes of the objects within the symbol retain their validity, i.e. they are layer or object specific.

Object specific display attributes are kept.

The objects in the symbol may also take over the properties of the layer upon which the symbol was inserted, or the display settings which you have explicitly assigned to the symbol (layer-related or object-related display).

Otherwise the layers and objects are will be inserted in the following way:
If the symbol contains layers, which have the same name as those in the model, the layer specific display attributes are taken over from the model layer.

If the symbol contains layers which do not exist in the model, these are added.

Search for and insert symbol using name

With extensive symbol libraries, the search for individual symbols using their names, can often be linked with a lot of work.

If the name of the symbol is known, the search can be shortened in the following way:

Search using the Symbol manager

1. Select the menu command ***Define, Symbols***.
2. Select a libary from the ***current library*** list.
 If no library is loaded, use the dialog box menu command ***Library, Add*** to load the library in which the symbol should be searched for or enter a library name and confirm with ***Enter.*** This library is loaded if it is in the current library directory. Otherwise a new library is created.
3. Start the dialog box menu command ***Symbol, Find***.
4. In the ***search for*** text box, enter the name of the symbol which is being searched for.
 Please note that upper and lower case are differentiated between whilst searching. Use the (*) as the wildcard for the following characters.
5. In the ***beginning with*** text box enter the name of the directory in which the search is to start.
 If the search is to start in the main directory, a slash can simply be entered (/). If entering subdirectories, these should also be separated by a slash.
6. If the search for the symbol is not simply to take place in the start directory entered, but also in the respective subdirectories, activate the ***search subdirectory*** check box.
7. All the symbols found in the library with the entered name are displayed in a list including their directory entry.

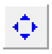
 Double click on the symbol name, if it is to be inserted in the drawing.

Search using the Symbol insert dialog box

1. Select the menu command ***Insert***, ***Symbol***.
2. Select a libary from the ***current library*** list.
 If no library is loaded, use the dialog box menu command ***Library, Add*** to load the library in which the symbol should be searched for or enter a library name and confirm with ***Enter.*** This library is loaded if it is in the current library directory. Otherwise a new library is created.
3. In the ***search for the following symbols*** text box, enter the name of the symbol which is being searched for.
 Please note that upper and lower case are differentiated between whilst searching. Use the (*) as the wildcard for the following

characters. Use the (?) as wildcard for one character.
All common search rules can be used as search criteria.

4. Start the search by clicking the command button ***Search now***.
5. Select the directory in which the symbol is to be saved by clicking on the respective one.
6. Tag the name of the symbol in the list of symbols.
   In the preview window pictures of the symbls are displayed.
   Using the list 

   it can be defined, who much previews are shown in the preview window.
   By clicking the icon  the geometry of the symbols is displayed as big as possible independent from the reference point.
7. Clich the command button ***OK*** and insert the symbol according to the settings determined using the menu command ***Settings, Symbols***.

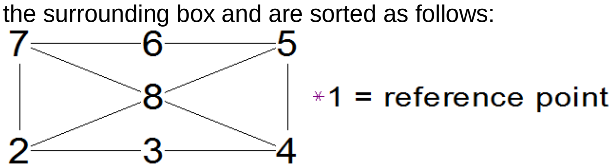
Insert the symbol according to the settings determined using the menu command ***Settings, Symbols***.
By typing the letter ***B***, or ***b*** you can switch to and from the box display and the complete display of the symbol’s contents.

If necessary, use one of the following additional options before positioning the symbol in the target point.

Keyboard entry

X	The objects found on the cursor are mirrored on the X-axis.
Y	The objects found on the cursor are mirrored on the Y-axis.
Left arrow key ←	The objects found on the cursor are rotated in an anticlockwise direction by the angle defined in the Status dialog box.
Ctrl + Left arrow key ←	The objects found on the cursor are rotated in an anticlockwise direction by 90°
Right arrow key →	The objects found on the cursor are rotated in a clockwise direction by the angle defined in the Status dialog box.
Ctrl + Right arrow key ←	The objects found on the cursor are rotated in a clockwise direction by 90°
Up arrow key↑	The objects found on the cursor are scaled larger by the factor defined in the Status dialog box.
own arrow key ↓	The objects found on the cursor are scaled smaller by the factor defined in the Status dialog box.

Ctrl+R	The objects found on the cursor are rotated dynamically about the cursor point.
Ctrl+S	The objects found on the cursor are scaled dynamically.
> **<**	switch between 7 additional insert points forward and backwards. The insert points will be calculated according to the surrounding box and are sorted as follows:



Please note:

When inserting a symbol, the display attributes of the objects within the symbol retain their validity, i.e. they are layer or object specific.

Object specific display attributes are kept.

The objects in the symbol may also take over the properties of the layer upon which the symbol was inserted, or the display settings which you have explicitly assigned to the symbol (layer-related or object-related display).

Otherwise the layers and objects are inserted in the following way: If the symbol contains layers, which have the same name as those in the model, the layer specific display attributes are taken over from the model.

If the symbol contains layers which do not exist in the model, these are added.

Select and insert symbol graphically

You will be able to select symbols both by name and call up a graphic display simultaneously by using the following method:

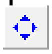
1. Activate ***Insert, Symbol***.

2. Select a libary from the ***current library*** list.
If no library is loaded, use the dialog box menu command ***Library, Add*** to load the library in which the symbol should be searched for or enter a library name and confirm with ***Enter.*** This library is loaded if it is in the current library directory. Otherwise a new library is created.

3. If necessary, you can open the directory structure in the left-hand section by clicking beneath the heading ***directory structure***

twice to open the directory tree structure together with any subdirectories that may exist.

4. In the ***search for the following symbols*** text box, enter the name of the symbol which is being searched for.
   Please note that upper and lower case are differentiated between whilst searching. Use the (*) as the wildcard for the following characters. Use the (?) as wildcard for one character.
   All common search rules can be used as search criteria.
5. Start the search by clicking the command button ***Search now***.
6. Click and select the directory containing the symbol.
7. Tag the name of the symbol in the list of symbol names displayed in the centre section.
   The preview window will show the symbols as small illustrations.
   You can use the list box 

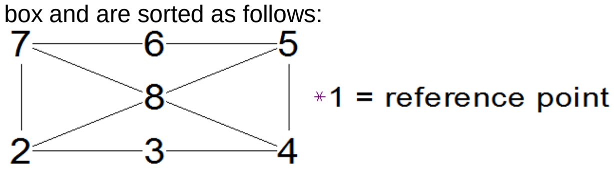
 to specify the number of fields to appear in the preview window.
   After clicking the icon , you will be able to display the symbol geometry independently of the position of the insertion point so that it will be as large as possible.
8. Click the ***Accept*** button and then add the symbol to the drawing using the defaults specified in ***Settings, Symbols***.
   By typing the letter ***B***, or ***b*** you can switch to and from the box display and the complete display of the symbol's contents.

   If necessary, use one of the following additional options before positioning the symbol in the target point.

Keyboard entry

X	The objects found on the cursor are mirrored on the X-axis.
Y	The objects found on the cursor are mirrored on the Y-axis.
Left arrow key ←	The objects found on the cursor are rotated in an anticlockwise direction by the angle defined in the Status dialog box.
Ctrl + Left arrow key ←	The objects found on the cursor are rotated in an anticlockwise direction by 90°
Right arrow key →	The objects found on the cursor are rotated in a clockwise direction by the angle defined in the Status dialog box.
Ctrl + Right arrow key ←	The objects found on the cursor are rotated in a clockwise direction by 90°
Up arrow key↑	The objects found on the cursor are scaled larger by the factor defined in the Status dialog box.

Down arrow key ↓	The objects found on the cursor are scaled smaller by the factor defined in the Status dialog box.
Ctrl+R	The objects found on the cursor are rotated dynamically about the cursor point.
Ctrl+S	The objects found on the cursor are scaled dynamically.
> **<**	switch between 7 additional insert points forward and backwards. The insert points will be calculated according to the surrounding box and are sorted as follows:



You can improve the overview by changing the size of the dialog box and of the three sections containing the directory structure, the symbol names and the symbol preview.
This is done by moving the cursor to the outermost edge of the window or to the vertical dividing line and then moving the cursor to its new position while pressing the left mouse button.

Please note:

When you are inserting a symbol, the display attributes of the objects within the symbol retain their validity, i.e. they are layer or object specific.

Object specific display attributes are kept.

The objects in the symbol may also take over the properties of the layer upon which the symbol was inserted, or the display settings which you have explicitly assigned to the symbol (layer-related or object-related display).

Otherwise the layers and objects are inserted in the following way: If the symbol contains layers, which have the same name as those in the model, the layer specific display attributes are taken over from the model.

If the symbol contains layers which do not exist in the model, these are added.

Load and insert symbols from the model

Using the ***Display Symbols In Model*** command from the ***Symbol Manager***, the symbols found in the model can be listed and repositioned.

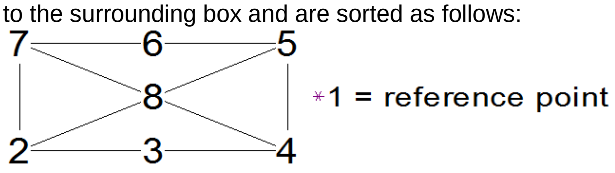
1. Select the menu command ***Define, Symbols.***
2. In the Symbol Manager, start the menu command ***Model, Display Symbols In Model.***
3. All the symbols that exist in the model are listed by name, independent of whether the respective symbol library is the symbol library currently accessed.
4. Double click on the entry in order to load the respective symbol.
5. Position the symbol in the drawing according to the settings determined using ***Settings, Symbols***.
   By entering the letter *B, b* whilst positioning, you can change the display between a box and the actual complete display of the symbol content.

   If necessary, use one of the following additional options before positioning the symbol in the target point.

Keyboard entry

X	The objects found on the cursor are mirrored on the X-axis.
Y	The objects found on the cursor are mirrored on the Y-axis.
Ctrl + Left arrow key ←	The objects found on the cursor are rotated in an anticlockwise direction by 90°
Left arrow key ←	The objects found on the cursor are rotated in an anticlockwise direction by the angle defined in the Status dialog box.
Ctrl + Right arrow key ←	The objects found on the cursor are rotated in a clockwise direction by 90°
Right arrow key →	The objects found on the cursor are rotated in a clockwise direction by the angle defined in the Status dialog box.
Up arrow key↑	The objects found on the cursor are scaled larger by the factor defined in the Status dialog box.
Down arrow key ↓	The objects found on the cursor are scaled smaller by the factor defined in the Status dialog box.
Ctrl+R	The objects found on the cursor are rotated dynamically about the cursor point.
Ctrl+S	The objects found on the cursor are scaled dynamically.
> **<**	switch between 7 additional insert points forward and backwards. The insert points will be calculated according

to the surrounding box and are sorted as follows:



Please note:

- When inserting a symbol, the display attributes of the objects within the symbol retain their validity, i.e. they are layer or object specific.
- The objects in the symbol may also take over the properties of the layer upon which the symbol was inserted, or the display settings which you have explicitly assigned to the symbol (layer-related or object-related display).

  Otherwise the layers and objects are inserted in the following way: If the symbol contains layers, which have the same name as those in the model, the layer specific display attributes are taken over from the model layer.
  If the symbol contains layers which do not exist in the model, these are added.

Disassemble symbols in model

If the symbols which exist in a model are to be disassembled, select the CAD Menu ***Process 2D, Disassemble Symbol*** or start the command using the icons

, .

This command can be used to disassemble symbols, i.e. disassemble them into their original 2D objects. The group structure contained in a symbol is moved to the group in which the symbol is included.

The symbol is disassembled one step at a time, i.e. if further symbols are contained in a symbol, these are not disassembled.

Use one of two options for selecting the required symbol:

With selection list
If symbols already exist in the selection list, these are disassembled directly after loading the command.

Without selection list

Use the ***Access to ... partial drawing*** command at the top of the 2D drawing window to define in which partial drawing the symbols to be disassembled should be determined.
Accept the symbols to be disassembled in the action list.

Please note:

If a symbol has been allocated attribute values, these are removed when the symbol is disassembled. This is not valid for attribute values which the symbol contains.

If you disassemble a symbol that has been scaled, the dimensions, finish characteristics, shape and position tolerances as well as weld/solder seam symbols in it will be regenerated. The dimension numbers will be adapted to their true values and the objects will be given their original measurements.

, Symbols can also be disassembled using the command ***Process2D, Disassemble 2D Objects***.

Completely disassemble symbols in model

If the symbols which exist in a model are to be completely disassembled, select the CAD Menu ***Process 2D, Disassemble Symbol*** or start the command using the icons

, .

This command can be used to disassemble boxed symbols, i.e. disassemble them into their original 2D objects.
The group structure contained in a symbol is subordinate to the group in which the symbol is embedded.

The symbol is disassembled one step at a time, i.e. if further symbols are contained in a symbol, these are also disassembled.

Use one of two options for selecting the required symbols:

With selection list
If symbols already exist in the selection list, these are disassembled directly after loading the command.

Without selection list
Use the ***Access to ... partial drawing*** command at the top of the 2D drawing window to define in which partial drawing the symbols to be disassembled should be determined.
Then identify the symbols to be disassembled.

Please note:

If a symbol has been allocated attribute values, these are removed when the symbol is disassembled. This is not valid for attribute values contained in the symbol.

If you disassemble a symbol that has been scaled, the dimensions, finish characteristics, shape and position tolerances as well as weld/solder seam symbols in it will be regenerated. The dimension numbers will be adapted to their true values and the objects will be given their original measurements.

,  Symbols can also be disassembled one step at a time using the command ***Process2D, Disassemble 2D Objects***.

Disassemble symbols whilst positioning

If you have activated the menu command ***Settings, Symbols*** and the ***Symbols when positioning*** option whilst disassembling, all symbols are disassembled automatically after being positioned. The symbols (nested symbols) contained in a symbol remain as symbols. The group structure contained in a symbol is subordinate to the group in which the symbol is embedded.

Disassemble completely
After you have activated the ***disassemble when positioning*** button, you can further determine whether nested symbols are to be disassembled into the lowest level. All symbols contained within the symbol are also disassembled with this.

Please note:

If a symbol has been allocated attribute values, these are removed when the symbol is disassembled. This is not valid for attribute values which the symbol contains.

If you disassemble a symbol that has been scaled, the dimensions, finish characteristics, shape and position tolerances as well as weld/solder seam symbols in it will be regenerated. The dimension numbers will be adapted to their true values and the objects will be given their original measurements..

Display symbols in model

If a list is to be displayed of all the symbols contained in the model, proceed as follows:

1. Select the menu command ***Define, Symbols***.
2. Start the menu command ***Model, Display Symbols in Model***.
3. In the dialog box which then appears, the names of the symbol libraries are listed including the complete directory path, from which symbols are to be taken over into the current model.

   If you activate the option ***details***, you will be shown the last saving date of a symbol in the library in addition to the number.

The ***show all*** option lists symbols with the number 0 whose graphic has been deleted from the model. Symbols without graphics remain in temporary memory during model editing. They are automatically removed when the model is closed. To clean up the list prematurely, you can use the Remove unused Symbols function.

Double click on the 🗀 icon in order to display the names and number of the individual symbols from the respective library.

Please note:

If symbols are inserted which are also to contain symbols, only the "outer" symbols are displayed.
If you double click on a symbol in the list, you can insert another symbol in the drawing. Simply drag the cursor into the drawing area and the positioning options.
If working a lot with symbols in a model and checking the number of symbols constantly, this dialogue window can be offered without the Symbol Manager by using a command in the user defined toolbar or a user defined menu.

In the dialog box ***Show Symbols in Model***, select a symbol and click ***Remove*** to remove the graphic of the symbol from the model completely.

Update symbols in model

Symbols which exist in a model remain unmodified, if the geometry or logical structure of the symbol is modified in the symbol library.

If the respective symbols in the model are also to be updated, proceed as follows:

1. Select the menu command ***Define, Symbols***.

2. Load the library whose symbols are to be updated in the model using the dialog box menu command ***Library, Add***.

Otherwise select from the ***current library*** list.

3. Start the dialog box menu command ***Model, Update Symbols In Model***.

4. All the symbols in the model, whose version is different to that of the selected library, are displayed in a dialog box. If no symbol has been changed, the display will be completely empty.

If only individual symbols of those displayed are to be updated, tag this in the list.

If all of the displayed symbols are to be updated, click on the ***Tag All*** command button.

5. If the update is to be carried out, click on the ***Update*** command button.
Otherwise, close the dialog box.

Please note:

The model symbols can only be updated with this menu command, if the complete library path and symbol name are identical in the model and library.
If this is not the case, the model symbols can be replaced by other symbols.

Allocated attribute values are also removed.

Replace symbols in model

To increase the speed for redraw, it is useful to use a simple symbol whilst processing the drawing rather than a complex one and then to replace this one with the required symbol just before the drawing is to be plotted.

There are other reasons as well, why existing symbols in a model should be replaced by other symbols.

Proceed as follows:

With selection list

1. Accept all the symbols into the selection list which are to be replaced by one and the same symbol.

2. Select the menu command ***Define, Symbols***.

3. Select a libary from the ***current library*** list.
If no library is loaded, use the dialog box menu command ***Library, Add*** to load the library in which the symbol should be searched for or enter a library name and confirm with ***Enter.*** This library is loaded if it is in the current library directory. Otherwise a new library is created.

4. Select the directory, then tag the name of the symbol that is to replace the one already in the model.

5. Start the dialog box menu command ***Model, Replace Symbols In Model***.

With action list

1. If the selection list does not contain any symbols, select the menu command ***Define, Symbols***.
2. Select a libary from the ***current library*** list.
 If no library is loaded, use the dialog box menu command ***Library, Add*** to load the library in which the symbol should be searched for or enter a library name and confirm with ***Enter.*** This library is loaded if it is in the current library directory. Otherwise a new library is created.
3. Select the directory, then tag the name of the symbol that is to replace the one already in the model.
4. Start the dialog box menu command ***Model, Replace Symbols In Model***.
5. If only one individual symbol is to be replaced, identify it with the cursor. If a number of symbols are to be replaced, take them over in the action list.

Please note:

The new symbols accept the position of the reference point, the scaling factor and the rotation angle of the previous ones.

Allocated attribute values are also removed.

Count symbols in model

The number of positioned symbol in the current drawing can be exported as a text file. The name of the symbol and the positioned symbols are listed in the drawing. The file with the extension *.TXT can be opened with normal word processing programs.

1. Select the menu command ***Define, Symbols***.
2. Start the menu command ***Model, Count Symbols***.
3. Select the directory in which the file is to be saved.
4. Define a file name for the text file.
5. Confirm the ***Save*** command button.

Open the *.TXT file with one of the word processing programs installed on your computer.

Reassign Libraries

Symbols within a model are identified using their names and the name, including the path of location, of the library. If a library is moved to a different location or is renamed, the symbols with a reference to this library cannot be updated any more. In this case use the option ***Reassign Libraries…***, to reference the symbols to the current location and name of the library.

1. Select the menu command ***Define, Symbols***.
2. Inside the dialogue window select the menu command ***Model, Reassign Libraries …***.
3. Select from the list of referenced libraries those you want to assign to another library.
4. Click the ***reassign*** button.
5. Select the location and the file name of the new library.
6. Confirm by clicking the ***Open*** button.

Remove unused Symbols

If you delete a symbol within a model and no identical symbol is in the model, this symbol will still remain in a temporary cache during the current session. To erase this symbols also from the temporary cache even during your current session please follow this steps:

1. Select the menu command ***Define, Symbols***.
2. Inside the dialogue window select the menu command ***Model, Remove unused Symbols …***.

Please note:

If a new Model is loaded or if the program is left, the unused symbols are also removed.

Attribute Formula for the Number of Symbols

In addition to the standard text attributes and numerical attributes you can create free configurable attribute formulas as described here.

Using the command '`SymbolCount`' a user attribute can be defined, which counts the number in a group, a partial drawing, a drawing and the complete model or creates a list of this symbols including their number.

Preconfigured formulas can be found in the dialogue window Define Attributes for *Symbol Count Sketchboard / Group* and *Symbol List Sketchboard / Group*.

Further free formulas can be defined using the following parameters.

```
SymbolCount(opAtt,where,list,symbolName='',format='',eWithPath=0)
```

Parameter:

`opAtt`	fix
`where`	in which structure the symbols should be found? 0 – group 1 – partial drawing 2 – drawing 3 – model
`list`	list or total amount? 1 – create list of all symbols with name and path 0 – total amount of all symbols
`symbolName`	Regular expression, describing the names of the symbols, which should be listed or '' for ‚all symbols' 1)
`format`	Format for the output, or '' for standard value
`eWithPath`	List symbol names including path? 0 – erase symbol path 1 – show symbol path

1) Example: `SymbolCount(opAtt,0,0,'.*/Chair_round')` = total amount of all symbols with the name "Chair_round" of the selected group. The symbol name always contains the path structure.

Examples:

Count of symbols in a group:

```
SymbolCount(opAtt,0,0)
```

Count of symbols in a model:

```
SymbolCount(opAtt,3,0)
```

List of all symbols in a partial drawing:

```
SymbolCount(opAtt,1,1)
```

List of all symbols in a drawing, where the name starts with c or C:

```
SymbolCount(opAtt,2,1,'.*/[cC][^/]*','%-10s : %3i',0)
```

```
symbolName : '.*/[cC][^/]*'
```

`.*/` any character string (.*) up to a /
`[cC]` small c or capital C
`[^/]*` any character string unequal /

```
format : '%-10s : %3i'
```

`%-10s` allocate 10 characters for the symbol name, adjust the rest left-aligned

`%3i` allocate 3 characters for the symbol count, adjust the number right-aligned

Count of all symbols in a drawing, where the name starts with c or C:

```
SymbolCount(opAtt,2,0,'.*/[cC][^/]*')
```

Rename symbol in library

In order to rename symbols saved in a symbol library, proceed as follows:

1. Select the menu command ***Define, Symbols***.
2. Select a libary from the ***current library*** list.
   If no library is loaded, use the dialog box menu command ***Library, Add*** to load the library in which the symbol should be searched for or enter a library name and confirm with ***Enter.*** This library is loaded if it is in the current library directory. Otherwise a new library is created.
3. Under ***directory structure*** open the directory tree and if necessary further subdirectories by double clicking on the  icon.
4. By clicking on the directory, determine the directory.
5. Tag the symbol name in the list under ***symbols***.
6. Start the dialog box menu command ***Symbol, Rename***.
7. In the ***new description*** text box, enter the name for the selected symbol.

Erase symbol from library

You can use the following method to erase symbols from a symbol library:

1. Select the menu command ***Define, Symbols***.
2. Select a libary from the ***current library*** list.
   If no library is loaded, use the dialog box menu command ***Library, Add*** to load the library in which the symbol should be searched for or enter a library name and confirm with ***Enter.*** This library is loaded if it is in the current library directory. Otherwise a new library is created.

3. Under ***directory structure*** open the directory tree and if necessary further subdirectories by double clicking on the icon.
4. By clicking on the directory, determine the directory.
5. Tag the symbol name in the list under ***symbols***.
6. Start the dialog box menu command ***Symbol, Erase***.

Erase symbols in model

Like any other object, symbols can be deleted from a model using the ***Object/Erase Selection*** command. For the case that a symbol exists a number of times in one drawing and all of these are to be erased, you can use the following procedure to delete them:

1. Select the menu command ***Define, Symbols***.
2. In the ***Model*** menu, select from the Symbol Manager, the option ***Display Symbols In Model.***
3. All the symbols that exist in the model are listed by name. The number of symbols in the model is listed in brackets behind the names.
4. Tag a symbol in the list and confirm the ***Delete command*** button. All symbols are then deleted from the drawing.

4.17 Use 2D standard parts

BeckerCAD includes a 2D standard part library, which currently contains:

holes
screws
nuts
plain washers/locking washers
pins
thread pins
through bolts
push-in bolts
welding neck flanges
roller bearings
steel construction profile

2D objects which are saved in a symbol, are scaled in the same way each time the symbol is scaled, i.e. for a screw saved as a symbol, a modification to the length also means a modification to the width and head.

Like a symbol, a 2D standard part is also managed as an object. However - different to symbols - the dimensions of the part geometry belonging to a standard part can be determined in any way within the limits of the predefined options. The possible dimensions are available in a database.

group
When inserting a 2D standard part, this is included in the current group.

colour, line type, line width
The geometry lines of the standard part are linked with the current layer for geometry, the centre lines with the current layer for centre lines and, if necessary, for thread lines the current layer.
Accept the display attributes from these lines (colour, line type, line width). The display attributes can only be modified by modifying these layer attributes.

If only 2D standard parts are to be selected using the cursor, activate 2D standard parts using the menu command ***Settings, Object Filter*** and deactivate all other objects.

Please note:

Dimensions can be brought onto 2D standard parts as point or object dimensions.
Using other geometry, only limited object points can be snapped on 2D standard parts, namely: ***endpoint, midpoint, centre, quadrant point, snap point, text reference***.

Select and insert 2D standard part

If 2D standard parts are to be inserted in the drawing, start the menu command ***Insert, 2D Standard Part***.

Selection of standard part

1. In the window headed ***Selection*** select the type of standard part required by clicking, e.g. *plain washers/locking washers*.
   In the ***Selection tree*** list, the selected type, in this case *plain washers/locking washers*, is displayed like a subfolder under *BeckerCAD 2D Features*.
2. In the ***Selection*** list, determine - if necessary, in a number of steps - by clicking, the exact type of standard part which is to be inserted, e.g.
   DIN 125 washer
   DIN 125a washer, without chamfer

These specifications are also taken over in the window with the heading ***Selection tree***.

If you have defined the general design for bolts, their individual parts are added to the ***Selection tree***.
Activate the command by clicking on the individual part in the selection tree if you wish to determine its type. The possible types are then displayed in the ***Selection*** window.

3. If, accidentally, an incorrect selection has been made, in the ***Selection tree*** click on the name where the selection was still correct (higher levels in the structure tree).
For example, you have selected *DIN 125a washer, without chamfer* but want to select *DIN125b washer, with chamfer* so click on *DIN 125 washer* in the selection tree.
The required washer can then be clicked on in the *selection* list.

   For *bolted assemblies*, if only the exact determination of an individual type is to be modified, click on it and then tag the required type under *Selection*.

4. Under the ***Parameters*** heading, click on the line with the required data or for *rolling bearings* the type name.

   If the ***Graphic preview*** check box is active, a standard part of the selected type is displayed in the bottom right block.

   Under the ***Graphic preview*** window, select one of the possible views of the selected standard part.

5. Complete or, if necessary, modify the necessary data for the dimension of the standard part:
To do so, use the ***Variable values*** command button to open a dialog box in which the entries can be determined.

Insert standard part

1. Select one of the predefined options (*0°*, *90°*, *180°* , *270°*) or enter the value in the ***angle*** text box.
By activating one of the *+10°* or *-10°* options, the current angle can be enlarged or reduced by the entered value.

2. For screw and bolts whose side views are to be inserted, the angle and length can be determined dynamically when positioning.
Activate the setting ***Position dynamically***.
If this setting is active, the entered angle is ignored.

3. Activate the command by clicking on the ***Generate*** command button.

4. Define the point on which the standard part is to be inserted.

5. If a screw or bolt is inserted and the ***Position dynamically*** setting is active, the angle and length can be determined dynamically.
In the case of fastenings when you specify the length you will also be defining the length of the counter bore.

The length of the screws or bolts are modify themselves according to the permitted lengths determined in DIN.

Position dynamically

When this setting is active, you can dynamically specify the angle and length of the standard part, depending on the view that has been chosen. If it is not active, the values will be specified according to the settings in the dialog box.

Edit 2D standard part

  After clicking on the displayed icons, the ***Process 2D, Edit 2D Standard Part*** command is started.

2D standard parts which have been inserted into drawings can be edited, i.e. their dimensions, view and position can be modified.

1. Identify the standard part to be processed.

2. The dialog box for the selection of 2D standard parts is then displayed. The standard part is displayed with the previously allocated dimensions.

3. If an individual part of a bolt is to be edited, click on this in the ***Selection tree*** window and, if necessary, select another individual part from the ***Selection*** window.

4. Under the ***Parameters*** heading, click on the line with the required data or, for *roller bearings*, the type name.

5. If necessary, change the view by selecting the new one from the list below the preview window.

6. Modify the position, by clicking on one of the options with the predefined angles or enter the required value in the ***angle*** text box.

7. Accept the modifications with ***OK*** or lose them with ***Cancel***.
Then the old standard part will be erased and replaced by a new one in the same position.

Disassemble 2D standard parts

,  The command can also be loaded using the displayed icon.

If 2D standard parts are to be processed, e.g. individual lines are to be erased, this is only possible if this standard part is disassembled.

Action - selection

1. If the selection list does not contain any 2D standard parts, load the command ***Process 2D Object, Disassemble 2D Standard Part***. Accept the 2D standard parts which are to be disassembled into the action list.

Selection - action

1. Accept the 2D standard parts which are to be disassembled into the selection list.

2. Load the command ***Process 2D Object, Disassemble 2D Standard Part****.*

Please note:

After disassembling, standard parts cannot be edited.

The exception: fastenings. After any of these has been disassembled once, you will still be bale to edit the separate components as standard parts.

After you have disassembled a fastening, a group will be created within the group structure and given the name 2DFea that will contain all the separate components used for the fastening. If a single standard part disassembled, a group with the name 2DFea will be created and it will contain all the objects in this standard part.

,  2D standard parts can be disassembled using the command ***Process2D, Disassemble 2D Object***.

Erase 2D standard part

2D standard parts can be erased using the command ***Erase Object/Selection***.

4.18 Use 3D-views

You can create the 2D views on the basis of the solids created inside the 3D-view window by first inserting **model views** of the solids into the 2D drawing from the 3D-view window.

Process Model Views

from ***BeckerCAD 3D***

 After clicking the displayed icon the icon bar ***Process Model Views*** is offered.

Using the commands in this icon bar you can create and edit section views, detailed views and partial views.

Also the commands to insert and delete solids in existing model and section views are contained in this icon bar.

Use model views

A model view is an object which is **associative** to the original solids in the 3D-view windows.
With a model view, it is not dealing with a "true" 2D object but with a 2D illustration of 3D objects.

Each model view forms a unit, which are brought into and drawn in a model in the following way:

Group
The generated objects are saved in the **current group**. This can be selected from the existing groups using group selection in the main toolbar. Group allocation of the objects can later be changed using the ***Model Explorer***. The groups to which the objects belong can later be modified using the ***Model Explorer***.

Colour, line type, line width
The visible solid edges and silhouettes in a model view are allocated the display attributes of the *Current drawing layer* or - if activated - the object specific attributes. Alternatively the colour can be assigned to the related 3D-objects.

The invisible edges and silhouettes are allocated the display attributes of the layer, which has been determined using ***Define, Layers*** as the layer for ***hidden edges*** or using ***Settings, 3D/2D Commands*** on the ***Model view*** option card. These attributed cannot be object specific or modified.

Individual display
Model views are generated using the parameters defined in ***Settings***, ***3D/2D Commands*** in the option card ***Model View***. The individual display can be altered as follows:

1. ,  Start the command ***Process Model Views, Edit Model View.***
2. Identify a model view.
3. Deactivate the option ***View according to global settings.***

4. Define the individual settings.

Centre lines in model views can be generated using the menu ***Centre Lines*** and the commands ***Centre Cross To Circle*** and ***Centre Lines To Parallels***.

The hidden edges and silhouettes are linked to the display properties in the ***Hidden Edges (3D) Layer*** defined using the ***Layer Explorer*** inside the ***Model Explorer*** or defined in ***Settings, 3D/2D Commands*** in the option card ***Model View***. This properties cannot be defined as object related properties.

To ensure that whilst processing your drawing, only model view can be identified, use ***Settings, Object Filter*** to activate the snap possibilities for model views and deactivate the other object types.

Updating

The menu command ***Settings, 3D/2D Commands*** can be used on the ***Model View*** option card to determine whether the views of models are to be updated automatically after each alteration to a 3D model, or whether it is to be selected manually by clicking the icon shown here as a red traffic signal in the 2D symbol bar.

Update single Model / Section Views

, Using the command ***Process Model Views***, ***Refresh Model View Separately*** single model and section views can be updated.

This can be useful, if a drawing consists of several model/sevtion views from different solids from the assembly. So a modification to a single solid doesn´t need to update all views.

Save and open model with model views

When a model is saved, the current status of the model views in it will be saved in the form of graphics.
This will mean that the models can be loaded much quicker, because the illustrations will not have to be calculated directly. In addition, 2D drawings with model views can be loaded without the solids they contain.

You must remember that when you load additional drawings from other models the model views will have no link to the associated solids.

Please note:

> If a model view is to be processed (add or erase 2D line objects, extend, shorten, interrupt, etc.), this must firstly be converted into “true“ 2D line objects using the command ***Process Model Views, Disassemble Model View*** or using the command ***Process2D,***

Disassemble 2D Object.
With this, freely editable 2D line objects are generated from the model view.
These no longer have a connection to the solids of the model in the 3D-view window and therefore are **not associative**.

If a section view is to be made from a model view, the model view cannot firstly be disassembled and erased at the same time.

Determine settings for model views

The 3D-view window can be used to derive the model views of solids and inserted in the 2D-drawing window.
The model view settings can be determined by means of the 2D symbol bar or in the menu command ***Settings, 3D/2D Commands***, on the ***Model view*** option card.

Settings, 3D/2D commands, Model View option card

This option card is for completing the following settings for model views:

display hidden lines on layer

This list field is for opening the layer selection.
You can use this option to specify a layer for linking all the hidden edges of all the model views. You can also use the Layer tree in the Model-Explorer to specify this layer.

display transition edges on layer

Tangential Edges in model views, so called transition edges, can be aligned to a predefined layer and can be displayed shortened.
 The layer for transition edges can be defined in the global settings for model views as well as in the **Layer Structure Tree** of the **Model Explorer** as described in Define layer change for special objects.

shorten by

Transition edges are drawn with the distance to the contour line defined here. If this value is higher than half of the line, the line will be shortened by half of its length.

Suppress double lines

This setting is for controlling how the hidden edges are displayed: if such edges are in the viewing direction behind a visible or hidden edge, the only edge to be displayed will be the nearest one in the viewing direction.

Display komplex hidden lines

If hidden lines are not displayed, even though the option display hidden lines is active, maybe it is a border case of calculation. By activating this command maybe also the missing lines can be displayed.

Display tangential lines

This must be used when tangential lines are to be displayed. Tangential lines occur, when the touching edge of two faces is rounded for example.

Display silhouettes

This can be used to display not only the model views with the solid edges but also the silhouettes, i.e. the contour lines of rounded solid surfaces that depend on the view chosen.
Points (endpoint, centre, snap point) can be snapped on these lines.

Display model view tagged

This can be used to determine whether the model views are to be displayed in "standard" colours or light grey, so that they can be distinguished from the "normal" 2D line objects.

Erase model view when disassembling

When ***Process Model Views, Disassemble Model View*** or using the command ***Process2D, Disassemble 2D Object*** is being used a copy of the model view consisting of 2D line objects will be made. This setting will determine whether or not the model view is to be erased immediately afterwards.

Disassemble Model Views to Groups

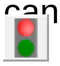
If the option ***Disassemble Model Views to Groups*** is activated in ***Settings***, ***3D / 2D Commands*** in the tab ***Model View***, a group structure is created for every disassembled model view.
The name for the main group of a disassembled model view is taken from the 3D Space (3D Solids). Below this main group a subgroup is created for every solid. If a group structure is already defined in the 3D Space, this structure will be built below the main group.

Box mode in dynamic transformations

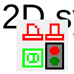
Model views and section views are usually displayed in a simplified box mode while dynamic transformations (Move, Copy, Rotate, Mirror, Scale) If the option ***Box Mode when Moving Views*** is deactivated in ***Settings, 3D / 2D Commands*** in the tab ***Model View*** in dynamic transformations, all the lines of model and section views are displayed and points can be identified on this lines.

Update model view automatically

When this setting is active, all the model views and section views will be re-calculated after alterations to the 3D model solids. In the case of complex solids or large numbers of views this procedure can take a considerable amount of time.



If you deactivate the setting, the update procedure can be started manually as required by clicking the red traffic light icon in the 2D symbol bar or for single views

,  using the displayed icons ***Process Model Views***, ***Refresh Model Views*** separately.

Take over 3D colors

Model and section views take over the color from the solids if this option is activated.
Please note:
If a model or section view is disassembled, which was displayed in 3D colors, the lines will also take over the solid color as object property. The layer property color will be ignored in this case.

Activate the option ***only in orthonogal views*** if the centre lines should only be displayed in an orthonogal view on the side view or top view of the geometry. Otherwise the centre lines are also displayed in a perspective view for example.

The display properties colour, line type and line width are assigned to the current layer for centre lines. The overhang is defined in the global settings for 2D axes, which can be adjusted in ***Settings***, ***3D / 2D Commands*** in the tab ***2D / 3D Axes***.

Taking over 3D colours

The colour of the geometry in model and section views can be take over to the colour of the related 3D objects by activate this setting.

Please note:
When disassembling model or section views displayed in 3D colours, the line elements will remain in the colour as object display. The layer colour will be ignored in this case.

Display quality of silhouettes

The quality of the display of curved silhouettes can be defined using the sliding control.
Curved silhouettes are approximated using polygons. The polygon segments are smaller or longer and the approximation is better or worse.

Example:

- Generate a torus in a 3D window.

- Derive model views from this torus: 3 standard views + perspective view
- Open the dialog box ***Settings***, ***3D/2D Commands***
- option card ***Model View***
- Alter the display using the slide control
- command button ***Accept***
In the perpective view the difference can be seen best.

Please note:

The individual display of model views can be changed:

1. ,  Start the command ***Process Model Views***, ***Edit Model View***.
2. Identify a model view.
3. Deactivate the option Display according to global defaults.
4. Change the parameters as needed.
   If in the global settings (menu command ***Settings, 3D-/2D-Commands,*** option card ***Model View***) the option ***update model views automatically*** is activated, the changes are displayed immediately after clicking the button ***Accept.***

Insert model views in 2D drawing

If views of solids are to be transferred to the 2D drawing, activate the required 3D-view window and determine the required settings for model views.

Follow this procedure:

1. Select the menu command ***Insert, Model Views***.
2. In the menu which then appears, determine which views are to be transferred:

   In addition to displaying individual views you can select any of the following combinations of views supported in their positioning by the cursor movements:

   The ***3 Standard Views (DIN)*** command transfers the top view, front view and side view from the left. The ***3 Standard Views+Persp (DIN)*** command transfers the same 3 as well as the standard perspective view.

   The corresponding ANSI views provide you with the underneath view, the side view from the right and the standard perspective view.

3. Accept the solids into the action list whose view(s) are to be transferred in the 2D drawing.
If the 3D model is structured in groups the desired solids can also be selected with the ***Group selection mode***.

   The 3D box is then calculated which contains all selected solids. A 2D box with diagonals is calculated from this (rectangle or parallelogram) and the size and position of the (first) view is displayed.

4. Activate the required 2D drawing.
5. If necessary, activate the partial drawing in which the view(s) are to be inserted.
6. Determine the position of the box. If one of the menu commands have been selected with which a number of views are generated, this contains the top view (DIN) or the under view (ANSI).
7. If one of the menu commands have been selected with which a number of views are generated, position the box which contains the front view.

   In reference to the position of the first view – this box can only moved upwards or downwards.

8. Position the side view from the left (DIN) or the side view from the right (ANSI).
The box containing the views can only be moved to the right or the left – in relation to the position of the front view.
9. If the menu command ***3 Standard Views+Persp*** has been selected, determine the position of the standard perspective view.

Please note:

If a model view contains the current view, this is aligned in such a way that edges, which run parallel to the Z axis of the GCS in the 3D window, run parallel to the Y-axis in the 2D window.

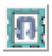
A model view does not contain "real" 2D objects, but illustrations of solids.
Illustrations of solid edges that are displayed as lines, circles or arcs, can be given a length, a diameter or a radius dimension.

When a solid is changed in the 3D model, and if, at the same time, the edges and their projection lines remain unchanged, the program will try to retain the dimensions and adjust them.

In order to generate "real" 2D objects from model views, use the command ***Process Model Views, Disassemble Model View*** or using the command ***Process2D, Disassemble 2D Object***. If the original model view is erased by doing so, any existing intersection lines will also be erased for a sectional display.

Shaded Model Views

from ***BeckerCAD 3D***

 Using the ***Insert***, ***Shaded View*** menu command, you can create a photo of your 3D model and place it as a pixel image in a drawing, as long as there are bodies in the 3D area and they have not been hidden by groups. The Shaded View displays the actual state of the current design; subsequent changes in the 3D area are not automatically updated.

Shaded views can be created from existing model views or from bodies in the 3D area. If you identify a model view, it is highlighted with the shaded view. For views of bodies in the 3D area, the current view is used.

If you want to place a *Shaded View* exactly on a 2D drawing, you can work with additional reference points in the 3D area and in the 2D drawing.

Please note:

The pixel image of a *Shaded View* is not inserted into the current group, but into the current partial drawing. If necessary, move the pixel image to a desired group afterwards.

The pixel images of *Shaded Views* are saved with the model and simultaneously moved to the default folder for pixel images when they are created. You retain a reference to the paged JPG file. External changes to the paged out image file will be applied the next time the model is opened.

If the pixel images of the *Shaded Views* contained in a model are not found in the original directory or in the specified folder for pixel images, the pixel image is moved to the default folder for pixel images when a model is opened.

Settings for Shaded Views

from ***BeckerCAD 3D***

After starting the ***Insert***, ***Shaded View*** menu command, you can make the following settings:

DPI (dots per inch)
Input field for the resolution of the dot density for the pixel image. The

higher the resolution, the better the output quality.
Please be aware: The file size of the pixel image, which is also stored in the model, also increases with increasing resolution.
Recommended setting: between 100 and 300 DPI

Size in Pixel
Display of the resulting size in pixels.

Size
Display of the resulting dimension of the pixel image in relation to the scale of the current partial drawing.

Model View
Start the command to add a shaded view to an existing model view.

Solids
Start the command to create a shaded view of selected bodies in the 3D window in the current view. After selecting the bodies to be displayed, the size of the pixel image is recalculated.

Antialiasing
With ***Antialiasing***, you can reduce the so-called step effect, which can occur when edges run diagonally on the screen, in various degrees. The higher the value, the more the edges are smoothed. The setting is taken from the global settings in the ***Settings*** menu, ***3D Display*** on the ***Solid Display*** tab and can be individually adjusted for the individual shaded view.

Drawing Order
The drawing order (-100 ... 127) defines which priority the shaded view should have over other 2D objects when drawing. The Shaded View covers all objects that have a lower drawing order.

Current View
Use this button to transfer the current view from the 3D window to the preview window. The size of the pixel image adapts to the new view at that moment.

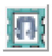
Reposition 3D and 2D
Used to precisely position a shaded view between a reference point in the 3D area and a point in a drawing; for example, to place a shaded view congruently behind a drawn view.

Reposition 2D
For positioning without a 3D reference or the previously defined 3D reference. Place the *Shaded View* at a desired position in the 2D drawing.

Preview of the 3D View
In the dialog you can see a preview of the pixel image that will be created.

Insert Shaded View

from ***BeckerCAD 3D***

 A ***Shaded View*** is a kind of photo of bodies from the 3D area, which can be placed in a drawing in the form of a pixel image. The shaded view can be created from an existing model view, the current view in the 3D window, or any other view. The different procedures are described below.

To insert a *Shaded View* into the active partial drawing, proceed as follows:

Shaded View from Model View

In order to add a photo of the 3D representation to a previously created model view, proceed as follows:

1. Activate the drawing with the desired model view.
2. Select the menu command ***Insert***, ***Shaded View***.
3. In the ***Shaded View*** dialog that opens, make the desired settings as described in the previous chapter.
4. Click the ***Model View*** button.
5. Identify a model view in the drawing. The view of the bodies contained in the model view is transferred from the 3D area to the preview of the dialog if the bodies are not hidden via groups.
6. Click the button ***Create*** or ***Create & Close*** to exit the command.

Creating a Simple Shaded View

Use the following steps to place a photo of any view of all visible bodies from the 3D area on a drawing.

1. Start the menu command ***Insert***, ***Shaded View***.
2. Select the desired view for the photo in the 3D window.
3. Click the ***Current View*** button if you want to include a view changed in the 3D window in the preview of the opened dialog.
4. If necessary, activate the drawing in which you want to insert the shaded view.

5. Click the button ***Reposition 2D***.
6. Move the cursor to the drawing. A frame indicates the dimension of the pixel image. Position the frame on the drawing at a desired position.
7. Click the button ***Create*** or ***Create & Close*** to exit the command.

Shaded View from selected Solids

Use the following steps to create a shaded view, if desired fitting exactly to a 2D view of selected bodies.

1. Activate the 3D window.
2. Select the desired view for the photo.
3. Start the menu command ***Insert***, ***Shaded View***.
4. Click the ***Solids*** button.
5. Select the solids for the shaded view.
   Either identify a single body, collect different bodies while holding down the *Ctrl* key, or select all visible bodies by section.
6. Specify a reference point on a solid that you want to use to place the *Shaded View* exactly in the drawing, if desired.
7. Switch to the 2D drawing e.g. via ***Ctrl+TAB*** or the ***Windows*** menu.
8. If you want to place the *Shaded View* congruently to a 2D drawing, define the 2D reference point in the drawing at the same geometry as before on the solid.
9. Click the button ***Create*** or ***Create & Close*** to exit the command.

Position several Shaded Views exactly to fit

If you want to place several different shaded views of all visible bodies from the 3D area one after the other in one drawing, proceed as follows:

1. Activate the 3D window.
2. Select the desired view for the first photo.
3. Start the menu command ***Insert***, ***Shaded View***.
4. Click the ***Reposition3D and 2D*** button.
5. Specify a reference point on a solid that you want to use to place the *Shaded View* exactly in the drawing.

6. Switch to the 2D drawing e.g. via ***Ctrl+TAB*** or the ***Windows*** menu.

7. Define the reference point in the 2D drawing.

8. Click the ***Create*** button.

9. Switch back to the 3D window.

10. Select the desired view for the next photo.

11. Click the ***Current View*** button to include the view in the preview.

12. Click the ***Reposition3D and 2D*** button, if you wish to define a new reference point in 3D

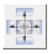
or

click the ***Reposition 2D*** button to continue with the previously defined reference point.

13. From here on, the previous steps are repeated. Depending on the previous choice, you can define the reference points in the 3D area and the 2D window one after the other or only the reference point in the drawing and ***Create*** the shaded view.

Parallel views to model views

from ***BeckerCAD 3D***

,  Using the displayed icons ***Process Model Views***, ***Parallel View from Model View*** parallel views to model views and section views can be created.

1. Activate the command ***Process Model Views***, ***Parallel View from Model View***.

2. Identify a model view or section view to get a parallel view from.

3. By default the parallel views are created according to the european standard DIN.
If a model view is placed right beside the identified view, a view from the left side is created. Click the ***X*** key, to create the views according to the ANSI standard.

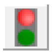
4. Move the cursor in the direction where the view should be placed. The views are linked horizontally or vertically. If the ***Shift*** key is pressed, the view from behind is created, if the ***Ctrl*** key is pressed, the view from the front is created.

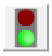
3. The action can be repeated as often as needed for other model views until the action is cancelled.

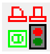
Update model views

If the menu command ***Settings, 3D/2D Commands*** on the ***Model View*** option card to activate ***Update model view automatically***, the views will be recalculated after each alteration to the 3D model.

You can avoid any distortions that might result from this process by deactivating the setting and completing a manual update.

 This is done by clicking the symbol with the red traffic signal in the 2D symbol bar.
This icon will only be displayed if the update procedure is to be completed manually and if alterations have been made in the 3D model.

 If the green traffic signal is displayed, the model views will correspond to the 3D model.

,  If the parameter ***Update model views automatically*** is not activated in ***Settings***,***3D/2D Commands*** in the tab ***Model View***,you can precisely update a single model view using the command ***Refresh Model Views separately*** in the ***Process Model Views*** (from ***BeckerCAD 3D***) icon bar.

Please note:

Clicking the green traffic signal again: the model views are recalculated and updated.

Convert model views in 2D line objects

  The command can also be loaded using this icon.

If individual lines are to be processed from the model views which have been inserted into the 2D drawing window, these must be disassembled, therefore into "real" 2D line objects.

Follow this procedure:

1. Select the menu command ***Settings, 3D/2D Commands*** and change to the ***Model view*** option card.
   Using the ***erase model view when disassembling*** check box to determine the required setting.
2. Accept all model views to be converted into the selection list.
3. In the CAD Menu, activate the ***Process Model Views >, Disassemble Model View*** command.

4. If model views already exist in the selection list, 2D line objects are generated from them.

Otherwise accept the model views to be disassembled in the action list.

Depending on the selected setting, the original model views remain or they are erased.

The 2D objects generated when disassembling a model view are inserted in the model structure in the following way:

Group
All objects are saved in the **current group**. This can be selected from the existing groups using group selection in the main toolbar.

The groups to which the objects belong can later be modified using the ***Model Explorer***.

Colour, line type, line width
The 2D objects generated from visible edges and silhouettes are linked to the **current drawing layer**. This can be selected from the defined layers using layer selection in the main toolbar.

If the invisible edges and silhouettes are to be displayed, the 2D objects generated from them are linked with the layer, which is determined as the layer for *hidden edges (2D)* using the ***Layer Explorer*** inside the ***Model Explorer***.

Thread lines and centre lines are assigned the the respective layers defined in the ***LayerExplorer***.

If for the model or section views the option ***Taking over 3D Colours*** is activated, the colour as object property is transferred to the disassembled line objects. The colours defined in the layer properties are ignored. The object properties for line objects can be edited later using the menu command ***Process***, ***Object Display***.

Please note:

Model views form the requirement for the generation of sectional displays. If they are disassembled and erased at the same time, they are no longer available for the menu command ***Insert, Section views***.

If, from a model view, a sectional display has been generated with intersecting lines, the intersection lines are also erased if the model view is disassembled and erased at the same time. The sectional display itself is retained.

, Model views can be disassembled using the ***Process2D, Disassemble 2D Object*** command.

Use sectional views

from ***BeckerCAD 3D***

If a model view of one or more solids has previously been inserted in the 2D drawing window, sectional views can be generated from the model view at the same time as an sectional identifier.

When you are creating a sectional view, two objects are generated: the sectional identifier and the actual sectional display.

They are inserted and drawn in the structure of a model in the following way:

Group
Both objects are saved in the **current group**. This can be selected from the existing groups using group selection in the main toolbar.
The groups to which the objects belong can later be modified using the ***Model Explorer***.

Colour, line type, line width
The solid edges and silhouettes in the sectional display are allocated the display attributes of the ***current drawing layer*** or - if activated - the object specific attributes.

If the display contains a hatching, this is linked with the hatching layer defined using the ***Layer Explorer*** inside the ***Model Explorer***. and receives its display attributes from this layer.
Thread lines and centre lines are assigned the the respective layers defined in the ***LayerExplorer***.

The elements of the sectional identifier are also linked with the currently selected *dimensioning layer*.
However, they are displayed in the colour defined using ***Settings, 3D/2D Commands*** on the ***Section View*** option card.
With the ***Process 2D Object, Process Section View*** command, the format of the intersection line can later be modified.

Update

The menu command ***Settings, 3D/2D Commands*** can be used with the ***Model View*** option card to determine whether model views and section views are to be updated automatically after an alteration to the 3D model, or whether this is to be completed manually using this red traffic light symbol in the 2D symbol bar.

If for the model or section views the option ***Taking over 3D Colours*** is activated, the colour as object property is transferred to the disassembled line objects. The colours defined in the layer properties are ignored. The object properties for line objects can be edited later using the menu command ***Process***, ***Object Display***.

Please note:

The section identifier is linked with two objects:
with the model view on which the intersection line is defined and with the respective sectional display. If one of these objects is erased, the sectional identifier is also erased.

However, if the model view is disassembled and erased at the same time, the identifier for the run of the cut is retained.

If the sectional display is allocated a hatching, this hatching can be processed or erased.

If the sectional display is inserted without hatching, it can be inserted later. To do so, identify the sectional display using the contour lines. Then identify one of the faces to be hatched.

Determine settings for sectional view

Using the menu command ***Settings, 3D/2D Commands***, the ***Section view*** option card can be used to determine settings for the ***Process Model Views, Section Views*** command and the ***Drawing Additions, Draw Section Run*** command.

These values are suggestions which can be modified when defining a sectional view.

Text values

Height
Defines the height of the identification character.

Width
Defines the width of the identification character.

Distance
Defined the distance between the endpoints of the section line and the identification character that is placed in the extension of the section line.

Inclination angle
If the sectional identification character is to be written 'normally', i.e. vertical to base line, an angle value of 90 should be entered here. If they are to incline to the right (italic), enter an angle value that is

smaller than 90. For values larger than 90, they incline to the left. The maximum permitted value is 135.
For TrueType fonts, the text for angles that are not 90° are written in italics – without taking the angle value into consideration.

Proportional
If this setting is active, the text is written proportional. This means that the value entered in the Distance text box is always the actual distance between two characters.
If this setting is not active, all characters are taken over as imaginary rectangles of the same width and the value for the distance defined the distance between the individual rectangles.
When using TrueType fonts, this settings is not taken into consideration.

Font
The currently set font is displayed. The command button next to it can be used to change the font.

Hatch
Determines settings for the hatching of sections in the inserted section view (only for sectional displayed of model views)

Automatically
If the setting is active, the sectional faces of the sectional view are automatically hatched when inserted.
The hatching is generated with the values determined using the menu command ***Settings, Hatch***.

Offset
If, for level section, two offset sectional faces border each other, both faces are allocated separate hatchings which are drawn with the same angle.
If the hatchings are to be drawn offset, enter a value between 0 and 1, e.g. 0.3. This value results in the next hatching being drawn offset by 0.3 times the spacing of the hatching lines.
With a value from 0 or 1, the hatchings are drawn without offset.

Partial View / Cut
If the setting is activated, the cuts in the material for partial views and partial cuts are represented by a line which is defined in the ***Display*** area.

Layout
In this area, the presettings for the individual components of the section identifier can be defined:

Display section run
If the setting is activated by the check box, the intersection line is displayed with the selected colour, otherwise it is not displayed.

Overhang up to arrow
Defines the length of the displayed dash-dot extension of the intersection line from its endpoint to the position of the arrow-head for the view direction.

Overhang over arrow
Defines the length by which the dash-dot intersection line is to be projected over the arrow-head for the view direction.

Length to break point
Defines the length with which breakpoints are to be identified for level sections.
This value is also used for the length of the intersection line parts which cut the outer lines of the model view.

Display arrows
If the setting is activated by the check box, the arrow-head for the view direction is displayed with the selected colour, otherwise it is not displayed.

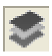
Arrow tip length
Defines the length of the side of the view direction arrow tip.

Arrow shaft length
Defines the entire length of the shaft for the view direction arrow.

Display text
If the setting is activated by the check box, the identifier characters are displayed with the selected colour, otherwise they are not displayed.

Display

Here, you can determine the display properties (colour, line type and width) and layer allocation of the section identifier.
Line 1 defines the properties of the section run.
Line 2 defines the properties of the section identifier.
Line 3 defines the properties of the section arrow.
Line 4 defines the properties of the line for partial views and partial cuts.

 By directly selecting a colour, line type or line width, the properties allocated to the current layer become inactive. The grey symbol shows that the property is object specific.
If the property is to be taken over from the current draw layer, click

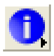
on the grey command button to the right of the selection. The command button becomes coloured and this shows that this is once again layer specific: 

The required layer for objects can be selected using the  command button.

Please note:

Since the order of the faces to be hatched in sectional display from model views cannot be checked, a value other than 0.5 should be used for a level section with more than two section planes, since neighbouring faces can be allocated with an offset value of 1.

Section-run identifiers can be disassembled into individual geometry objects using the commands ***Disassemble Dimension*** and ***Disassemble 2D Object***. The actual section-run identifier is retained by doing so, in order that any respective sectional view is not deleted.

  The display properties of sectional views can be changed using the command ***Information, Edit Object***.

Derive sectional view from model view

from ***BeckerCAD 3D***

Sectional views of solids can be completed only if you have first used the ***Insert, Model Views*** command in the 3D window to insert a model view in the 2D drawing window.

You will afterwards be able to define straight-line sections of this view

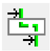
- as full sectional views,
- as level sectional views
- as an angled section in rotational solids
- as partial cuts (described in the following chapter ***Partial Cut from Model View***)

Section views can now also be derived from other section views, half section views or model views consisting break out sections. The new section view will in this case always be created from the original solid, so that the result is not double sectioned.

If the model view contains the views of several solids, you can also specify which solids are to be used for the sectional display. The display that you obtain showing the other solids will depend on the viewing direction you have chosen.

Firstly, if necessary, define the required settings for the format of the sectional display and the intersection line using ***Settings, 3D/2D Commands*** and the ***Section view*** option card*.* Using ***Settings, Hatch*** the values for hatching can be determined if the sectional view is to be hatched automatically.

Section-run identifiers can be disassembled into individual geometry objects using the commands ***Disassemble Dimension*** and ***Process2D , Disassemble 2D Object*** command. The actual section-run identifier is retained by doing so, in order that any respective sectional view is not deleted.

1.  ,  In the 2D drawing window, select the command ***Process Model Views, section view to model view...***.
2. Identify the model view for which the sectional display is to be generated. If you want to create a partial cut, read on in the chapter Partial Cut from Model View.
3. If necessary, in the dialog box, select whether a ***full cut,*** a ***step cut*** or a ***Rotation cut*** is to be defined.
4. In the ***identifying character*** text box, enter the character with which the start of the intersection line should be identified.
5. Define the start point within the model view or on one of the outer lines.
   Please note:
   After finishing the definition, the section line will automatically be elongated on the end of the model view according to the determined settings. Therefore, define the end of the section line within the model view or on the outer edge.
6. Define the course (partial lines) of the intersection line dynamically with the cursor, by entering a value in the Status dialog box and/or by defining a point.
   The direction of the lines can be moved dynamically in 15° angle increments. If an angle is to be predetermined, enter the value in the Status dialog box.
7. Repeat step 7 for all partial lines or in a rotation cut of a rotational solid of the intersection line
   As for the starting point, the endpoint of the intersection line should lie within the model view or on one of its outer lines.
   If you define the intersection line of a step cut, quit the definition by pressing the right mouse button.
   The cut for a rotational solid is defined by the start point, the next point on the rotion axes and the endpoint of the section line.

8. If necessary, modify the suggested value for the format of the intersection line in the text boxes of the Status dialog box.

9. Define the position of the view direction arrows dynamically with the cursor and confirm using the left mouse button.

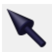
10. If the model view contains multiple solids, it can be defined if all solids are cut or only selected ones. There are the following options:

Right mouse button click
the section view is generated and can be placed. All solids are cut.

Double click right mouse button
the command is cancelled.

Selection of single solids to be cut
Identify the solids to be cut in the model view or in the 3D window, to be accessed by the ***Windows*** menu command or the ***Ctrl-key***. The solids are tagged in the selection colour. They can be deselected by repeated identify.
Click the right mouse button once to generate the section view. Switch to the 2D window again and place the section view.

Additional selection filter in the section view dialog window

 ***Select bodies to be cut***
All solids in the 3D window are selected. Switch to the 3D window and identify the solids not to be cut. They will be deselected. Click the right mouse button once to generate the section view. Switch to the 2D window again and place the section view.

 ***Deselect bodies to be cut***
If solids are identified before, with this command all solids are deselected to start a new selection list for example. Only the selected solids are shown as cut solid in the section view.

 ***Inverse selection***
The selection list will be inverted. Only the selected solids are shown as cut solid in the section view.

11. If the sectional display is not to be taken over in the standard alignment in the drawing, enter the required rotation angle in the ***Angle*** text box of the Status dialog box.

12. Position the sectional view by moving in a horizontal or vertical direction. If you want to have the option of moving the display in

any direction, you will first have to erase the entry in the ***Move*** fields in the status dialog box and then confirm this action by pressing *Enter*.
Confirm the position with the left mouse button.
The section view can optionally be placed in a different partial view than the related model view.
To do so, change to the partial drawing while the preview is on the Cursor as described here.

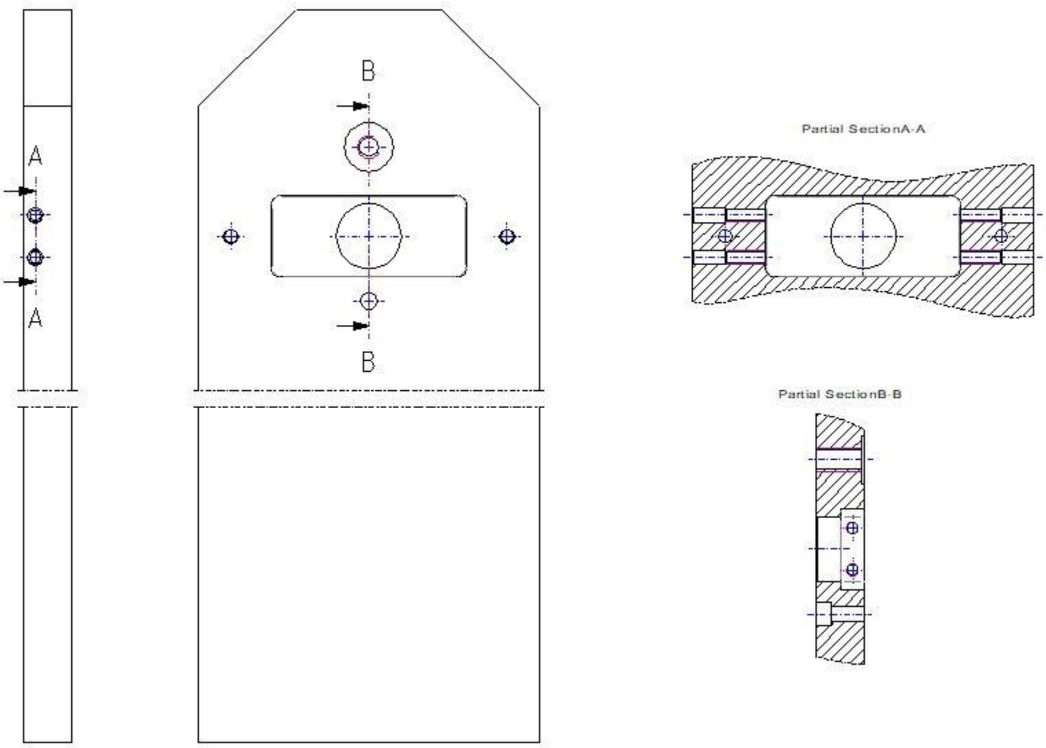
13. Place the label fort he section view.
The label is created with the text parameters defined in the current text style defined using the menu command ***Settings***, Text. The display properties are linked to the settings in the current layer for texts.
Before placing the label for the section view, it can be edited in the status dialogue window.

Please note:
The label is a text without any reference to the section line. To change the label of the section view use the commands to process texts.

Partial Cut from Model View

from ***BeckerCAD 3D***

,  A partial cut is used to display a specific section of a model view, which can be shown in section or contains the partial area of a view. After you have identified a model view, draw the cutting line in the area to be displayed over 2 points. Then place the view that is at the cursor and then select the part area to be displayed by drawing a circle, a rectangle, or a closed spline curve.



In order to be able to create partial cuts of objects, a model view must have previously been inserted into the 2D drawing window using the menu command ***Insert***, ***Model Views***.

Partial cuts can also be derived from other sectional views, half-cuts and views with outbreaks. The partial cut is in each case formed by the original bodies, so the result is not "double" cut.

If the model view contains the views of several bodies, it is also possible to define the body to which an intersection is to be created. From the other bodies, the view is displayed according to the selected direction of view.

If necessary, use the ***Settings***, ***3D/2D Commands*** in the ***Section View*** tab to define the desired presets for the text values of the intersection and the presentation of the intersection. The cut line in the partial view / partial cut delimits the area to be displayed in a partial cut and a partial view. Under ***Settings***, ***Hatch***, you define the values for hatching, if the section view is to be hatched automatically.

1.  ,  In the 2D window select the command ***Process Model Views***, ***Insert Section View***.

2. Select ***partial cut*** in the dialogue window.

3. Identify the model view to which you want to create the section view.

4. In the ***Ident. Character*** filed, enter the letter that you want to use to mark the start and end of the cut line.
5. Define the start point of the cut line.

   The cutting line merely indicates the section, but not the area to be displayed.

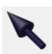
   Define the course of the cutting line dynamically with the cursor, by entering the values in the status window and / or by defining a point.
   The direction of the cut line can be moved dynamically in 15 ° angular steps. If a specific angle is needed, enter the value in the status window.
6. Define the endpoint of the cut line.
7. If necessary, change the suggested values for the format of the cutting line in the fields of the status window.
8. Define the position of the directional arrows dynamically with the cursor and confirm with the left mouse button.
9. If more than one body is displayed in the model view, you can select whether all or only selected bodies should be displayed. Choose one of the following options:

Right mouse button
the section view is generated and is placed dynamically on the cursor. All bodies are shown cut.

Double-click right mouse button
aborts the command.

Selection of solids to be cut
If you want to display only selected bodies, you must identify them in the model view or after you have switched to the 3D window via the ***Ctrl + Tab*** key combination, the bodies. The bodies marked in the selection color are shown in section view. Press the right mouse button once to create the sectional view. Then use the shortcut ***Ctrl + Tab*** to change back to the 2D window in which the model view is located and place the section view.

Additional options in the dialogue window Section

 ***Select Bodies to be Cut***
After selecting this option, all objects in the 3D window are selected. Switch to the 3D window and identify the bodies that are not to be cut. These are deselected.

Press the right mouse button once to create the sectional view. Then, use the Window menu or the ***Ctrl + Tab*** key combination to return to the 2D drawing in which the model view is located to place the section view.

Deselect Bodies to be Cut

If you have previously selected bodies, you can deselect all bodies by pressing this button, for example, to define a new selection list. Only the selected bodies are shown cut.

Inverse Selection

After pressing this button, the previous selection is reversed. Once all bodies have been selected, then all bodies are deselected after pressing this button. Only the selected bodies are shown cut.

10. If you do not want to include the section view in the drawing in the standard orientation, enter the desired angle of rotation in the ***Angle*** field of the status window.

11. Place the section view by moving it horizontally or vertically.
If you want to move the display in any direction, first delete the entry in the ***Move*** field of the status window and confirm with Enter. Confirm the position using the left mouse button.

If the section view is to be placed on another partial drawing, switch to the desired partial drawing as described in Enabling partial drawing.

12. Place the section label.
This is created with the text parameters of the current style for labeling - menu ***Settings***, ***Text***. The appearance properties depend on the current slide settings for labels.

Please note:
The automatic section label is generated as text without reference to the cut line. If you subsequently change the letter of the marking on the cutting line, use the commands of the text processing to adapt the section label to the cutting representation.

13. Use the circle, rectangle or closed spline curve to define the range to be visible.
Before you define the first point of the range shape, use the ***X*** key to toggle between the range shapes that appear in the prompt line.

Please note:
Section annotations can be disassembled into single geometry

objects using the commands ***Disassemble Dimension*** and ***Process 2D***, ***Disassemble 2D Objects***. The section label is retained in this case so that a respective sectional view is not deleted.

Process sectional view

1. , , Start the command ***Information, Edit Object***.

If you wish to process an existing sectional display and/or the respective section identifier, proceed as follows:

1. Activate the command ***Process Model Views, process Section View/Run***.

2. Identifiy a sectional view or the section identifiier on one of its lines.

3. Change the parameters in the dialog box ***Format.*** The option card Display is used to define display properties. Deactivate the option ***Display according to global defaults*** when defining object related settings.

Please note:

, Section identifier can be disassembled using the command ***Process2D, Disassemble 2D Objects***.
The actual section-run identifier is retained by doing so, in order that any respective sectional view is not deleted.

Shift / Extend Intersection Line in Model Views

from ***BeckerCAD 3D***

; The command ***Process Model Views***, ***Shift/Extend Intersection Lines*** is started by clicking the displayed icons.

Using this command the breakpoints, the intersection segments or the label of intersecting lines can be processed or extended.

This command has been enhanced by the option to move intersection segments parallel and by the option to toggle the display direction.

Move Label

1. Identify the label of the intersection line.

2. Place the label at the desired position.

Move intersection segment parallel

1. Identify the segment within the geometry to be cut (not near the display arrows).

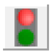
2. Click in the middle of an intersection segment, which should be moved in parallel.
3. Define the distance of the parallel movement using the status dialogue window or with the snap options in the point definition menu.(middle mouse button)

Shift Breakpoint

1. Identify the segment within the geometry to be cut (not near the display arrows).
2. Identify the intersection segment near the break point or in full sections on the side where the point should be moved.
3. The breakpoint or endpoint can be shifted to the final position using the 2D Snap Menu.
4. Repeat the action for other breakpoints, switch to the command to divide segments of the intersection line with the hotkey ***X*** or stop the action.

Please note:

Be aware that this command is not valid for rotational cuts with stepped sectional lines. This command only supports full cuts and step cuts with parallel view direction arrows.

The section view is automatically updated when the option ***update model views automatically*** is active.
 Otherwise the section views can be updated using the command ***Update Model Views***.

Extend Intersection Line

1. Identify an Intersection Line in the drawing.
2. Switch to the mode to divide the intersection line with the ***X*** hotkey.
3. Identify the intersection line at the position to be divided into 2 segments.
4. Repeat the action for other breakpoints, switch to the command to shift breakpoints with the hotkey ***X*** or stop the action.

Switch View Direction

1. Identify the view direction arrow of an intersection line.
2. Toggle the view direction an confirm by mouse click.

or

1. Identify a section run in the drawing.
2. Click the **Y** key. The view direction is switched.
3. End the command with ESC or 2 x right mouse button.

Section View in Model Views On /Off

from ***BeckerCAD 3D***

; The command ***Process Model Views***, ***Section View On/Off*** is started clicking the displayed icons.

Using this function it can be defined in section views if solids should be viewed as cut or not cut.

1. Start the command using the displayed icons.
2. Identify a section view.
3. Identify the solids, which should be changed from cut to not cut or the other way round.

Please note:

The section view is automatically updated when the option ***update model views automatically*** is active.

Otherwise the section views can be updated using the command ***Update Model Views***.

The hatchings in the solids, defined as to be cutted, are generated according to the properties defined in ***Settings, Hatch*** and are assigned to the layer for hatches or with the object relations.

; Subsequently the display properties can be edited by using the shown commands ***Face Definition/Process Hatch*** and by identifying the hatch.

Display of Model Views and Section Views

from ***BeckerCAD 3D***

By default model views and section views are displayed as defined using the menu command ***Settings***, ***3D/2D Commands***, ***Model View***.

In addition, now single views can be edited to get their individually display:

1. , Start the command ***Process Model Views, Edit Model View***.

2. Identify a model view or section view or several model or section views by dragging a rectangle or with the pressed Ctrl key. Unlock the Ctrl key before selecting the last model or section view.

3. **Model view**
   Change layers or define object related display properties.
   Deactivate the option ***Display according to global defaults.***

   Please note:
   When disassembled a Model View the 2D lines are linked to layer selected here.

   Sectional view
   Changes to the section identifier can be made in the option card ***Format***. In the option card ***Display*** settings can be changed for the display of sectional views.

4. Click the command button ***Accept*** to update the drawing.
   Click the ***OK*** command button to update the drawing and leave the dialog box.

Please note:
The quality of the display of curved silhouettes can be defined using the sliding control.
Curved silhouettes are approximated using polygons. The polygon segments are smaller or longer and the approximation is better or worse.
This definition can be made in the global defaults using the menu command ***Settings***, ***3D/2D Commands*** as well as individually for every model view as described above.

Altered settings are object related and are saved with the model. They can be reactivated even when in between a layer relation was defined or the global settings were activated.

Create Detailed Views from Model Views

from ***BeckerCAD 3D***

,  Detailed views in different scales can be created from model and section views as well as from 2D objects using the command ***Process Model Views, Detailed View***.

Detailed views from model and section views are associative to the 3D model; changes to the 3D model will effect the detailed view automatically or when selecting the command ***Update Model Views***.

The scale of the detailed view is defined by a partial drawing. If there is not yet a partial drawing with this scale defined this can be done within the command. The area to be detailed can be marked either with a rectangle or with a circle. The original and the detailed view can be labelled with/out a reference line.

1. Select the partial drawing which contains the model or section view or the 2D geometry to be detailed. Check if the relevant layers and groups are active.
2. ,  Select the command ***Process Model Views, Detailed View***.
3. Define a ***label*** for the detailed view.
4. Select from the list of existing ***Partial Drawings*** the scale for the detailed view.
   If a partial drawing not yet exists define a ***New Partial Drawing*** by giving a ***Scale*** and a ***Name*** and confirm with ***Create***. The default for the name is ***Detail*** if no other name is given.
5. Select the ***Frame*** a ***Reference Symbol*** and the ***Scale Label***. The size of the scale label can be influenced by the size factor.
6. In the tab Drawing Options the display as well as the layer assignment of the label at the original and the detailed view can be defined.
7. Select the ***OK*** button to create the detailed view.
8. Depending on the frame choosen, define the area by dragging a rectangle or a circle. It can be switched between rectangle and circle before defining the first point using the **X** key.
9. Also with the **X** key you can switch between the left and the right text reference point. Place the label at the original.
10. Place the detailed view. The partial drawing has been switched automatically.
11. Again with the **X** key you can switch between the left and the right text reference point. Place the label at the detailed view.

Please note:

The settings defined in the dialog window are saved with the model. Generally these settings should also be saved in your template (TPL).

,  The frame of a detailed view is not a contour defining line. To hatch a detailed view the command ***Create contour lines from frame*** can be used. After creating the detailed view select the

command ***Information***, ***Edit Object*** and identify the frame. By clicking the icon ***Create contour lines from frame*** a circle or rectangle, depending on the type of the frame, will be generated on top of the frame.

Detailed views from model and section views without a reference to a solid in the model will be modified to real 2D curves and are no more model or section views. The reference is lost for example if a copy of the original drawing is added to the current model using the command ***Add drawing***.

Edit Detailed Views from Model Views

from ***BeckerCAD 3D***

, The command ***Process Model Views***, ***Edit Detailed View*** is started using this icon.

With this command the label of a detailed view and the frame can be edited.

Move Label

1. Select the partial drawing of the original or the detailed view.

2. Identify the label to be moved.
By pressing the **X** key the text reference point of the label can be switched from the left to the right, while it is on the cursor.

Edit Frame at Original

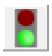
Changes to the form and the size of the frame only makes sense in detailed views of model or section views, because only in this cases the detailed view will change. Detailed views from 2D objects must be changed manually or newly created.

1. Select the partial drawing of the original.

2. Identify the frame of the original.

3. Depending on the type of the frame define a new rectangular or circular frame.
Before defining the first point, with the x key you can switch between a circular or rectangular frame.

The scale of the detailed view can by changed by changing the scale of the partial draw. Select the menu ***Settings, Drawing*** and edit the scale.

 After changing the frame the detailed view will be updated automatically or by pressing the icon ***Update Model View***.

, or The commands ***Move*** or ***Move Dynamically*** in the icon bar ***Transform*** can be used to move the frame.

, To edit the Detailed Views Parameter, e.g. the Label, the Reference Symbol or the Display Options use the command ***Information, Edit Object*** (key command ?) and identify the frame of the original or the detailed view.

Create Partial Views

from ***BeckerCAD 3D***

, Model and section views are derived from one or more solids. A view is displayed as a whole. To display only a part of this complete views the command ***Process Model Views***, ***Partial View...*** can be used.

The cut in the material can be represented by any line. ***The Partial View / Cut*** settings can be found in the ***Settings***, ***3D/2D Commands*** menu on the ***Section View*** tab.

To create a partial view from a model or section view do the following:

1. Select the partial drawing containing the model or section view. Be sure to activate the relevant layers and groups.
2. Refresh the model or section view using the icon ***Update Model View***.
3. Start the command ***Process Model View, Partial View...***
4. Identify the model or section view.
5. Define the area to remain displayed by a rectangle, a circle or a closed spline curve.
   Before defining the first point, with the **X** key you can switch between the different options displayed in the status line.

After defining the form, the view will be reduced to the selected area automatically or by clicking the icon ***Update Model View***.

Delete Partial View

To get the complete view back select the menu command ***Process Model Views***, ***Partial View*** and cancel the command (without defining a frame) by pressing the right mouse button or the ***Esc*** key.

Process cut line display

, Start the command ***Information, Edit Object*** and identify a line within the partial view to process the display properties of the cut lines.

Process Partial View / Partial Cut

Depending on the type of processing, the following commands are available for processing a partial cut derived from a model or section view.

, You can use the ***Process Model Views***, ***Partial View*** command to change the type and size of the shape for the partial view that you want to display.

from ***BeckerCAD 3D***

Processing the type and size of the partial view

1. Activate the partial drawing on which the model or section view is located. If necessary, check whether layers and groups are activated.
2. If necessary, refresh the model or section view with the ***Update model views*** command.
3. Start the command ***Process Model Views***, ***Partial View***.
4. Identify a line within the partial view.
5. Use a circle, a rectangle, or a closed spline curve to define the new visible area.
   Before you define the first point of the new area shape, use the ***X*** key to toggle between the three possible area shapes displayed in the prompt line.

   Pressing the right mouse button instead of selecting a new range shape, the partial view is removed and the complete section view is displayed again.

Edit display

, If you want to edit the display of the section line or the section, use the command ***Information***, ***Edit object***.

1. Activate the partial drawing on which the model or section view is located. If necessary, check whether layers and groups are activated.
2. If necessary, refresh the model or section view with the ***Update model views*** command.
3. Start the command ***Information, Edit Object***.
4. Identify the section line or a line within the partial view (except hatching).
5. Make the desired changes and confirm with ***OK***.

Insert solid in view

from ***BeckerCAD 3D***

, After clicking on the displayed icons, the ***Process Model Views***, ***Insert Solid In View*** command is started.

This command can be used to later insert solids into model and section views that have already been derived.

1. Identify the model and section views where solids should be inserted using a selection window or with the pressed ***Ctrl-key***.
2. Change to the 3D area using the key combination **Ctrl+TAB**.
3. Identify the solid(s) to be inserted into the model views.
4. Activate the ***Update Model Views*** button in the upper area of the screen.

Please note:

Solids are inserted to section views in any case when they are inserted in the model view this sectin view is derived from. Update the model views if necessary.

Disassemble Model View and Section View

from ***BeckerCAD 3D***

, After clicking on the displayed icons, the ***Process Model View***, ***Disassemble Model View*** command is started.

If individual lines are to be processed from the model views or the derived section views which have been inserted into the 2D drawing window, these must be converted, therefore into "real" 2D line objects.

Follow this procedure:

1. Select the menu command ***Settings, 3D/2D Commands*** and change to the ***Model view*** option card. Using the ***erase model view when disassembling*** and ***disassemble model views to groups*** check box to determine the required setting.
2. Accept all model views and/or section views to be converted into the selection list.
3. In the CAD Menu, activate the ***Process Model Views, Disassemble Model View*** or the ***Disassemble 2D object*** command.
4. If model views or section views already exist in the selection list, 2D line objects are generated from them.

   Otherwise accept the model views to be disassembled in the action list.

   Depending on the selected setting, the original model views remain or they are erased.

   The 2D objects generated when disassembling a model views or section views are inserted in the model structure in the following way:

   Group All objects are saved in the **current group**. This can be selected from the existing groups using group selection in the main toolbar.
   The groups to which the objects belong can later be modified using the ***Model Explorer***.

   Colour, line type, line width The 2D objects generated from visible edges and silhouettes are linked to the **current drawing layer**. This can be selected from the defined layers using layer selection in the main toolbar.

   If the invisible edges and silhouettes are to be displayed, the 2D objects generated from them are linked with the layer, which is determined as the layer for *hidden edges (2D)* which is defined as ***Hidden Edges (2D) Layer*** using the ***Layer Explorer*** inside the ***Model Explorer***.

Please note: Model views form the requirement for the generation of sectional displays. If they are disassembled and erased at the same time, they are no longer available for the menu command ***Insert, Section views***.

, Model views can also be disassembled using the command ***Process 2D, Disassemble 2D object***.

4.19 Use attribute

Individual objects or groups can be allocated additional information with the help of attributes.

This information can be **text**, **numeric values** or **files**.

Text and numeric values can, for example, be used to output them in lists. They can be constant, be entered freely or taken over using a SQL prompt from databases or tables. It is also possible to define them using formulas. The formulas are offered in the programming language *Python* and can be edited or extended in any way.

Files can contain additional information to the object, be it in the form of text, graphics, audio or video file.

Terms and procedure

An attribute is an **information carrier** - comparable to a field on a card index for accessories in a store:
It provides a place for entering information.

A card index contains different fields for the necessary data belonging to an accessory:
e.g. for the part name, storage position, quantity and dimensions.

Each field is allocated for a specific **type** of information. For one field, this could consist of text - like the part name, for other fields, it could consist of digits - like the dimensions.
Other fields sometimes contain a standard entry or the note to an illustration of the part in a catalogue.

So that the entries are made in the "correct" field, each field is allocated a specific **name**.

The entry made in a field is the **value** of the information.

Procedure when using attributes

1. **Definition of the information carrier**

   The **information carrier** is defined for a new attribute with the respective determinations:

   Type of value to be entered (text, numeric, file)

Presettings for the **value** (fixed or variable, if necessary database as source of the value)

Presettings for the **relation** in the drawing

Clear **name** for identification when selecting the attribute, e.g. order number, storage location, etc.

2. Allocating an attribute value

Link a object or group with the attribute value.

If no fixed value is predetermined in the information carrier, it can be entered now or determined from model data or determined from a database or table using the ODBC interface. Formulas calculate their values automatically, after the list of attributes are updated.

Depending on the presettings for the relation in the drawing, the attribute value in inserted in the model as text (attribute text) visible or invisible, active (identifiable) or not active.

3. Displaying or editing an attribute value

The allocated attribute values can be displayed or edited.
In the case of links with text, graphic, video or audio files, these can be displayed (played) or even processed.

4. Create attribute list

The allocated attribute values can be collected in a list whose format can be predefined.

Please note:

If an attribute is to accept database information by an SQL prompt using an ODBC interface, this interface must previously be defined using the Windows *system settings*.

Defined information carriers of attributes are saved in a template. This means that when using this template, they are available in each new model for allocation.

Define information carrier of an attribute

Before 2D objects or groups can be allocated attribute values, the information carrier of the attribute must be defined.

This should be carried out in a template file (*.TPL), since the defined information carrier will be available in each new model which is based on this template.

1. Select the menu command ***Define, Attributes***.

2. According to the attribute type, select the respective option card – ***Text attributes***, ***File attributes*** or ***Numeric attributes***.
3. In the ***Attribute value*** text box, enter a clear name for the attribute. i.e. the name must not be in the attributes list already.

   Please note:

   When you are working with the standard sheets supplied as part of the module software, you should note that they contain attributes already.
   In order to avoid naming conflicts in attributes, before defining your own attributes you should insert a standard sheet (menu command ***Settings, Drawing***).
4. Use the right-hand section of the option card that is selected to specify the nature of the attribute text that are to be inserted in a drawing when allocated to objects or to groups:

   Display in model
   This will determine whether or not the attribute text is visible.

   Selectable attribute text
   This will determine whether or not attribute text can be identified later with the cursor.
   When you are allocating the attribute text, they ought not to be visible. The setting can be deactivated at any time later by altering the entry.

   Position relative to object
   This setting is only valid for attributes which are allocated to 2D individual objects.
   If the check box is active, the position of the attribute text refers to the object: If the position of the object is modified, the position of the attribute text is modified in the same way.
   If the setting is not active, the position of the attribute text is independent of the object.

   Position under last
   The first attribute text allocated to a 2D object or group can be freely positioned.
   When this setting is active, any attribute text allocated afterwards will be positioned automatically with the line spacing defined in the current text style under the last allocated attribute text.
   If the check box is not active, the attribute text can be freely positioned.

Copy with object
If this check box is active, the attribute text allocated to an object is also allocated to the copy after a copy has been carried out.

Copy with object division
If this check box is active, the attribute text which is allocated to a 2D line object, is also allocated the new parts of the object when the object is being divided.

Delete with last text
In the case, the attribute should be deleted with the last text linked to an object or group, activate this option.

5. Activate one of the options that are presented in the right-hand section of the option card that has been selected. This will determine how the *Attribute Value* is to be defined.

   The name of these options can be found in:

   Options for text attributes

   Options for file attributes

   Options for numeric attributes

6.  Confirm the settings by clicking the ***Create Attribute*** command button.

   If the attribute value is determined using a SQL prompt, the ***Define SQL prompt*** dialog box is displayed in which presettings can be determined for the definition.
   The respective name can be found in the Define SQL prompt chapter.

   If the attribute value is not determined using an SQL prompt, the name of the attribute that is generated will be entered in the list in the lower section of the dialog box.
   If an attribute is generated with a name which already exists in the attribute list, the name for the new information carrier is increased by ".1". This is because each of the names must be unique.

7. In the list mark those attributes, which should be assigned to the same update procedure. Several attributes can be collected with the pressed ***CTRL*** or ***Shift key*** according to Windows standards.
   Click the button ***Update attributes...***.
   Activate or deactivate the options, related to which operation the previously marked attributes should be updated:
   ... on user request
   ... after model load
   ... before model save

... before print

When several attributes with different settings were marked, the option will be greyed. In this case the option must be activated or deactivated again, to have the same options for all selected attributes.

Confirm the settings by clicking the ***OK*** button.

Please note:

With the option ***...on user request*** the attribute values in the model are updated with the command ***Attribute***, ***Update*** in the menu ***Extras***.

Options for text attributes

Determine using SQL prompt

The attribute value is to be determined from a database via the ODBC interface. Depending on how the query has been defined, the value may be fixed or variable.

Using keyboard entry

The attribute value is to be determined by typing in a text: there may be any value.

In the text box for the ***Suggested Value*** you can type a text that can be edited before being allocated.

Advice on using symbols:

If a text attribute of this kind has been allocated to the main group that is saved in a symbol, the text may be offered automatically for editing after the symbol has been positioned.

Formula

The attributes in the list are created by formulas, defined in the prgramming language PYTHON. The list can be extended as needed. Further informations can be found here.

After selecting an attribute from the list, the according formula is entered in the field ***suggested value*** and can be edited there.

Example to edit a formula.

The predefined formula length x.xx contains the formula

```
'%.2f'%dLength(pOpAtt)
```

which outputs the length of a 2D curve with 2 decimal places.

To reduce the number of decimal places to one, please exchange the 2 by a 1.

If the unit ‚mm' should be added, extend the formula by mm:

```
'%.1f mm'%dLength(pOpAtt)
```

List of predefined attribute formulas

Date / Time

Date dd.mm.yy

```
DateTime('%d.%m.%y')
```

Date dd.mm.yyyy

```
DateTime('%d.%m.%Y')
```

Time hh:mm

```
DateTime('%H:%M')
```

Time hh:mm:ss

```
DateTime('%H:%M:%S')
```

Objekt Names

Symbol Name

```
SymbolName(opAtt,0)
```

Symbol Name including Path

```
SymbolName(opAtt,1)
```

Group Name

```
GroupName(opAtt,0)
```

Sketchboard Name

```
SketchBoardName(opAtt,0)
```

Draft Name

```
DraftName(opAtt,0)
```

Model Name

```
ModelName(opAtt,0)
```

Model Name including Path

```
ModelName(opAtt,1)
```

Values taken from other attributes

useful to combine values or display values differently formatted

AttributeValue(@Name)

```
AttributeValue(opAtt,'@Name','')
```

AttributeValue(@Quantity)

```
AttributeValue(opAtt,'@Quantity','0')
```

Area / Length

Length of 2D Element x.xx

```
'%.2f' % Length(opAtt)
```

Length of 2D Element x.xxx

```
'%.3f' % Length(opAtt)
```

Area of 2D Face x.xx

```
'%.2f' % Area(opAtt)
```

Area of 2D Face x.xxx

```
'%.3f' % Area(opAtt)
```

Area (Group)

```
AttributeValue(opAtt,'Area x.xx','?.??',2)
```

This suggested value returns the value of the first given attribute within a group as attribute value. The attribute value "?.??" is returned, if the highest group doesn´t contain the given attribute. By using the parameter "2" the given attribute can also be assigned to a sub group.
With this you can get the area within a 2D part, if the area attribute was assigned to a face (opAtt), to be used as an additional information in the BOM.

Environment Variables

User Name

```
unicode(os.getenv('USERNAME'),'cp1252')
```

Computer Name

```
unicode(os.getenv('COMPUTERNAME'),'cp1252')
```

Sketchboard / Draft

Sketchboard Scale

```
SketchboardScale(opAtt)
```

Sketchboard Scale 'XYZ'

```
SketchboardScale(GetSketchboardWithName(GetParentDraft(CastTo(opAtt, 'ICADdyOpUserAttInst').Owner),'XYZ'))
```

Symbols

Symbol Count (Sketchboard)

```
SymbolCount(opAtt,1,0)
```

Symbol Count (Group)

```
SymbolCount(opAtt,0,0)
```

Symbol List (Sketchboard)

```
SymbolCount(opAtt,1,1,'.*','%-12s : %3i',0)
```

Symbol List (Group)

```
SymbolCount(opAtt,0,1,'.*','%-12s : %3i',0)
```

Name (3D-Group)

```
GetChild(GetApplic().TargetManager().TargetOp3DSolids,'XOpGr
oup','ICADdyOp3DGroup').Name
```

The suggested value "Name (3D Group)" returns as attribute value the name of the first group from the 3D Solids area.

Physical properties

Material

```
Material(opAtt)
```

Density x.xx

```
'%.2f g/cm^3' % Density(opAtt)
```

SurfaceArea x.xx

```
'%.2f cm^2' % SurfaceArea(opAtt,0.001)
```

Volume x.xx

```
'%.2f cm^3' % Volume(opAtt,0.001)
```

Weight x.xx

```
'%.2f kg' % Weight(opAtt,0.001)
```

Options for file attributes

The attribute value is the link to a file (text, drawing, audio, etc.) that can be generated, processed or even only displayed with a suitable program.

This will not be possible, unless the program chosen can be called up by means of the *File Name* as parameter so that it will set up the file or open it.

Generate with

This will specify the name of the program (the entry to include drive and path) for creating the file specified in the ***File Name*** box.

 This is the icon to be used to specify the program via the file selection. The name will be offered in the next two boxes as well.

Process with

This will specify the name of the program (the entry to include drive and path) for processing the file specified in the ***File Name*** box.

 This is the icon to be used to specify the program via the file selection.

Display with

This will specify the name of the program (the entry to include drive and path) for displaying or running the file specified in the ***File Name*** box.

 This is the icon to be used to specify the program via the file selection.

File Name

This box is used to enter the name of a file as the one to be offered. The name can be edited before an attribute is allocated.

Please note:

Only if a program name has been entered for ***Process with*** and ***Display with***, can the selected file be displayed or processed when displaying or modifying the attribute values.

Options for numeric attributes

The attribute value for numeric attributes is a number. This can be determined or suggested using one of the following options:

Determine using SQL prompt

The attribute value is to be determined from a database via the ODBC interface.

Using keyboard entry

The attribute value is to be determined by typing in a number.
In the text box for the ***Suggested Value*** you can type a text that can be edited before being allocated.

Formula

The attributes in the list are created by formulas, defined in the prgramming language PYTHON. The list can be extended as needed. Further informations can be found here.
After selecting an attribute from the list, the formula is entered in the field ***suggested value*** and can be changed there.

Decimal places

This will specify the number of decimal places to be used for a numerical value.

Create User Defined Attribute Formulas

New text attributes and numeric attributes can be defined by formulas in the programming language PYTHON. In the following you can find some examples for formulas and how to define own attributes.

The formulas for own attributes can be defined in two ways:

In a seperate text file:

In the TLW directory open the text files ***AttStrgExpressions.txt*** for text attributes, ***AttRealExpressions.txt*** for numeric attributes.
Enter the formulas into this text files according to the described scheme. The formulas are reread every time the dialogue window ***Define Attributes*** is opened. For newly defgined attributes the program must not be restarted. Select the desired attribute from the list of ***formulas***. The according formula is taken over in the field ***suggested value***.

Define attributes in the dialogue window:

(Menu ***Define***) Enter the formula for the desired attribute directly in the field ***suggested value***.

 To create a new attribute based on a predefined formula, activate the option ***formula*** and click the icon ***Create Attribute***. The settings for the behaviour of the attribute texts can be defined in the right part of the dialogue window.
If the attribute is created in a template, this attribute is available in all new models based on this template.

Text Files for Attribute Formulas

In the above mentioned text files the entries for the dialogue window ***Define Attributes*** are described in two following lines.
In the first line enter the description of the formula for the selection list; in the second line define the formula for the desired attribute.
Other lines, containing free texts, have to start with the # sign, empty lines are being ignored.

Example:

```
# Formula for the attribute ’Date’ with a
# two digit year
Date dd.mm.yy
DateTime('%m.%d.%y')
```

Python Formulas

The Python formel interpreter evaluates the **given formula** in the form:

```
result = <given formula>
```

The resulting value ***result*** is assigned as value to the attribute. If an eeror occurs during the evaluation, *BeckerCAD* will return an error message and the attribute value is set to '?'.

In the formula interpreter some Python moduls, variables and commands are predefined, which are described as follows.

Prdefined Python Moduls

The following modules are offered in the Python formelula interpreter:

os	for the operating system,
re	for regular terms
math	for mathematical functions
time	Modul for date and time commands
codecs	Modul for the conversion of fonts

Example

The command `getenv` from the Python-Modul `os` determines the value of a windows environment variable.
So for example the user name can be determined with the formula `os.getenv("USERNAME")`.

In the following some predefined functions are described.

Predefined Variables:

opAtt

opAtt is the Variable for e.g. the *group name* or *the length of an object*. It references the attribute for which the value should be calculated and must be given as an argument for the function *GetGroupName* or *Length*.
Example for an attribute value Length with two decimal places:
`'%.2f' % Length(opAtt)`

Predefined Functions:

DateTime(*format*)

determines the actual date and time as text. *Format* is a text, which can contain special characters allowed for the system function *strftime*.

The most important special characters are:

%a	Abbreviated weekday name
%A	Full weekday name
%b	Abbreviated month name
%B	Full month name
%d	Day of month as decimal number (01 – 31)

%H	Hour in 24-hour format (00 – 23)
%I	Hour in 12-hour format (01 – 12)
%m	Month as decimal number (01 – 12)
%M	Minute as decimal number (00 – 59)
%R	Time as %H:%M
%S	Second as decimal number (00 – 59)
%T	Time as %H:%M:%S
%V	The Iso 8601 week number as decimal number (01 – 53)
%W	Week of year as decimal number, with Monday as first day of week (00 – 53)
%y	Year without century, as decimal number (00 – 99)
%Y	Year with century, as decimal number
%%	Percent sign

GroupName(opAtt)

returns the group name, where the attribute is attached to, as a string. If the attribute is attached to an element, *GetGroupName* returns the name of the group where the element is allocated

SketchboardName(opAtt)

returns the name of the sketchboard, where the element is allocated, as a string

DraftName(opAtt)

returns the name of the draft, where the element is allocated, as a string

ModelName(opAtt,*withPath***)**

returns the model name as a string. The flag *withPath* defines if the complete path of the file should be included (*withPath* =1) or not (*withPath* =0)

SymbolName(opAtt,*withPath***)**

returns the symbol name as a string. The flag *withPath* defines if the complete path of the symbol should be included (*withPath* =1) or not (*withPath* =0)

Length(opAtt)

returns the length of a 2d element (line, circle, ellipse or face) as a floating point value

Area(opAtt)

returns the area of a 2d face as a floating point value

GetApplic()

returns the *BeckerCAD* Application Object, to get e.g. the current group, drawing or the 3D space:

```
GetApplic().TargetManager().TargetOp2DGroup
GetApplic().TargetManager().TargetOp2DSketchboard
GetApplic().TargetManager().TargetOp3DSolids
```

GetChild(opObj , objClass, objInterface)

determines the first object of the specified data type *objClass* starting from the given component*opObj***,** and returns the interface *objInterface* from this object.

Examples for data types:

`'XOpGroup'`	`Group`
`'XOpSkB2Draft'`	`partial drawing`
`'XOpDraft'`	`drawing`
`'XOpFace'`	`2D face`
`'XOpStraSeg2'`	`line`

Examples for interfaces:

```
'ICADdyOp2DGroup'
'ICADdyOp3DGroup'
'ICADdyOp2DStraSeg'
'ICADdyOp2DFace'
```

AttributeValue(opObj, name, default, where=0)

AttributeValue(opAtt, name, default, where=0)

determines the value of the attribute with the name *name* or returns the value *default* , if the attribute cannot be found. The command **AttributeValue** accepts as a first parameter, in addition to the predefined value **opAtt,** also any *BeckerCAD* Object, e.g. a line, 2D face, group, drawing, partial drawing etc. The parameter *where* defines the search depth inside the group structure:

0 = The attribute, to deliver the attribute value, is only searched for at *opObj*.

1 = The attribute, to deliver the attribute value, is only searched for at *opObj* and all directly contained objects

2 = The attribute, to deliver the attribute value, is searched for at *opObj* and all contained groups and objects.

Examples

In the following you can find some examples of Python Formulas. In every example a formula is displayed with the font `Courier` .

Please note, that the complete formula is written in one line, wordwraps must be ignored.

In Python the **+** is used to connect texts. The name of the model and the drawing, divided by a blank can be assembled as followed:

```
ModelName(opAtt,0) + " " + DraftName(opAtt,0)
```

Current Date and Time

```
DateTime('%d.%m.%Y %H:%M')
```

Part-Name + Part-Length

```
AttributeValue(opAtt,'BT-Benennung','')+"x"
+AttributeValue(opAtt,'BT-Länge','')
```

This is an example for a combination of attributes. In this example the new attribute should be the combination of the attribute values from "Part-Name" and "Part-Length": Flat Material 100x25x240.

Name (3D Group)

```
GetChild(GetApplic().TargetManager().TargetOp3
DSolids,'XOpGroup','ICADdyOp3DGroup').Name
```

The suggested value "Name (3D Group)" returns as attribute value the name of the first group from the 3D Solids area.

Area (Group)

```
AttributeValue(opAtt,'Fläche x.xx','?.??',2)
```

This suggested value returns the value of the first given attribute within a group as attribute value. The attribute value "?.??" is returned, if the highest group doesn´t contain the given attribute. By using the parameter "2" the given attribute can also be assigned to a sub group.
With this you can get the area within a 2D part, if the area attribute was assigned to a face (opAtt), to be used as an additional information in the BOM.

Attribute for the Scale Factor of certain Partial Drawings

With the help of a attribute formula, you can output the scale factor of certain partial drawings automatically in the title block. A separate attribute must be generated for each scale, that is, for each individual drawing, whose scale is to be displayed. The link between the part drawing and the attribute is made by naming the part drawing, which is thus defined once and cannot be changed later.

General approach

using the example of a standard sheet whose geometry is in a group:

1. Select the menu command ***Define, Attributes***.
2. In the ***Attribute value*** area, activate the option ***formula*** and select the predefined attribute formula from the list next to it ***Sketchboard Scale 'XYZ'***.
3. In the ***suggested value:*** line, replace the text ***XYZ*** with the name of a respective partial drawing for which the scale is to be adopted. Do not make any further changes to the formula!
4. In the field under ***Attributes:*** edit the automatically adopted designation ***Sketchboard Scale 'XYZ'*** and enter a unique name for the attribute. The specified name should not be contained in the attribute list.

   Please note:

   > If you work with the standard sheets belonging to the delivery, note that these also contain attributes.
   > To avoid naming equivalence, insert a standard sheet before you define your own attributes (menu command ***Settings***, ***Drawing***).

5. In the right part of the dialog, set the defaults for the attribute text behavior. In this example, the following options should be activated:

   Display in model
   Determines whether the attribute texts should be visible in the drawing

   Selectable attribute text
   Determines whether the attribute texts should be identifiable with the cursor.
   When assigning attribute texts, these should be visible. The default setting can be deactivated later by changing the information carrier.

   Position relative to object
   This default applies only to attributes assigned to 2D objects.
   If it is active, the position of the attribute text refers to the object. If the position of the object is changed, the position of the attribute text changes in the same way.
   If the default is not active, the position of the attribute text is independent of the object.

Delete with last text
If you want to delete an attribute with the last text of the object or the assigned group, select this option.

6. Confirm the defaults by clicking the ***Create Attribute*** button.

   The name of the generated attribute is entered in the list at the bottom of the dialog box.

   If an attribute is created with a label that already exists in the list of attributes, the label for the new attribute is extended by a sequential number starting with '.1'.

7. Specify when to update the attribute.
   Select it in the list of existing attributes and click the ***Update attributes ...*** button.

   Activate or deactivate the desired options:
   ... on user request
   ... after model load
   ... before model save
   ... before print

8. Close the dialog.

Assign the attribute value to the top group in which the geometry of the standard sheet is located:

9. Select the menu command ***Extras, Attributes, Allocate***.
10. Select the previously defined attribute in the list.
11. Check the ***Selection***. The attribute is assigned to a ***Group***.
12. Start attaching the attribute value by clicking the ***Attach*** button. Identify a line from the standard sheet that you know is in the top 'main group'.
13. Now set its position.

Define SQL prompt

When defining the data field for a component text, it can be determined that the value should be defined using a SQL prompt from a database or table.

The same is valid for the definition of an information carrier for a text or numeric attribute whose value is determined using an SQL prompt from a database.

The requirement for this is that the respective database program has been installed on your computer and the respective ODBC interface has been configured using Windows system settings.

If you are familiar defining SQL queries, you can enter the query in the field ***SQL Query*** and verify it by clicking the ***Test*** button. (Example). Otherwise the dialogue will help to define a correct SQL query by filling the fields *Table*, *Column*, *Offer selection on assigning attributes* and *Select row by value of another attribute*.

1.  Determine the connection to the ODBC interface using the command button in the ***ODBC source*** line.
Then select the required data source.

2. If a number of databases are available, in the ***Table*** list, select the one from which values are to be determined.

3. Select the ***Column***, by which the attribute should be determined.

4. Define, in which way the attribute value should be determined:

Offer selection on assigning attributes
When assigning the value for the attribute, the values from the column defined under 3. are offered.

Select row by value of another attribute
Select from the list the previously defined attribute, by which the attribute value in the line below should be determined.

Insert fixed value
If the attribute value should be a fixed value choose the option ***Select row by value of another attribute*** and complement the SQL query, by entering a fixed value instead of the variable relation, for example:
`SELECT 'Price' FROM 'Pricelist' WHERE "Product" = 'Wood'`

5. Use the ***Test*** command button to display the result of the SQL prompt in the list editor.
Close the editor.

8. Confirm the definition of the SQL prompt using ***OK*** or ***Cancel***.

Please note:

The predefined SQL query for an attribute value will be displayed in the line ***suggested value*** in the dialogue window ***Define Attributes*** and can be changed there also.

In the field ***SQL query*** every valid Select definition can be entered. Within the query you can reference to other attributes using the

form$('Attributename').
If the query delivers one result, then this value will be entered when Assigning attributes and when Updating attributes. Non valid queries will deliver the value '?'.
If the query delivers more than one result, a dialogue window is offered when assigning or updating attributes, where the results will be listed. While automatic updating, e.g. before storing or printing, this selection dialogue is not offered. The attribute values will remain unchanged.

SQL prompt example

In this example a table assignment by name is defined in an external spreadsheet. The table assignment is to be displayed graphically, with changes from the spreadsheet to be included in the drawing. A symbol table group with a number of chairs was created. To each chair an attribute NAME.1, NAME.2 etc is assigned, which is to be determined by SQL query from the spreadsheet with a table number. The table group gets the attribute *TableNumber*, which is determined during the placement of the symbol on *keyboard input*. Outside of *BeckerCAD* an ODBC link is applied to the data source *TableAssignment* with the table *Assignment*.

Content of the table *Assignment*

TableNo.	Name1	Name2	Name3	Name4
1	Müller	Schmitz	Maier	Schneider
2	Fischer	Weber	Wagner	Becker
3	Schulze	Hoffmann	Schäfer	Koch

In *BeckerCAD*, using the menu command ***Extras, Attributes***, the SQL query related to the spreadsheet for the single names is defined as follows:

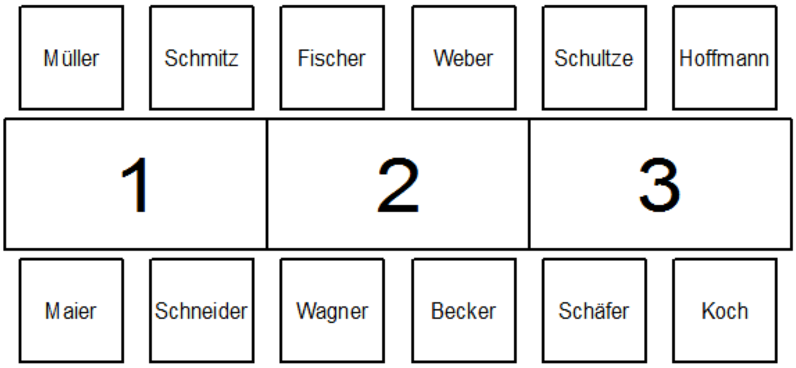
SQL Query for Attribute Name.1

Data source:	**TableAssignment**
Table: (in the data source)	**Assignment**
Row:	**Name1**
Select row by *BeckerCAD* attribute value ermitteln:	**TableNumber**
Spalte in der Tabelle	**TableNo.**

results in the SQL query:

```
SELECT "Name1" FROM "Assignment" WHERE "TableNo." =
$(TableNumber)
```

In the drawing the names are assigned to the related chairs:



Please note:

If you activate the option ***Offer selection on assigning attributes*** in the dialogue window ***SQL Query Definition***, the names can be assigned to the chairs during the placement of the symbol.

Modify information carrier of an attribute

If a mistake has been made when defining an information carrier or attribute values which have already been allocated are to be made visible/hidden or selectable/not selectable, modify the attribute information carrier:

1. Select the menu command ***Define, Attributes***.
2. According to the type of attribute, select the respective option card – ***Text Attributes***, ***File Attributes*** or ***Numeric Attributes***.
3. Select the attribute name whose information carrier is to be modified, by clicking on one within the list. This is then displayed in the ***Attribute*** text box.
   To change the properties of several , they can be collected with the pressed Strg key.
4. If the attribute value is <u>not</u> determined using a SQL prompt, determine the defaults as when defining an information carrier.
   Confirm the modifications by activating the ***Update*** command button.

   If the attribute value is determined using a SQL prompt, activate the ***Edit SQL Prompt*** command button. As when defining a SQL prompt -determine the required options and rules for the determination of a value.

Please note:

If an attribute information carrier is modified, this also affects all the attributes in the drawing with this information carrier.
If, for example, the setting do not display in model, all attribute text which are based on this information carrier will also be made invisible.

Delete attribute information carrier

If the information carrier of a defined attribute is to be deleted, proceed as follows:

1. Select the menu command ***Define, Attributes***.
2. According to the type of attribute, select the respective option card - ***Text attributes***, ***File attributes*** or ***Numeric attributes***.
3. Select the attribute name whose information carrier is to be deleted, by clicking on one within the list. This is then displayed in the ***Attribute*** text box To delete multiple attributes, collect them in the list with the pressed ***Ctrl*** Key or mark the area by dragging a rectangle.
4.  Activate the command by clicking on the ***Delete Attribute*** command button and confirm the security query with ***Yes***.

Please note:

If the information carrier of an attribute is erased, all the text allocated these attributes will be erased.

To delete only the values of attributes linked to an object or group, use the menu command ***Extras, Attributes, Allocate***. Select the ***Current attribute***, whose value should be deleted and ***Remove*** it, by identifying the respectve object.
You can also delete individual attribute values in the dialog Edit attribute.

Allocate attribute values

Attribute values can be allocated to individual 2D objects or groups. If determined during the definition of the information carrier - this is entered as visible text in the drawing.

This text is linked with the current layer for geometry. They will accept the display attributes from here. Font and text size is taken from the current text style defined in ***Settings, Text***.

1. Select the menu command ***Extras, Attribute, Allocate***.

2. According to the type of attribute, select the respective option card - ***Text attributes***, ***File attributes*** or ***Numeric attributes***.
3. Select the required attribute by clicking on one within the list.
4. In the case of text attributes and numeric attributes for which the keyboard is to be used, as well as with file attributes you can edit the entry in the ***Value*** box to specify whatever attribute value you want.

   For all other attributes, the value is determined.
5. **Allow multiple attachment to one element**
   If an attribute should be attached to an object / group multiple times, this option must be activated. Otherwise a message appears, if the attribute is already attached to this object / group.
6. Define a selection, to which elements the attribute should be attached to:

   Groups
   Attributes can only be attached to groups, if they contain objects. Activate this option, to attach the attribute to a group or *all higher structure levels* by identifying one objects inside the group.

   3D Solids
   Activate this option to attach the attibute to solids in the 3D window.

   Objects according to selection
   Using the menu command ***Settings***, ***Object Filter*** a selection can be made for those objects to attach attributes to.

   Before the identification of an object, a context menu can be loaded using the middle mouse button in which this setting can be modified.

   Please note:

   With some combinations of attribute type and determination of how the value is to be defined, a certain default must be selected. For example:

   groups
   Text attributes with the calculation of a ***formula*** for a ***group name***.
   Text attributes with the calculation of a ***formula*** for a ***symbol name***.
   Numeric Attribute with the calculation of a ***formula*** for the ***length*** or ***area***.

allocate any
all other attribute types and options

6. Start the allocation of attribute values by activating the ***Attach*** command button.
   Identify the required object or an object that belongs to the required group.

   For the numeric attribute ***Area of 2D Face**,* a 2D face must be identified, for ***Length of a 2D Element*** a 2D line object. The numeric value is taken over from the selected object.

   When allocating an attribute, whose value is determined using a variable SQL prompt, *the* ***Edit SQL Prompt*** dialog box is displayed.

   Select the required attribute value and confirm with ***OK***.

7. If the attribute value is the first which is allocated to an object or group, now determine the position. This is also valid for further attribute values, for whose information carrier the ***position under last*** option has been deactivated.

   If the ***position under last*** option is active in the information carrier for further attribute values, this text is positioned with the line spacing of the current text style predefined in the menu command ***Settings, Text*** under the previous text.

Please note:

If a group which is not to contain any objects, is to be allocated attributes for a list entry, proceed as follows:
Within this group, generate any object. Allocate the group the required attribute and then erase the object.

Attribute values can also be saved with a symbol. As long as the symbol is not disassembled, only the attribute values of the "main group" of symbols can be accessed. These will be detached from the symbol and then allocated to the symbol as object related attribute.
This means that they are available for displaying, editing and creating an attribute list.

Define and edit attribute value using variable SQL prompt

When allocating and editing attribute values, which are to be determined using a SQL prompt with the ***variable entry*** option, the ***Edit SQL Prompt*** dialog box is displayed.

In the top part of the dialog box, the names of the ODBC data source, the database and the columns are displayed from which the attribute value being searched for is determined.

In the list under ***value***, the list is offered which was entered when defining the SQL prompt in the ***cross reference using*** text box. Activate the command by clicking on the required line of the list under the ***value*** heading. The respective attribute value is determined.

If the selection is to be carried out using another column, click on the required column name in the ***cross reference using*** text box.
With this modification, only another column is displayed for selection. The value of the attribute is always determined from the column whose name is displayed in the ***value*** text box.

Display attribute values

If allocated attribute values are to be displayed (played), proceed as follows:

1. Select the menu command ***Extras, Attribute, Display***.

2. Activate the command by clicking on the object or a object which belongs to a group.

If the middle mouse button is pressed before clicking on the object, a context menu can be used to select whether the attribute value is to be displayed.

In the dialog box which then appears, the attributes allocated to the object are displayed with their ***description*** and ***value***.

3. If the attribute value is a file name, for which a program has been entered in the ***display with*** text box when defining the information carrier, the program for displaying or using the file can be started by activating the ***Display*** command button.

Please note:

An additional option for displaying the attribute values of a group, is the ***modell explorer:***
In the structure tree or in the list view start the context menu on the group and select the command ***Edit attributes (Shift+F4)***.
To display or edit the attributes of an object, mark the object in the list view. Using the key command ***Shift+F4*** or the context menu command ***Edit attributes*** the attributes of the object, for example the entries for the standard sheet, can be changed.

Edit attribute values

If allocated attribute values are to be modified, proceed as follows:

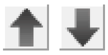
1. Select the menu command ***Extras, Attribute, Edit***.
2. Activate the command by clicking on the object or an object which belongs to a group.

   If the middle mouse button is pressed before clicking on the object, a context menu can be used to select whether the attribute value is to be displayed.

   In the dialog box which then appears, the attributes allocated to the object are displayed with their ***description*** and ***value***.
3. The content of the fields under ***value*** can be modified freely using the keyboard. Additionally, the following options are available:

   file attribute
   If the value is of a file name, for which a program has been entered in the ***process with*** text box when defining the information carrier, the program for displaying the file can be started by activating the ***Process*** command button. The entered file can be opened for processing, if necessary.

   SQL prompt
   If the attribute value is a SQL prompt whose value is variable (is not fixed nor determined using the value of another attribute), the prompt can be started using the ***SQL*** command button.
   The ***Edit SQL Prompt*** dialog box is then displayed for determining the attribute value.

    **Change sorting**
   Use the arrow keys at the bottom of the dialog to change the order of the attribute list.
   Select an attribute and then press the corresponding arrow key to move the attribute up or down in the list.

    By clicking the displayed icon, you delete the selected attribute value from the list and from the object or the assigned group.

Please note:

An additional option for editing the attribute values of a group, is the ***model explorer***:
In the structure tree or in the list view start the context menu on the group and select the command ***Edit attributes (Shift+F4)***.

If the "main group" within a symbol has been allocated attribute values, these can be edited directly after insertion, if one of the necessary options was activated before insertion using the menu command ***Settings, Symbols***.

Remove attribute values

Attribute values, which as text have been allocated the attributes ***display in model*** and ***selectable attribute text***, can - like other objects - be selected and erased individually.

If **all** values of an attribute, e.g. the reference to a DIN, are to be removed from all objects or groups of the model, it is more useful to proceed as follows:

1. Select the menu command ***Extras, Attribute, Remove***.
2. Using the option cards, select the type of attribute whose values are to be removed – ***Text attributes*****,** ***File attributes*** **or** ***Numeric attributes***.
3. Select in the list under ***current attribute*** by clicking on the attribute name.
   It can be deselected by clicking on it again.
4. If the ***Value*** text box is empty, all values allocated to the selected attribute will be removed.
   If a value is entered here, only the attribute values are removed which are identical to this value.
5. The command is carried out by activating ***Remove***.

Please note:

Dependent on the database program used, attribute values which are determined using SQL prompts can contain spaces which follow the actual entry as characters or numbers.

Attribute values, which determine the value of another attribute value using a SQL prompt, should not be removed. A SQL prompt would then be missing a necessary parameter and the attribute value is replaced by a ?.

Update attribute values

For every numeric attribute and text attribute it can be defined separately, when the value of the attribute in the model should be updated.

The associative attribute values for length and area automatically change with their information owner.

The settings, when an attribute should be updated, can be made as follows:

General Process

1. Select ***Attributes*** from the menu ***Define***.
2. In the list mark those attributes, which should be assigned to the same update procedure. Several attributes can be collected with the pressed ***CTRL*** or ***Shift key*** according to Windows standards.
3. Click the button ***Update attributes...***.
4. Activate or deactivate the options, related to which operation the previously marked attributes should be updated:
   … on any modification
   ... on user request
   ... after model load
   ... before model save
   ... before print
   When several attributes with different settings were marked, the option will be greyed. In this case the option must be activated or deactivated again, to have the same options for all selected attributes.
5. Confirm the settings by clicking the ***OK*** button.

Please note:

With the option ***...on user request*** the attribute values in the model are updated with the command ***Attribute***, ***Update*** in the menu ***Extras***.

Updating values which are also concerned with SQL prompt, can only be carried out if the connection to the entered databases is guaranteed. If this is not the case, the attribute text allocated up until now is retained.

Values which are later edited "manually", are also updated.

Select/deselect using attribute values

Attribute values which have been allocated to objects or groups, can also be used to accept the respective objects into the selection list or remove them from it.

Proceed as follows:

1. ,   Select the command ***Select, Select Using Attribute*** or ***Deselect Using Attribute***.
2. Using the option cards ***Text attributes*****,** ***File attributes*** or ***Numeric attributes***, select the type of attribute according to which the object selection is to take place.
3. In the ***current attribute*** list, determine the attribute which is to be used to select objects.
4. In the ***Value*** text box, enter a value, if only attribute values are to be searched for which correspond to this entry.
If this text box does not contain an entry, all attributes are taken into consideration which have the selected name.
5. Start the selection by activating the ***Select*** or ***Deselect*** command button.

   All objects determined using the attribute values are taken over into the selection list or removed from it - depending on the menu command selected in step 1.

Format and create attribute list

In order to create an attribute list, select the menu command ***Extras, Attribute, Create Attribute List***.

The necessary entries should be made in the respective option cards:

Using the ***Select attribute*** option card, select the attributes which are to be taken over in the list.

Using the ***Format Columns*** option card, determine the format the list columns i.e. titles, alignment, etc.

Using the ***Structure settings*** option card, determine from which group level the determination of the attributes is to take place, whether identical attribute values are to be added and in which form the list entries are to be sorted.

Select attributes for the attribute list

The menu command ***Create Attribute List*** can be used to create a list which contains the attribute values allocated in the current model.

1. Start the menu command ***Create Attribute List***.
The ***Create list*** dialog box is displayed.

2. Select the ***Select attribute*** option card in order to determine which attribute values are to be taken over in the list and which order they should have:

   By clicking, tag one of the attribute names from the list of ***Model attributes*** and transfer those which are to be taken over in the attribute list into the ***List entries***:

   Using the ***Insert >>*** command button, the tagged name is inserted in the ***List entries*** list. If another attribute name has been tagged by clicking, the additional name is inserted in front of the previous one in the list.

   If the ***Append >>*** command button is selected, the tagged name is added to the end of the ***List entries*** list.

   The ***<< Remove*** command button can be used to remove a tagged name from the ***List entries*** list or - if none is tagged - the first name.

   In addition to the names of the component text allocated in the model, the ***Model attribute*** list also contains the standard attributes position number *@No* and *@Number*.
   @No can be used to ensure that the BOM contains an additional column for quantity. If, on the ***Structure settings*** option card, the ***cumulate lines*** check box is active, the number of identical combinations of component text is entered in this column.

3. If the attribute list is created with the current settings and are to be displayed in the list editor, click on the ***Display List*** command button.

   Change to the ***Format columns*** option card if the format of the BOM is to be modified (titles, column separator, text alignment).

   Change to the ***Structure settings*** option card if additional defaults for the structure of the BOM are to be determined (cumulate identical entries, sort sequence, etc.).

Format attribute list columns

The menu command ***Create Attribute List*** can be used to generate a list which contains the attribute values allocated to the current model.

1. Start the menu command ***Create Attribute List***.
   The ***Create list*** dialog box is opened.

2. In the ***Select attribute*** option card, determine which attribute values are to be taken over in the list and which order they should have.

3. Change to the ***Format columns*** option card in order to determine the output format of the list entries:

   In the ***List entries*** list or ***Titles*** list, tag the entry or title which is to be modified.
   Enter the required text in the text box under ***Titles***. The selected ***List entries*** are displayed unmodified.

4. In the ***column separator*** text box, determine the character which is to separate the entries in the attribute list, if for example, these are to be imported from a database program.

   Select ***Justified***, if the columns are to have a fixed width.
   In such a case you must select one of the options for arranging the text in the columns.

   In this case, tag the respective names or titles in the lists at the top of the dialog box, if the ***Alignment*** of the respective text in the columns is to be determined.
   The width of the column aligns itself to the longest component text entered. This is displayed in the ***character*** text box, if a name or title is tagged in one of the top lists. The total width of the list is displayed in the ***line length*** text box.
   The ***Update*** command button can be used to determine and display the current values.

3. If the attribute list is to be generated with the current settings and displayed in the list editor, click on the ***Display List*** command button.

   Change to the ***Structure settings*** option card if additional settings concerning the attribute list structure are to be determined (cumulate identical entries, sort order, etc).

Determine structure settings for attribute list

The menu command ***Create Attribute List*** can be used to generated a list which contains the component text allocated to the current model.

1. Start the menu command ***Create Attribute List***.
   The ***Create list*** dialog box is opened.
2. In the ***Select attribute*** option card, determine which data fields are to be taken over in the list and which order they should have.
3. Change to the ***Format columns*** option card in order to determine the output format of the list entries.
4. Change to the ***Structure settings*** option card to determine additional settings concerning the attribute list structure.

5. If the ***cumulate lines*** check box is active, the list contains every combination of component text.

   If the check box is not active, identical combinations of component text are only displayed in one line of the list.
   If the list contains a column with the standard attribute *@No*, the respective amount of entries which have this specific combination is also displayed.

6. If the ***acquire object-related data*** check box is active, when generating the list, all attribute values are taken into consideration, i.e. even those which are allocated to individual 2D objects. Otherwise, only values are entered in the list which are allocated to groups.

7. Activate the command by clicking on the list next to ***group to be disassembled***.
   In the dialog box which then appears, select the name of your current drawing sheet or one of the subordinate subgroups (or objects). Starting at this hierarchical level within the structure, the text allocated to components and component groups is determined and taken over into the list.

8. Activate the setting ***Disassemble by one level*** in the ***structure resolution*** text box, if only the attribute values of the ***group to be disassembled*** and their directly subordinate groups (or objects) are to be taken over in the attribute list.

   The ***Disassemble completely*** setting results in all the attributes of the ***group to be disassembled*** and all their subordinate groups (or objects) are taken over in the attribute list.

9. The ***sort according to*** text box, can be used to define the column, according to whose entries, the order of the lines in the list are to be determined.
   Additionally, select one of the ***Sort*** types.

10. If the attribute list is to be generated with the current settings and displayed in the list editor, click on the ***Display List*** command button.

Insert attribute list in drawing

In order to insert an attribute list in the drawing, proceed as follows:

1. Generate the attribute list using the menu command ***Extras, Attribute, Create attribute list***.

2. The BOM is then displayed in the *BeckerCAD* text editor and can be edited and saved.
3. If necessary, modify the BOM. The select the menu command ***Edit, Tag all***.
4. Select the menu command ***Edit, Copy***. The highlighted text is saved in the Windows clipboard.
5. Close the dialog box for the *BeckerCAD* text editor using ***File, Exit***. Also close the dialog box for the menu command ***Extras, Attribute, Create attribute list*** using the Command button ***Exit***.
6. ,  Select the command ***Text, Define text*** using the illustrated icons.
7. Position the cursor in the input text box for the text and insert the contents of the Windows clipboard using the key combination **Ctrl V**.

   The text can now be inserted in the drawing.

Close ODBC connection

The menu command ***Extras, Attribute, Close ODBC*** can be used to close all ODBC connections to databases or tables.

This is useful, for example, if, when aligning the connection, it has been determined that the databases are to be used exclusively. In this case, such a database could not be processed, as long as the connection with the defined SQL attributes still exists.

If a command is carried out later, whilst processing, for which an attribute value or component text is to be determined using a SQL prompt, the connection necessary for this is reconnected.

4.20 Use pixel drawings

In *BeckerCAD* pixel drawings of different formats can be loaded.

Determine pixel drawing settings

Using the menu command ***Settings, Pixel Drawing***, the general defaults for pixel drawings can be determined.

The defaults defined here can be taken over for positioning each time a pixel drawing is loaded using the menu command ***Extras, Pixel Drawing, Insert***.

If these defaults are saved in a template, they are available in all models which are based on this template.

In the dialog box which is then displayed, the following defaults can be determined:

drag as filled-in
The CAD Menu command ***Transform, Geo Points*** can be used to move pixel drawings quickly and easily between their geo points or to scale them.

If this setting is active, the modifications to the positions and sizes in the pixel drawing will be followed by a recalculation and a new display.

If the setting is not active, the modifications will be calculated only for the frame representing the pixel drawing. After the modifications have been completed, the pixel drawing will be recalculated and drawn again.

insert with frame
This command is used to insert the pixel drawing with a frame inside the drawing area.

The frame can later be switched on/off for individual pixel drawings using the menu command *Extras, Pixel Drawing, Frame On/Off*.

Display
The options in this part of the dialog box can be used to change how the pixel drawings are to be displayed after insertion. They can be displayed in their ***original colours***, in ***grey levels*** or monochrome (***black/white***).

Please note:

> You will be able to change the position of coloured pixel drawings more quickly or reduce or enlarge them with the CAD Menu command ***Transform, Geo Points***, if the setting is temporarily on grey shades or on monochrome.
> In addition, the setting ***drag as filled-in*** should be deactivated: this will save considerable time, particularly when a large pixel drawing has to be redrawn.

Insert pixel drawing

The following method can be used to insert pixel drawings into the active partial drawing that the current drawing contains:

1. Select the ***Extras, Pixel Drawing, Insert***.
2. Determine the file names and, if necessary, drive and folder.
   The name of the selected file is displayed in the ***current file*** text

box and the dimension in the X and Y directions are displayed in the ***Dimensions*** block.

3. In the ***file type*** text box, determine the required pixel format. Please note that LZB compressed TIF files are not supported.
4. Activate the ***retain page relation*** check box, if the pixel drawing is to be inserted using the ***variable size*** option, without it being distorted.

   Otherwise, select the ***original size*** option.
5. Activate the ***Insert Pixel Drawing*** command button and define the first corner of the pixel drawing.

   If the ***original size*** option is selected, the pixel drawing is inserted.

   If the ***variable size*** option is selected, define the opposite corner of the pixel drawing. The pixel drawing is then inserted in the frame so generated.

When using larger pixel drawings, the insertion could take more time.

Please note:

Pixel drawings are not inserted in the current group but in the current partial drawing. Move the pixel drawing into the desired group if needed.

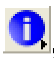
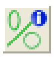
A pixel drawing can be inserted using Drag&Drop. If you pull the pixel drawing into an opened drawing, the pixel drawing will be saved in this drawing. If you let the pixel drawing drop into the program window, a new drawing is set up. The drawing borders are automatically adapted to the drawing borders on the pixel drawing.

Pixel Drawings are saved in the model and still keep their reference to the original source. So it is still ensured that changes to the source will be taken over when loading the model again.

If pixel drawings are not found in the original directory or in the current folder for pixel drawings when loading the model, the pixel drawing will be saved in the current folder for pixel drawings.

The folder for the Pixel Drawing can be changed or a new Pixel Drawing can be selected as follows:

Graphic Selection

1. ,  Activate the command ***Edit Object.***
2. Click on the frame of the Pixel Drawing.
3. Click the displayed command button  and select the desired file.

Using the Model Explorer

1. Activate the component (group, partial drawing, workplane) which contains the Pixel Drawing in the structure tree.
2. Open the context menu by clicking with the right mouse button on the 

entry in the list view and select Properties or double click on the entry.
3. Click the displayed command button  and select the desired file.
5. Click the command button ***Open***.
6. ***Close*** the dialog box.

Transform pixel drawing using two control points

A two point transformation can be used to adjust a pixel drawing size and orientation to fit the 2D drawing.

Use the following method:

1. Select the ***Extras, Pixel Drawing, Transform*** command.
2. Identify the pixel drawing to be transformed using the cursor.
3. In the largest possible window, determine two control points in the pixel drawing whose direction and distance are to b used as the reference line.
4. Define two points in the drawing.
This distance and direction determines the new length and direction of the reference line.

Please note:
If lines of a transformed pixel drawing are to be vectorised, they must firstly be fixed.

Invert pixel drawing

The ***Extras, Pixel Drawing, Invert*** command will only have any effect on monochrome pixel drawings. The process changes the colours of the background and foreground.
Identify the pixel drawing to be inverted.

Display pixel drawing with/without frame

Depending on the pixel drawing settings, pixel drawings can be inserted directly with or without frames.

The menu command ***Extras, Pixel Drawing, Frame On/Off*** can later be used to switch the frame display of individual pixel drawings on/off.

1. Identify the respective pixel drawing in order to switch the frame on or off.

4.21 Drawings and partial drawings

When *BeckerCAD* is started with the template STANDARD.TPL, that is supplied as standard to all users, a window with a 2D drawing sheet will be available in addition to the 3D window.

If the model you are working on is to refer to several 2D Drawings rather than just one, you can do so by using the Model Explorer to create them.
Several view windows can be opened for each drawing. A different view can be displayed in each window, i.e. a different view of the drawing. You can also change from one window to another while you are processing the drawing. The menu commands Window, Split Horizontally and Split Vertically can be used to save space since the current view window can be split.

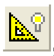
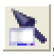
Any number of partial drawings can be worked on when you are dealing with a drawing.
These are work sections with a name and a scale that can be allocated authorised user access and display attributes of their own.

The advantage of partial drawings is that you will be able to create a standard sheet, for example, that will contain a text box, separate sections of an assembly drawing or detailed drawings to different scales, each separated from the others for easier processing.

Please note:

When you intend to use different scales in one 2D drawing, you will have to use the appropriate number of partial drawings.

Each of the 2D objects is completed in separate partial drawings that is treated as the active partial drawing for this purpose.
In each of the partial drawings that is not active you can specify whether the objects in it is to be available for snapping points. (With access read only or without access). You can also specify the type of display for the partial drawings that are not active: visible or hidden.

,  ,  When you are processing, transforming and dimensioning 2D objects, you can use the displayed icons at the top of the 2D window to select whether the objects that are in the partial drawing that is not currently active are also to be considered.

Please note:

The commands ***Transform, Move*** or ***Move with copy*** can be used to move or copy 2D objects of a drawing or partial drawing to another.
Firstly, the objects must be taken over in the selection list before loading the command.

Use a number of drawing

When you are working in a model that consists of several 2D drawings, e.g. with several separate drawings, you find it very helpful to move or copy all the objects in them to a single composite drawing.
An alternative is for you to load additional drawings from other models whilst processing the model. This is done with the menu command ***File, Open Model***.

Using the menu command ***Insert, New 2D Drawing*** or using the ***Model Explorer***, you can set up new drawings within a model. Existing drawings can be renamed using the ***Model Explorer*** and erased. Closed drawing windows can be opened again, by defining the drawing in the ***Model Explorer*** as the active 2D/3D context and executing the ***View, New 2D Drawing View*** menu command, or using the ***New, New View*** context menu that opens on a drawing in the ***Model Explorer***.

In order to select one of the opened 2D drawing windows as the active window, activate the required area using the ***Model Explorer***, use the list at the end of the ***Window*** menu or change windows by pressing ***Ctrl Tab***.

Set up new drawing

Use the following method to set up an additional 2D drawing:

Using menu command

1. Select the menu command ***Insert, New 2D Drawing.***

Using the Model Explorer

1. Open the ***Model Explorer*** using the ***View*** menu.

2. Select the ***Model Explorer*** tab.

3. Tag the topmost folder of the structure (path with model name).

4. Select the key combination ***Ctrl+N*** or use the context menu to select the ***New / 2D Drawing*** option.

Please note:

Each drawing can be allocated their own dimensions and partial drawings.

The layer structure and other 2D specific settings are valid, independent of the selected drawing.

Add drawing

If you need to add any 2D drawings to a drawing already in use, though they are saved in another model, you can use the following method:

1. Select the menu command ***File, Open Model***.
2. Use the ***Search in*** field in the dialog box to find the folder specified in ***Settings, Folders*** for the models.
   You may have to look at a different folder.
3. You will see a list of MOD files in which you can tag the one you want; if it is not there, type the name of the file in the box.
4. You can display a preview of the drawings in the model by activating ***Show Preview***.
5. You will see a list of the drawings on the lower left and you can then tag the one you want to load.
6. Click ***Add drawing***.

Please note:

The ***Add Drawing*** command can be used to add a copy of the original drawing. No references exist between the original drawing and the added drawing.
If you simply want to take over objects from the added drawing into your opened model, you should delete the drawing using the menu bar using the ***Model Explorer*** and selecting the ***Drawing, Erase*** command, after you have moved the required objects dynamically into your drawing with or without a copy. Simply closing added 2D windows is not sufficient. The window is still saved in the file.

The Drawing Manager can be opened as required using the ***Model Explorer***:
- tag any 2D drawing in the Model Explorer
- select the option ***New, Drawing Manager*** from the context menu

Please note:

If the drawing that you load in addition contains model views, they

no longer have a link to the 3D model from which they were extracted.
This means that these model views are constant illustrations. They can be processed further by using ***Process Model Views, Disassemble Model View*** or ***Disassemble 2D Objects*** replacing them with 2D linear objects.

Rename drawing

The description of the current drawing is displayed in the heading of the 2D drawing window if the window has been reduced. If the display is full-screen it is in the header of the main window.

Use the following method to alter the name of a drawing:

1. Using the ***View*** menu, open the ***Model Explorer.***

2. Select the ***Model Explorer*** tab.

3. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All***.

4. Tag the 2D drawing you wish to rename and press the ***F2*** function key or select the ***Rename*** option from the context menu.

5. Enter the new name and confirm the entry with the Enter key.

Delete drawing

If you no longer require a 2D drawing, the partial drawings within this and the objects and structures too and wish to delete them, proceed as follows:

1. Select the menu command ***View, Model Explorer***.

2. Select the ***Model Explorer*** tab.

3. In the tree structure, tag the drawing to be deleted.

4. Press the ***Del.*** Key on the keyboard or select the ***Delete*** command from the context menu.

5. Answer the prompt with ***Yes*** if you are sure that the selected drawing is to be deleted.

Determine settings for drawing

The dialog box ***Settings, Drawing*** can be used to determine the settings for the currently selected 2D drawing. Open the dialog box using the menu command ***Settings, Drawing*** or using the ***Model Explorer***, by opening the context menu on any partial drawing and selecting the ***New, Drawing Settings*** option. Please ensure that the 2D drawing is active.

<u>sheet format</u>

The alterations that are made in this section affecting a sheet format or a display of the drawing borders are activated by clicking the ***Accept*** button.
The ***currently set*** size of the drawing is displayed in the header of the 2D drawing or – if in full-screen display – in the main window.

standard sheet format
This list can be used to select a standard format from DIN A0 to DIN A4 as the sheet format in portrait or landscape format.
If the dimensions of this format are larger than those of the ***maximum print area*** of your output device, either select a smaller format or enter suitable values in the ***current size*** text boxes.

current size
Select this option, if the maximum print face of your output device is smaller than the selected standard format.

In this case, determine the ***maximum print area*** using the menu command ***File, Print*** and enter this or smaller values for the X and Y directions.

Once values have been entered in these text boxes, the entry ***Freely defined*** is displayed in the ***standard sheet format*** text box.

define drawing window
If the dimensions are to be determined by dragging a rectangle using the cursor, click on this command button and define the rectangle in the 2D drawing window.

Create standard sheet format
After clicking the displayed icon a new standard sheet format is inserted in the list of standard sheet formats with the current name, size and the coordinates for the origin.
Please note:
Define user defined standard sheet formats in a template (*.TPL), to use them for new models automatically.

Change standard sheet format
Click this button after the size or the origin of the marked standard sheet format has been changed, to accept the changes.

Delete standard sheet format
Click this button, to erase the current standard sheet format from the list.

origin
If values are entered in these text boxes, this results in a displacement of the drawing area from the origin of the GCS in the direction of the X and/or Y-axis.
If, for example, the values 10, 10 are entered, this means that the left bottom corner of the defined drawing area no longer has the coordinates 0, 0, but the coordinates 10, 10.

display drawing boundaries
If this check box is active, the borders of the defined drawing sheet are drawn as grey lines for an optical check.

insert standard sheet
This check box can be used to call up a dialog box for selecting a standard sheet symbol that will be inserted in the partial drawing set aside for the standard sheets.
Further information on this is to be found in the chapter ***Generate and use own standard sheets***, ***Insert or replace standard sheet****.*

Partial drawings and details

In this section of the dialog box you can set up the partial drawings or erase them; you can also edit their attributes. Further information on this is to be found in the following sections.

Set up partial drawing

When you want to work in a 2D drawing with different scales, you have to set up the appropriate number of partial drawings.
If, for example, the main scale in the drawing is 1:10 and you want to draw a detail showing something twice the size as an enlargement, you must set up an additional partial drawing with the scale of 1:5.

Using the dialog box Settings, Drawing

1. Select the ***Settings, Drawing*** command.
2. Type a ***name*** for the partial drawing.
3. Specify the ***scale*** for the partial drawing.
4. Where necessary, specify the status, i.e. the access rights for the partial drawing.
   This status will only be relevant if the partial drawing has not been

selected as the current partial drawing and if the mode ***Display Status*** is active (grey symbol).

read only
Objects in the partial drawing can be identified with the cursor in order to define new points.
Parallels to line objects in other partial drawings can be drawn to the identical scale.

without access
Objects in the partial drawing cannot be identified with the cursor.

do not display
Objects in the partial drawing will not be displayed.

5. Click this icon to conclude the definition of the partial drawing. It will then be inserted in the list of partial drawings.

   If there is already another partial drawing with the same type, name and scale, you will be offered the choice of creating a different name automatically (yes) or of altering the name yourself (no).

<u>**Using the Model Explorer**</u>

1. Using the ***View*** menu, open the ***Model Explorer..***
2. Select the ***Model Explorer*** tab.
3. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All***.
4. Tag the 2D drawing under which you wish to set up a new partial drawing.
5. Press the key combination ***Ctrl+N*** or select the option ***New, Partial Drawing*** from the context menu.

Further information, concerning how to later process the scale or the status (access rights) for the new drawing can be found under the heading ***Process partial drawing properties***.

Please note:

Each partial drawing lies at the root of its own group structure and of all associated data, e.g. information on the attributes and the components.

Layer structure and the settings related to the 2D drawing apply to all the partial drawings.

Process partial drawing attributes

To process the attributes relating to a partial drawing proceed as follows:

- in the ***Model Explorer*** select the context menu command ***New, Drawing Settings*** on a partial drawing

or

- select the menu command ***Settings, Drawing***.

Settings for displaying partial drawings

These two icons are for specifying the colours indicating access rights to all the partial drawings that are not active.

 **display status** (grey symbol)
This display mode must be active to display in accordance with the access rights that have been specified.

 **suppress status** (coloured symbol)
The objects in all the partial drawings will be displayed in accordance with the attributes that are specified in the layers or with reference to the objects themselves.

Settings for individual partial drawings

The name, the scale (but not for a standard sheet partial drawing) and access rights for partial drawings can be altered as required.

First select the line in the partial drawing list and then complete the alterations in this line.

name
Type the new name and then press *Enter* to confirm.

scale
Type the values in the boxes and then press *Enter* to confirm.

When the scale is altered the position in the GCS and the lengths of existing objects will not be changed; the display will be either to a smaller or to a larger scale relative to the sizes in the drawing.

The text and dimensions will retain their original sizes. Their positions cannot, in some cases, be adapted automatically and so you will have to do this yourself.
The spacing of the hatching will not be altered.

status

This can be used to change the status, i.e. the access rights to the objects in a partial drawing that is not active and is tagged with the symbol. If the symbol is yellow it shows that the partial drawing is active, otherwise its grey.
The display will be in accordance with the status defined for the access rights, if the display mode ***Display Status*** is active.

read only
Objects in the partial drawing can be identified with the cursor to define points.

without access
Objects in the partial drawing cannot be identified with the cursor.

do not display
Objects in the partial drawing will not be displayed.

Change Status in the Model Explorer

The status of partial drawings (*visible* / *invisible* or *active* / *inactive*) can also be changed in the context menu of the ***Model Explorer***.

If a partial drawing or a workplane is set to invisible also all the objects inside are invisible. In a workplane also the frame of the workplane is invisible. Partial drawings and workplanes with the status invisible and with that implicitely with the status inactive are marked with the symbol .
When deactivating a partial drawing or workplane, the contained objects are not pickable. This status is marked with the symbol .

The status of a partial drawing or a workplane can be changed in the ***Model Explorer*** as follows:

1. Select the command ***Model Explorer*** in the menu ***View***.
2. Tag the partial drawing or workplane to be changed.
3. Start the context menu by pressing the right mouse button placed on the partial drawing or workplane and select the desired option in the sub menu ***Visibility / Pickability***.
For this, also the listed key combinations can be used.

in the structure tree	in the detailed view	key command
Toggle visibility	Toggle visibility	O
Toggle pickability	Toggle pickability	P
Toggle visibility and apply to subtree		Strg+Alt+O

Toggle pickability and apply to subtree		Strg+Alt+P
Enable visibility of tree	Enable visibility	Strg+O
Disable visibility of tree	Disable visibility	Shift+Strg+O
Enable pickability of tree	Enable pickability	Strg+P
Disable pickability of tree	Inaktiv	Shift+Strg+P

Please note:

The name of a partial drawing can be changed directly in the ***Model Explorer***. In the ***Model Explorer***, tag the required partial drawing and press the ***F2*** function key. Then enter the required name and confirm the entry with the Enter key.

Activate partial drawing

2D objects are always drawn in the current partial drawing.
Provided the user has been granted access rights, points on 2D objects in other drawings can also be snapped. If these partial drawings are drawn to scales different from the one for the current partial drawing, the visible point will always be located.

When you are processing 2D objects you can also use objects that are in other partial drawings. In such a case you will be able to use a context dialog to specify the partial drawings that are to be taken into consideration.

A partial drawing can be activated in a number of different ways:

Select in the 2D symbol bar

Open the list 

to the very left of the 2D toolbar and select the required partial drawing.
The scale of the active partial drawing is displayed in the header of the 2D drawing window or – if in full screen display – in the header of the main window.

Select via object

 Click the icon in the symbol bar and then identify an object from those in the partial drawing.

Selection using the *Model Explorer*

For the setting  ***Activate selected components*** in the ***Model Explorer***, a partial drawing automatically becomes the active partial drawing (and also the target area for new objects) if this is identified in the tree structure.

For the settings  ***Do not activate selected components*** you have the following options available to activate a partial drawing:

Tag the required partial drawing and press the key(s)

- ***F12*** or select the option ***Declare to current 2D/3D context*** from the context menu if the partial drawing is also to be activated as the target area for new objects.

or

- ***Shift+F12*** (context menu: ***Declare to current work area***) if you only wish to activate the partial drawing. A previously activated group within this partial drawing however remains as the active target area for new objects.

Select in dialog box
Activate the ***Settings, Drawing*** command. Select the partial drawing from all those in the list and then click . Then the grey symbol turns to yellow.

Please note:

A partial drawing is similar to a construction plane in the 3D window in that is described as an **active 2D section**.
This means that the ***Process, Erase Active 2D Section*** command can be used to erase all the 2D objects and the group structure that occur in a partial drawing.

You can use the ***Transform, Move****,* without or with copy command to transfer 2D objects from one partial drawing to another.

Copy partial drawings

If you wish to copy the whole contents of a partial drawing i.e. groups or the component structure as well as all the objects into another partial drawing, you can do this as follows:

1. Using the ***View*** menu, open the ***Model Explorer***.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. By clicking, select the partial drawing (2D drawing) or the workplane (3D model) you wish to copy.
4. Press the key combination ***Shift+S*** or select the ***Select / Select All Contents*** option from the context menu.

5. Select the command ***Transform, Move With Copy***.
6. As the starting point, determine a point with defined coordinates, e.g. X = 0.0, Y = 0.0.
7. in the list of partial drawings, select the partial drawing in which the structure and the objects are to be inserted.
8. For the target point, enter the same coordinates as for the starting point.

Erase partial drawing

The following methods can be used, depending on whether an entire partial drawing with all the components in it is to be erased or only the geometry is to be erased:

Erase geometry in partial drawing

1. Using the *View* menu, open the *Model Explorer*.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
3. By clicking, select the partial drawing (2D drawing) or the workplane (3D model) you wish to delete.
4. Press the key combination ***Ctrl+Alt+S*** or select the ***Select / Select All Geometry Objects*** option from the context menu.
5. Select the command ***Object/Delete Selection***.

or

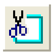
1. Make sure that the ***access to the current partial drawing*** is set.
2. Define a selection mode:

   Selection mode: All
   All 2D line objects touched or inside a selection by window are completely selected.

   Selection mode: Inside
   Only those 2D line objects are selected which are completely inside the selection window.

   Selection mode: Outside
   Only those 2D line objects are selected which are completely outside of the selection window.

Important:
Using this definition all the elements outside of the window are selected and maybe deleted if you choose so, even those elements which maybe are outside of the visible area of your model.

 **Selection mode: Clip**

2D line objects are clipped at the frame of the selection window. The Clip frame is shown in a green rectangle.
The following objects cannot be clipped:

Dimensions, Hatches, Symbols, 2D standard parts, finish characteristics, form- and position tolerances, welding symbols
Using this command a hatch can only be deleted by mouse click not by selection window.

3. Select the command ***Erase Object/Selection***.
4. Define a rectangle with the pressed mouse button covering the area to be deleted.

<u>**Erase geometry and group structure in partial drawing**</u>

Using the menu:

1. Define the partial drawing as the current drawing.
2. Select the menu command ***Process, Delete 2D Area.***
   All 2D objects and the group structure will be deleted.

Using the *Model Explorer:*

1. Using the ***View*** menu, open the ***Model Explorer***.
2. Select the ***Model Explorer*** tab.
3. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***
4. By clicking, select the partial drawing (2D drawing) or the workplane (3D model) whose geometry and group structure you wish to delete.
5. Press the key combination ***Shift+S*** or select the ***Select / Select All Contents*** option from the context menu..
   <u>or</u>
   In the list view, tag all the objects and groups.
6. Press the ***Del.*** Key on the keyboard.

<u>**Erase partial drawing and geometry**</u>

Using the menu:

1. Select the menu command ***Settings, Drawing.***
2. In the list of partial drawings, select the partial drawing to be deleted.
3.  After clicking on this button, the partial drawing with the group structures within it will be deleted.
   If the partial drawing contains objects, a security prompt is displayed before the deletion is carried out.

Using the *Model Explorer:*

1. Using the ***View*** menu, open the ***Model Explorer***.
2. If necessary, open the structure branch using the ***F11*** key or using the ***Context menu / Open Branch*** or ***Open All.***

   A pixel drawing can be inserted using Drag&Drop. If you pull the pixel drawing into an opened drawing, the pixel drawing will be saved in this drawing. If you let the pixel drawing drop into the program window, a new drawing is set up. The drawing borders are automatically adapted to the drawing borders on the pixel drawing.
3. By clicking, select the partial drawing (2D drawing) or the workplane (3D model) you wish to delete.
4. Press the ***Del.*** Key on the keyboard.
   If the partial drawing contains objects, a security prompt is displayed before the deletion is carried out.

4.22 Generate and use own standard sheets

If you do not want to work with the standard sheet symbols that belong to the delivery in the standard sheet library, you can define any number of your own. These must be saved in the standard sheet library called NORMBLAT.SYL that is in the folder named \PROGRAM\SYL when you receive your software.

You can use a different folder by setting it up with the menu command ***Settings, Folders*** as the folder only for standard sheet symbols, e.g. the folder set up as part of the installation procedure with the name .\SSH.
Then you can move the file NORMBLAT.SYL into the new folder.

To create your own standard sheet, create a separate model under the name with which the symbol is to be saved in the standard sheet library. Then select a sheet format and create the standard sheet. By using the menu function ***Define, 2D Standard Sheet, store as***

geometry in Model, the standard sheet is included in your standard sheet library.

In general, you should create a separate model for each standard sheet, from which the standard sheet symbol can be generated.
If you want to change existing standard sheet symbols later, use the command ***Define, 2D Standard Sheet, store as geometry in Model*** to create a corresponding model for editing the standard sheet.

General process

1. Select the menu command ***File, New Model***, and use the STANDARD.TPL as template for example.
2. Save the model under the name with which the standard sheet is to be entered in the library.
3. If you are in the 3D window activate the 2D window using the menu command ***Windows***.
4. In the menu command ***Settings, Drawing*** define the desired size for the standard sheet in a 1:1 scale.

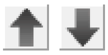
   When determining the sheet format, please observe the notes on the maximum print area.
5. Activate the partial drawing with type ***Standard Sheet*** in the ***2D Drawings Settings*** dialog. .
6. Preferably create your own layers for the geometry of the standard sheet.
7. Draw a frame geometry and the title block.
8. Place the field names such as *name, drawing number* and *scale* in the title block.
9. Place the variable texts, starting with a #, which should be offered for editing when inserting a standard sheet.
   All texts beginning with '#' are converted into attributes when the symbol is saved, e.g. the text '#`Naming`' is converted into a text attribute '`Naming`'.

   Other text attributes (e.g. formulas for Model name with path or automatically generated date) must be defined in advance. Then enter the name of the defined attribute, also beginning with a #, in the title block.
10. If desired, insert a logo into your title block using ***Extras, Pixel Drawings, Insert***.

11. Call the menu command ***Define, 2D Standard Sheet, store as symbol in standard sheet library***.

12. After you have confirmed the messages, the ***Edit Attributes*** dialog opens.

    In the fields in the *Value* column, you can enter standard values that serve as text proposals when you call up the standard sheet and can be edited if necessary.

     Use the arrow keys shown to change the order of the attributes. Select an attribute in the list and click on the respective icon.

     To remove an attribute from the list, select the attribute in the list and click on the ***Delete selected attribute*** button.

13. Click ***OK*** to save the symbol in the standard sheet library.

    With ***Cancel*** only the changes in the ***Edit Attributes*** dialog are discarded. The symbol is always saved.

Please note:

If attributes already existed in the model of the standard sheet, their settings are used. Before you create the standard sheet, you should therefore verify the settings of the existing attributes or delete attributes that are not required.

Insert or replace standard sheet

If standard sheet symbols contained in the NORMBLAT.SYL standard sheet library are to be used as drawing frames with text field for your drawing, proceed as follows:

1. Change to a 2D drawing window and then select the menu command ***Settings, Drawing***.

2. Select the menu command ***Insert, Standard Sheet***.
   The dialog box for the selection of a standard sheet is then displayed.

3. Open the ***Standard sheet selection*** list and click on the name of the required standard sheet.
   In the ***current drawing*** dimensions text boxes, the currently determined size of the drawing is displayed, in the ***new drawing*** dimensions, the size of the selected standard sheet is displayed.

4. Start the insertion of the standard sheet using ***OK***.

5. Select whether the new drawing dimension are to be taken over or the old ones are to be retained.

If the partial drawing already contains a symbol from the standard sheet library, this is erased.
Then, the selected standard sheet symbol is inserted in the origin of the *standard sheet area (*partial drawing) or at the position of the previous standard sheet symbol.

6. In the dialog box which then appears, all variable text of the text field are displayed which can be edited. Enter the required text.

If there was already a standard sheet symbol that also contains text attributes, the old texts will be inserted in the new text field, provided that they are attributes of the identical type and with the identical name.

If self-defined standard sheet symbols are used which do not have attributes, this work step cannot be carried out.

Please note:

The standard sheet library named NORMBLAT.SYL is in the folder named .\PROGRAM\SYL. as part of the software you receive.
If you want to use a different folder, you must set it up by using the menu command ***Settings, Folders*** as the folder for standard sheet symbols, e.g. the folder supplied and named .\SSH.
Then move the file named NORMBLAT.SYL into this folder.

Modify standard sheet entries

If you use standard sheet symbols from the delivered standard sheet library for your drawings, the text field entries which are allocated as attributes of type text, can later be modified in the following way:

One method you could use to change entries is to insert the same standard sheet symbol a second time. If you do not want to do this, use the following method:

1. In the 2D toolbar, activate the *Standard sheet area* partial drawing.

2. Select the menu command ***Extras, Attribute, Process***.

3. Using the middle mouse button (or Ctrl + right mouse button), load a context menu and select the option ***Objects according to Selection***.

If the standard sheet symbol has been disassembled, the attributes can only be selected using the group.

4. Identify any line of the standard sheet symbol.
A dialog box is then displayed in which you can modify all variable text of the text field.

Please note:

Another possibility to change texts in standard sheets:
Open the ***Structure layer tree*** inside the ***Model Explorer*** and tag the standard sheet. Click with the right mouse button on the ***Symbol*** in the right side of the Model Explorer and select the option ***Attributes, Edit attributes*** to change the texts.

If you use standard sheet symbols in which the text is not inserted as an attribute but using the ***Text, Define Text*** command, you cannot use the procedure described above.

In this case, the symbol must firstly be disassembled using the ***Disassemble 2D Objects*** or ***Process 2D, Disassemble Symbol*** command.
The text can then be edited using the ***Text, Edit*** command.

4.23 Print drawings

The currently selected 2D drawing can be output using an installed Windows printer driver. All the objects that are visible will be printed.

Draw and print according to scale

If your drawings are to be dimensioned correctly and output on a specific paper format, e.g. DIN A4, proceed in the following way:

Determine maximum print face

In order to determine which dimensions the maximum face printer can use, select the menu command ***File, Print***.

If a different printer or output size is to be modified, select the ***Setup*** command button. Carry out the necessary modification and confirm with ***OK***.

Now, according to their settings, the print dimensions of the ***maximum print area*** are displayed in the second line of the dialog box.
If the format DIN A4 landscape has been selected for the alignment, the maximum print face is usually smaller than this format e.g. 287 x 198.

Select portrait or landscape for printing. The preview is updated automatically.

Determine drawing size

If you wish to generate your drawing (firstly) **without a standard sheet**, activate the 2D drawing window and select the ***Settings, Drawing*** menu command.
Using the ***standard sheet format*** list, select the required DIN format. The dimensions belonging to this format are entered in the ***current size*** text boxes.
If one of these values is larger than the maximum print area, a drawing with these dimensions cannot be printed 1:1.
In this case, select a smaller DIN format or in the ***current size*** text boxes, enter the values of the maximum print area.
For the given example values these values are 287 x 198.

If you wish to generate your drawing **with a standard sheet**, the drawing sizes can be taken over from this when inserting the standard sheet symbol.

Insert standard sheet

Using the menu command ***Settings, Drawing, Insert Standard Sheet***, insert the required standard sheet. Select whether its dimensions are to be taken over for the drawing sizes.

The partial drawing reserved for standard sheets should only be used for the standard sheet, i.e. only for the drawing frame with text field.

Determine of partial drawings

When dimensioning lengths and distanced, the actual values are always determined. This means that **all 2D line objects must be drawn with their actual dimensions**.
On the selected format, objects could be selected which are a maximum of 287 mm long in the X direction and 198 mm in the Y direction.

If the work piece to be drawn has larger dimensions, e.g. 1300 x 500 mm, enter 1:5 for the ***scale***.
In this way, all objects which are drawn within this partial drawing, are displayed in reduced form.
Calculated, with this scale, a drawing area of 287*5 = 1435 x 198*5 = 990 is available.

If you want to work in the same drawing using **different scales**, e.g. to turn a part of your drawing into a detail illustration, you must use an additional partial drawing with the appropriate relative scale:
With reference to an initial scale of 1:5 this means a scale of 1:2.5 will result in an enlargement twice the original size.

Generate drawing
Activate the partial drawing drawn to the main scale that has been specified, e.g. 1:5.
The objects in the partial drawing *standard sheet area* become inactive and therefore cannot be processed or erased accidentally.

Draw all objects in their actual dimensions or accept the model views from solids which have also been displayed in their actual dimensions in the 3D-view window.

Once you wish to produce a detailed display of one individual part of your drawing, copy the appropriate objects from the active partial drawing into the additional partial drawing that has been defined.

Print drawing
When the drawing is printed all the objects that are visible will be printed.
The display of the objects can be suppressed by specifying, in the case of the layers or the partial drawings, that they are not to be displayed and thus not printed.

In order to print the drawing to the correct scale, select the menu command ***File, Print***. Enter the value 1:1 for the ***print scale***.
If necessary, check the size using the ***Preview*** command button. Start the printing using the ***Print*** command button.

If the maximum print area is larger than the dimension of the drawing, move values can be entered in the text boxes by which the drawing is to be moved vertically or horizontally on the sheet.
The total of the drawing dimensions and the values for the move cannot exceed the dimensions of the maximum print area.

If the output must not be carried out according to scale, for the ***drawing units*** the ***fit into print area*** option can be selected. A value for the print scale can also be entered. This determines by which factor the drawing is to be reduced or enlarged for the output.

Determine print settings

 Click the ***Print*** button in the 2D window or start the menu command ***File***, ***Print***. A dialog box is displayed which can be used to determine the settings for printing, start the print procedure as well as save the print settings in a model.

current printer
The name of the Windows standard printer is displayed in this text box.

If the drawing is to be output on another printer, select the ***Setup*** command button in order to select another printer.

max. print area
Shows the dimensions (in mm) of the maximum area, which can be used by your plotter/printer relating to the determined paper size.
This value is taken over from the Windows printer settings and cannot be entered.

When the drawing is to be printed out to scale, you must not create the drawing itself or the standard sheet for it, until the drawing sites that are specified (x,y) are not greater than the maximum print area that your printer can cope with.

number of copies
Determines the number of print outs.

Output in file
If this setting is active, the print output is redirected to a file in the set Print directory. The file name text box is available for any entry to be made in it if required. You have the following options available to you for generating the print file:

- The file contains the current model file name with the extension PRN, if no name is entered in the ***file name*** text box and the switch ***Default: model name*** has been activated.
- If you enter the name in the ***file name*** text box and have not activated the switch ***Default: model name***, the file has the current drawing name with the extension.
- If you define a file name in the entry line without an extension, independently of the switch setting ***Default: model name***, the print file is generated with the file name entered and the extension PRN.
- If you define a file name in the entry line with an extension, independently of the switch settings ***Default: model name***, the print file is generated with the entered file name and the defined extension.
-  Click the displayed icon and define the folder and the file name for the print file.

Fit into print area
Select this setting if you would like to print the drawing to the currently selected paper size (enlarged or reduced).
The drawing is printed to scale with this setting only if the current dimensions of the drawing are equal to the print area of the currently selected paper size.

Print scale

After you have deactivated the setting ***fit into print area***, you now have the following setting options if the drawing is not to be printed to the print area but rather to scale.

- Select a desired scale from the list of predefined print scales.
- Enter into the ***print scale*** input field: A value in **percent** for the scale. An example of an input if the drawing is to be printed in a reduced scale of 1:2 is:
  50% (confirm input with the Enter key)
- Enter into the ***print scale*** input field: (in the percent display area) a **factor** for the scale. An example of an input if the drawing is to be printed in a reduced scale of 1:10 is:
  0.1 (confirm input with the Enter key)
- Enter into the ***print scale*** input field: (in the percent display area) a **scale** to which it should be scaled. An example of an input if the drawing is to be printed in a reduced scale of 1:25 is:
  1:25 (confirm input with the Enter key)

scale line widths

On screen the 2D linear objects will be displayed in the line width that was allocated with reference to layers or objects, if ***Settings, Options*** was used to activate the option ***Display thick lines***. If this option is not active, they will be displayed with a width of one pixel.

Independent of the display on the screen, line widths are always output with their defined values when printing.

If the option is not active, the line widths are output as they are displayed.

If the option is active, the factor for the print scale also has an effect on the line widths.

print black/white

If this setting is active, all visible 2D objects are printed with the colour black.
If the setting is not active, the visible 2D objects are output according to the printer settings or possibilities, in colour or grey levels.

Please note:

If you are working with a number of partial drawings and wish to output the objects they contain in colour, firstly select the menu command ***Settings, Drawing***.

 Activate the setting ***Suppress status*** (coloured symbol)

and all the objects are displayed an printed according to their object or layer specific colour.

print texts clipped
Activate this setting if you are using a **pen plotter**. By doing this your clipped texts and scale texts will be printed, but will be without the clipped area that would otherwise be plotted as a filled area.

Deactivate this setting, however, if you are working with a grid device: this is the only way to take care of the on-screen clipping of the texts even when they are to be printed.

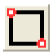
drawing boundaries
Activate this option if all objects which are within the set drawing borders of the 2D drawing, are to be plotted.

current section
Activate this option in order to plot the currently displayed screen content.

user defined boundaries
Activate this option to print a defined section of the drawing. All elements outside the defined section will not be printed.

Boundaries by values: In the input fileds the rectangular size can be defined by X and Y coordinates; (x1,y1) and (x2,y2). the values must be seperated by comms. Please consider that the values are adjusted to the paper size and not to the drawing coordinates if a scale different from 1:1 is used.

Geometry box
All visible objects in the drawing will be printed – even those, outside the page format.

Boundaries by 2 points in the drawing: After clicking the  icon the dialogue window is minimized and the rectangle can be defined by two points. After defining the second point the dialogue window is opened again. The defined coordinates of the section are shown in the input field and can be edited there.

defined size
The size that the face should actually be, can be entered here, in which the output of the drawing is to fit in to. These values cannot be larger than the value entered in the maximum print area text box.

If the values determined from the printer driver for the ***max. print area*** of your printer are larger than the output size of the drawing, the actual horizontal and vertical dimensions can be entered here.

If, on a DIN A0 sheet for example, a drawing is to be output in DIN A4 format, the actual dimensions of the drawing can be entered here and a specific position determined for the drawing on the DIN A0 sheet by entering values in the ***displacement*** text box.

displacement
These values result in a zero point displacement of the drawing on the paper.
This, for example, offers the possibility of positioning four DIN A3 drawings on a DIN A1 sheet or to leave a margin free on the left of the sheet.

The sum of the values for the defined size and displacement in horizontal and vertical directions cannot extend the dimensions of the ***maximum print area***.

Orientation
In this area the orientation for the print is defined. You can select between

Upper - Left	Upper - Middle	Upper - Right
Middle - Left	Middle - Middle	Middle - Right
Lower - Left	Upper - Middle	Upper - Right

Additionally the print can be oriented tot he printable area or the paper edge and can be moved horizontally or vertically.

Rotation
Select, if the drawing should be rotated for the print. This can be defined in 90° steps.

Printable Area
This values are mainly for information. They display the size of the maximum print area. If the area, in which the drawing should be printed in should fit, but is smaller, the print area can be reduced here. Depending if the printer can print without a margin, the margin left and top is displayed. If the size and the margin should be changed, the sum of both must be smaller then the maximum print area.
The values can be taken over from the current printer driver or from previously defined model parameters.

drawing size
The actual dimensions for outputting the drawing are displayed for checking purposes, i.e. a window is calculated which contains all

drawing objects, even objects which are possible outside the defined drawing borders.

If the dimensions of this section are to large, a warning message is displayed during the print process.

Preview
The black rectangle identifies the paper size.
The green rectangle identifies the ***max. print area***.
The blue rectangle identifies the ***defined size***.
The red rectangle identifies the position of the ***drawing***. If the respective command button is pressed, a preview of the drawing to be printed is displayed in addition to the dimensions of the area.

Save print settings in model

If the print settings and the scale are to be saved in a model, select the menu command ***File, Print***.

1. Determine the respective settings in the dialog box which appears.

2. Activate the command by clicking on the ***Save*** command button. When next loading the dialog box, this set is displayed again.

Please note:

If the model contains drawing sheets with different dimensions, the settings must be adapted according to the dimensions of the sheet to be printed.

By using the menu command File, Save/Load Parameter Sets, the plot settings can be loaded and taken over from a MPS file.

Printer setup

If a different printer to the Windows standard printer is to be used for output or the settings and attributes are to be modified, select the menu command ***File, Print***.

1. Using the ***Setup*** command button, load the Windows dialog box for printer setup.

2. Carry out the necessary modifications and confirm with ***OK***.

Print directly from the interface

You can print out the content of the active window with the currently set print parameters using a user-defined icon. Using the menu command

File, Print you can set the print parameters and also start the print procedure. Proceed as follows:

<u>Using the menu command ***File, Print***:</u>

1. If necessary, using Setup, determine another printer to the Windows standard printer (e.g. a PDF-Printer) or modify the settings.
2. Determine the required settings for the output of the drawing. Check the size and position in the preview window.

Start the print procedure with Print.

<u>Using a user defined icon:</u>

1. Firstly create an icon with which the print procedure is to be started. This is described in the chapter Use user defined symbol bar.
the print command can be found in the Symbol bar dialog box in the structure under General CAD Commands / In/Output.

2. If the Argument text box is left empty, after the command is selected, the active window is printed with the current print parameters and the current standard printer. This standard printer is saved for the duration of the work session and further print procedures (also with other models) are carried out with this printer.
Only once *BeckerCAD* is restarted, will the active standard printer be determined the first time the command is pressed.
You also have the opportunity to enhance the print procedure with parameters in the Argument text box. Enter the parameters as follows:

<Default/CADdy/empty><PICTURE/SCREEN/empty><File name>

<u>Parameter 1</u>

<Default**>** determines, for each print procedure, the standard printer currently defined under Windows.

<CADdy**>** uses the printer last used for the menu command Define, Print.

<Printer name>
for direct printing via a named printer, e.g. a PDF printer to create

PDF´s.

<> if no parameter is entered, the standard printer used for the first print will also be used for further print procedures during this work session.

Parameter 2

<PICTURE**>** prints within the current drawing boundaries

<SCREEN**>** prints the section currently visible in the window

<> takes over the current setting in the File, Print dialog box

Parameter 3

<File name**>** if, in the File, Print dialog box, you have activated the setting output in file, you can enter a name for the file. Please take note of the setting options available in the Determine print settings chapter.

Please note

If you only wish to define the 2nd and 3rd parameters, you must enter the empty field for the first or the first and second parameters.

Print with Watermark

A watermark or a text can be added to the print of a drawing externally (e.g. paper, PDF) as well as into the clipboard.

To do so create a text file in the program directory with the name

WaterMark.txt

And copy the content of the example shown below into this file. With the following parameters the watermark can be adjusted. The parameters with a default value are optional.

Parameter

`font=<FontName> (Default: Arial)`
Name of the font

`colour=<r g b> (Default: 240 240 240)`
Colour of the text in RGB values

`scale=<factor> (Default: 1.0)`
Scale factor for the text

`direction=[horizontal | vertical | diagonal] (Default: diagonal)`
Wright direction

`preprint | postprint`
If the value `preprint` is set, the text will be behind the geometry, if the value `postprint` is set the text will be on top.

`readonce`
If this parameter is set, the file ***WaterMark.txt*** will only be read once in a CADdy session, so that changes to the file will only be active when the program is started again. Without this parameter, the file will be read for every new print.

Text input

`Sektion [text]`
Below the line [Text] up to the end of the file the text to be printed can be defined. The text can contain several lines as well as empty lines.

Example file

```
#
# This is an example file WaterMark.txt to print
# with a watermark
#
# Copy the content of this file to a text file with the name
# WaterMark.txt and save the file into the program directory to activate
# the print with a watermark
#
#

# Font (default: Arial)
font=Times New Roman

# Colour in RGB values (default: 240 240 240)
colour=200 200 200

# This factor is used to scale the text (default: 1.0)
scale=1.2

# Wright direction of the text (default: diagonal)
#direction=horizontal
#direction=vertical
direction=diagonal

# if 'preprint' is defined, the watermark will be printed before the
# geometry
preprint

# if 'postprint' is defined, the watermark will be printed after the
# geometry
#postprint
```

```
# if readonce is defined, the file WaterMark.txt is read only once in a
# session. Changes to the file will only be effective after a new program
# start. (default: not set!)
#readonce

#
# Text to be printed as watermark:
[text]
WARNING: This is a watermark defined in the file WaterMark.txt

The watermark can also contain empty lines.
```

4.24 Import and export 2D data

To exchange 2D-data with other programs you have the following options in *BeckerCAD:*
CADdy PIC-files and DXF-/DWG-files can be converted directly into the actual drawing.
The objects of the actual drawing can also be stored in this formats for the import into *CADdy* or in all other programs supporting these formats.

In addition, the content of the 2D window can be copied into the clipboard.

Copy 2D-data into another model

To transfer 2D-data from one model to another, besides the function to add a drawing to a model, you can also use the symbol definition. Start the function Define Symbol and store the 2D objects you wish to transfer into a Symbol.

Open the new model and the symbol library, insert the symbol and disassemble it.

Import 2D data

Using the menu command ***File, Import***, files can be converted and read in *CADdy* PIC or *CADdy* INF/LAY format and in DXF/DWG format.

Import CADdy PIC files

Select the ***File, Import, CADdy PIC File*** command in order to load a *CADdy* PIC file.

When executing this command, the PIC file is imported into the active 2D window of the current model. If necessary, set up a new model or a new drawing within the model if you do not wish to import the PIC file into the opened active 2D window.

For the case that you have accidentally imported the drawing in an incorrect file or an incorrect drawing, you can undo the step just made using the ***Undo*** command button.

Please refer to the information from chapter ***Importing CADdy files: technical information.***

1. In the ***CADdy Settings*** option card of the dialog box which then appears, determine the drive, folder and names of the *CADdy* drawing to be loaded in the ***drawing file*** text box.

   This can be entered in the text box or be searched for using the command button and then selecting the file.

2. If an info file (*.INF) and layer file (*.LAY) already exists with this name in the same folder, their drive, folder and names are entered automatically in the ***layer file*** and ***info file*** text boxes. Otherwise, enter the file names or determine them if necessary, using the command button.

   If no info file is entered, the object specific display attributes of the *CADdy* objects are taken over.

3. If the *CADdy* drawing to be loaded contains symbols, determine the folder in the ***CADdy symbol folder*** text box, in which the symbol files are to be searched for.

   Use the command button in order to select a folder and add to the list.
   If a folder is to be removed from the list, tag the respective one and then activate the command button.

4. In the ***CADdy font folder*** text box, enter the folder in which the *CADdy* fonts, determined with the info file, can be found. This can be entered in the text box or be searched for using the command button and then a folder selected.

5. In the ***File CADdy Plotter Settings*** file, enter the path and the file name for the *.PLN file defined for pen allocation.

   This can be carried out by making entries or by clicking the command button and then selecting a file.

6. **Only** for the case that no INF file is loaded when importing, you can use the ***take Over Display Properties*** command button to define whether the ***colours***, ***line types*** and ***line thickness*** of the objects are to be taken over as layer specific or object specific. If this option is activated, object properties are allocated respectively.

Using the colour command button in *BeckerCAD*, determine the required colour for the *CADdy* palette colour 0.

7. The ***Open Protocol File*** command button can be used to open the import protocol of the last import in order to be able to view any useful information or errors that may have occurred.

8. In the area of the ***CADdy objects*** determine the objects to be converted, which are then to be taken over in *BeckerCAD* as object oriented.

Fill areas are not imported if the option is deactivated. If the option is activated, the fill areas are displayed by polylines.

Colour areas are not imported if the option is deactivated.

Images (pixel drawings) are not imported if the option is deactivated.

Dimensionings are taken over as geometry if the option is deactivated, not as associative dimensioning. As such, however a 1:1 illustration of the PIC file guarantees dimensioning. "Old dimensions" in *CADdy* drawings are generally taken over as geometry. Every non associative dimension lies in a separate group.

Standard parts are imported as pure geometric data if the option is deactivated (individual elements such as lines and circles) and can no longer be edited as 2D standard parts.

9. Change to the ***CADdy^{++} Settings*** option card in order to make the necessary determinations for archiving symbols, the scale definition and units.

10. In the ***Accept symbol*** block, select one of the following options:

accept all symbols
All symbols in the target library are taken over which are contained in the *CADdy* drawing.

check all symbols
This checks whether symbols of the same name exist in the target folder of the target library. If this is the case, confirm whether the symbol is to be overwritten in the target folder.
New symbols are taken over without a prompt.

only accept new symbols
Only the symbols from the *CADdy* drawing which do not exist in the target library are taken over.

11. Under the heading **accept scale**, activate one of the following options.

The scales available in the *CADdy* drawing file are interpreted and taken over as partial drawing orientated.

This option is necessary for older *CADdy* drawing files, since the scale information for the drawing is saved in the *CADdy* INFO file. Warning! With this setting, it is imperative that the respective *CADdy* INFO file is available.

This option is necessary for older *CADdy* drawing files, if the scale is controlled using an INFO file, but no INFO file exists. Within this option it is necessary to enter the scale required for the conversion of the *CADdy* drawing file.

12. If the PIC file contains text, in which an apostrophe appears, the text that follows this character is not displayed in *CADdy* and not printed.
If this text or text parts however are to be imported, activate the option ***accept hidden text***. They are retained during a work session.

13. Start the *CADdy* drawing import with ***OK***.
Since the current model is erased before importing, determine whether this should be saved.

14. Determine the scale to be used for the drawing to be converted.

Accept scale from partial drawing

Accept scale from *CADdy* Info file

Define scale manual

15. Layer transfer

Names beginning with numbers: Activate this option if, for *CADdy* import, not only the layer description but also the layer number is taken over. The layer numbers are placed in front of the layer description.

Fill out numbers by three places: Activate this option if all layer numbers are to be taken over with three numbers. Layers that have not received a name in *CADdy*, are assigned numbers. If the option

names beginning with numbers is activated, all layers will be allocated starting with numbers.

Generate all layers: If this option is activates, not only occupied layers but all 512 layers are imported from *CADdy*.

16. Determine the unit to be used for the drawing to be converted. Example: drawing in unit mm, architecture drawing in cm; dm or m
17. If the *CADdy* PIC file contains component information, change to the ***Components*** option card and carry out the settings defined in the ***Take over component information from PIC files in BeckerCAD*** chapter.
18. Start the import of *CADdy* drawings using ***OK***.
During the import procedure, this can be cancelled at any time by pressing any key.
19. The drawing boundaries of the PIC drawing and the current *BeckerCAD* drawing, cannot be compared to each other when importing. Open the dialog box ***Settings, Drawing***, deactivate and activate the drawing boundaries one after another. The drawing boundaries of the imported PIC file are then to scale.

Please note:

CADdy drawing files can also be imported using Drag&Drop. Info and layer information are not taken into consideration when doing so. A PIC file imported in this way, is generated in a new window of the current model if you let the file drop within the program window. If you drag the PIC file into an opened drawing window, the PIC file will be imported into this drawing.

Import from CADdy INF and LAY files

The settings saved in a *CADdy* INF file and a *CADdy* LAY file can be loaded by using the ***File, Import, CADdy PIC File*** command.

1. The option card ***CADdy Settings*** includes ***drawing file****:* leave this blank.
2. The entries in the text boxes for the ***layers file*** and the ***info file*** must contain the names of the info file (*.INF) and the layers file (*.LAY) that are to be loaded; an alternative is to click the icon to locate them.

Please note:

Info files can be imported using Drag&Drop.

Since font and dimension sizes in *BeckerCAD* could be related to the paper and are therefore scale independent, these values should be defined in the *CADdy* INF file for the scale of 1:1.

Only the occupied layers of a *CADdy* drawing are taken over into *BeckerCAD*.

Import *CADdy* files: Technical information

When *CADdy* files are being imported the program will take the following objects/structures into consideration:

PIC files

- Drawing sizes
- Partial drawings will be taken over with their names and scales
- Drawings will be loaded with the colours and line widths and line types allocated as attributes.
  During this procedure the line types will be altered so that they approximate those utilised in Windows.
  User-defined line types will not be supported.

Line types defined by the user are recognised and are also available in *BeckerCAD* as user defined line types (menu ***Define, User Defined Line Types***)

- *New* dimensionings from *CADdy* Version 14.0 onwards, are known and dealt with as dimensions. For those set with the "old dimensions" switch in the *CADdy* dimensioning parameters or drawings from older versions, these are taken over as lines and texts.
- Dimensionings edited in *CADdy* or dimensionings of scaled objects, are identified in *BeckerCAD* by an underline.
  Using the menu option ***Settings, Dimensioning***, the underline for all edited dimensions can be deactivated. For individual dimensions you can deactivate the underline using the command ***Dimension, Edit Dimension Properties***.
- Hatchings are taken over as individual lines combined together in a group.
- Clusters will be taken over as groups that will be given CLUSTERnn as their names.
- Standard parts which are not available in *BeckerCAD*, are taken over as line objects.

- Line widths (pen thickness) defined using layers are not taken over.
- Fillings and colour areas are not taken over.
- Walls from architecture are taken over as parallel lines.
- Only *CADdy* layers are taken over. They are inserted on the layer structure under the layer with the description *Standard*, whereby unnamed layers retain their number as the description, all others retain the descriptions entered in the LAY file.

  If the layer *Standard* is renamed, a layer is set up below this layer with the description *Standard*, under which all *CADdy* layers are inserted.

  No individual layers are set up for dimensions. Instead they are linked with the current layer for dimensioning. If the dimensioning in *BeckerCAD* has a dimensioning structure, the partial elements of the dimension are linked with these layers.

INF and LAY files

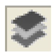
If *CADdy* INF files and, possibly, *CADdy* LAY files are being imported, the following data will be taken over:

- Below the layer with the description Standard, the loaded layers are set up with the original description or with an automatically numbered description. Layers whose description already exists, are not set up.

  If the layer *Standard* is renamed, a layer is set up below this layer with the description *Standard*, under which all *CADdy* layers are inserted.

  Colour and line types of layers are taken over.

Please note:

In *BeckerCAD* the object-related display attributes take precedence over attributes specified for layers.
If you want to ensure that all the objects are displayed using the layer attributes, you will have to select all the objects first. This is done by activating the ***Process, Object Display*** command and then deactivating the setting for *object specific* by clicking .

Since font and dimension sizes in *BeckerCAD* could be related to the paper and are therefore scale independent, these values should be defined in the *CADdy* INF file for the scale of 1:1.

CADdy drawings that are to plotted to a specific scale can be scaled by means of a scale factor, i.e. the drawing dimensions and the sizes of all the elements in these drawings can be enlarged or reduced by this factor. This setting will also effect the line widths allocated to the elements as attributes.

Symbols

If you want to be able to transfer all A-symbols that you have set up in *CADdy* over to *BeckerCAD*, there is the option of loading all the symbols which lie in a directory into one *CADdy* PIC file using a Plus program <SybinPic.vab>.
Finally import the *CADdy* file into *BeckerCAD* to save all symbols in the currently set symbol library and they are then directly available with their descriptions.

Please note:

So that a symbol library remains clear in its overview in *BeckerCAD*, it is advisable to only take over a certain amount of symbols in one step. Therefore, structure your *CADdy* symbols first of all and combine them into specific topics before loading them into the *CADdy* drawing.

Import DXF/DWG files

With the version *BeckerCAD 12* a new DWG/DXF converter has been integrated. This converter can be selected directly from the menu command ***File, Import, DXF/DWG File*** ....

In case the data transfer with the new converter is not satisfactory, you can start the old 32-bit converter via a user defined symbol bar. It is located in the ***General CAD Functionality***, ***2D Import/Export*** folder.

Import with the current converter

In order to load files in DXF or DWG format, select the menu command ***File, Import, DXF/DWG File***.

When executing this command, the DXF/DWG file is imported into the active 2D window of the current model. If necessary, set up a new model or a new drawing within the model if you do not wish to import the DXF/DWG file into the opened active 2D window.

If the DWG-file contains 3D SAT data, a top view is generated during the import and placed as a symbol in the active 2D drawing.
In addition, the SAT bodies can also be imported from the DWG file and imported into the 3D area. To do this, activate the ***Import SAT bodies*** switch in the ***DXF / DWG Import*** dialog. If bodies are found in blocks and are used more than once, the structure is adopted and displayed as groups.

For the case that you have accidentally imported the drawing in an incorrect file or an incorrect drawing, you can undo the step just made using the ***Undo*** command button.

1. In the ***Source file*** text box, determine the drive, folder and name of the file to be imported.

   This can be entered in the text box or be searched for using the  command button.

2. **Unit / Factor**
   If the file to be imported contains information about the unit used during creation, this information is determined automatically. Otherwise, select the unit in which the DXF/DWG file is to be imported from the list or enter a factor in the form of a number. The unit or factor determines the magnification or reduction factor for the imported elements. (For hall plans or floor plans, for example, the unit Meter usually makes sense when importing).

3. **Check and adjust**
   This option is enabled by default and corrects errors in the DWG or DXF file during the import process.
   For files with large amounts of data, this option may delay the import process. Therefore, you can optionally deactivate this option.

4. **Take page size**
   The sheet format is read from the file to be imported and adapted in *BeckerCAD*. However, if you want to keep the current drawing dimensions, deactivate this option.

5. **Move to origin**
   All objects are combined in an imaginary box, which is moved to the coordinate origin.

6. **Adjust view**
   The view is adapted to the imported objects so that they are displayed full window.

7. **2D objects**
   2D objects are imported into the active drawing.

8. **3D top view**
   If the DWG file contains 3D-SAT data, a top view is created during import and placed as a symbol in the active 2D drawing.

9. **3D objects**
   Falls die DWG-Datei Körper im SAT-Format enthält, werden die Körper in das 3D-Fenster importiert.

10. Exclude Tab

Use this tab to exclude objects from the import. The import process can be accelerated by activating the relevant option.

Frozen layers

For example, layers are frozen to improve performance in CAD software and reduce recovery times for complex drawings.

Hidden layers

Objects on these layers are not visible.

Invisible elements

These are individual object types that have been hidden.

11. Start the DXF/DWG file import with *OK*.

Please note:

Symbols are created from dimensions during the import process in *BeckerCAD*. This ensures that the representation of the dimensions is transferred 1:1 from the original drawing to the model.

Blocks are converted into symbols during the import process in *BeckerCAD*. After the import, these are only located in the model.

DXF and DWG files can be imported using Drag&Drop. A file imported in this way is generated in a new window of the current model, if the file is dropped in the program window. If the file is dragged into an open drawing, the DXF/DWG file is imported in the drawing.

Import with the old converter

In order to load files in DXF or DWG format, select the menu command ***File, Import, DXF/DWG File***.

When executing this command, the DXF/DWG file is imported into the active 2D window of the current model. If necessary, set up a new model or a new drawing within the model if you do not wish to import the DXF/DWG file into the opened active 2D window.

If the DWG-file contains 3D SAT data, a top view is generated during the import and placed as a symbol in the active 2D drawing.
In addition, the SAT bodies can also be imported from the DWG file and imported into the 3D area. To do this, activate the ***Import SAT bodies*** switch in the ***DXF / DWG Import*** dialog. If bodies are found in blocks and are used more than once, the structure is adopted and displayed as groups.

For the case that you have accidentally imported the drawing in an incorrect file or an incorrect drawing, you can undo the step just made using the ***Undo*** command button.

1. In the ***source file*** text box, determine the drive, folder and name of the file to be imported.

   This can be entered in the text box or be searched for using the  command button.

2. Objects not to be imported can be deactivated before converting. This can speed up the import significantly.

3. Activate the option ***Ignore extrusions*** if objects should be imported which are not in the x/y plane.

4. **Change drawing boundaries**
   The sheet format in *BeckerCAD* is usually adjusted to the boundaries of the imported drawing. If the current sheet formats should not be changed, please deactivate the option **Change drawing boundaries**.

5. ***Check & Adjust***
   As a default this option is activated and corrects errors during the import of DWG or DXF files.

   In very big files this option can take some time and can therefore be deactivated.

6. **Import SAT bodies**
   Activate this option if the DWG file contains 3D SAT bodies and you want to import them into the 3D window.

7. Activate the ***Settings*** option card and activate the default ***accept dimensions as symbols*** check box, if the display of the dimensions should be identical to the origin. They are to be taken over in the model as symbols and saved in the determined target library.
   Please be aware: If this option is deactivated, the dimensions are regenerated as real, intelligent *BeckerCAD* dimensions. **Especially adjusted formats like processed dimensions can be lost or can be displayed different to the origin.**

8. From the list, next to the selection option ***unit:***, select the unit in which the DXF/DWG file is to be imported or enter a factor. The unit or factor defines the increase or decrease of the imported elements. (for hall plans or groundplans this is for example, usually imported in metre units).

9. Activate the convert ***AutoCAD R12*** *fonts* check box if the fonts which are found in the determined AutoCAD font files are to be converted. Only original AutoCAD V12.0 fonts are supported with the exception of so-called Big Fonts.

10. In the ***ACAD fonts folder*** text box, enter the folder which contains the AutoCAD font files.

    This can be entered in the text box or be searched for using the  command button.

11. Choose the option ***Disassemble AutoCAD block references*** if all the blocks should be disassembled before converting. This can speed up the import significantly.

12. Activate this option, if the file to be imported contains external references which are also available.
    The files, where the external references are pointing to, must be in the original directory or in the same directory as the imported file.

13. Under the heading ***Accept symbol***, select one of the following options:

    only accept symbols in model
    All blocks are taken over in the model as symbols, which are contained in the selected DXF/DWG file. No additional symbol library is set up.

    accept all symbols
    All blocks which are found in the selected DXF/DWG file are taken over as symbols in the model and defined symbol library.

    check all symbols
    This checks whether symbols of the same name exist in the target folder of the target library. If this is the case, confirm whether the symbol is to be overwritten in the target folder. New symbols are taken over without a query.

    Only accept new symbols
    Only symbols for which no symbol of the same name exists in the target library are taken over.

14. In the ***target library*** text box, determine the drive, folder and name of the symbol library in which the blocks contained in the file and, if necessary, dimensioning as symbols are to be archived.

    This can be entered in the text box or be searched for using the  command button.

15. Within the symbol library, determine the folder in which the symbols are to be archived.

This can be entered in the text box or be searched for using the command button.

16. When you have selected the source file, the fonts used in the DWG/DXF files are given. If you wish to change the font allocation change to the ***Font allocation*** option card.

Click the displayed icon alligned to the text to be changed. The ***Font selection*** dialog box if offered. Double click on the font or tag the font and click the ***OK*** command button.

17. Start the DXF/DWG file import with *OK*.

Please note:

DXF and DWG files can be imported using Drag&Drop. A file imported in this way is generated in a new window of the current model, if the file is dropped in the program window. If the file is dragged into an open drawing, the DXF/DWG file is imported in the drawing.

Technical information

The information given immediately below will be taken into account during import procedures from 2D DWG-and 2D DXF files:

Object classes dealt with

ARC	circular arcs
ATTDEF	Attribute definitions
ATTRIB	Attribute text
BLOCK	Symbol
BODY	Top view as symbol, if needed 3D body
CIRCLE	full circles
DIMENSION	Dimensions
ELLIPSE	Ellipses
FACE3D	Face (projection on x-y plane)
HATCH	Group with name "h_Hatchname". All hatching patterns are supported.
IMAGE	BMP, JPG, PCT, PCX, PNG, TGA, TIF
INSERT	Symbol placed in the model
LEADER	Individual geometry element

LINE	lines
LINE3D	Line (projection on x-y plane)
LWPOLYLINE	Lines
MLINE	Group with name "Mline.
MTEXT	Text
MULTILINE	Lines
POINT	Points
POLYLINE	Polylines
PROXY	Imported as symbols
REGION	Top view as symbol, if needed 3D body
SEQEND	Hidden object class
SOLID	Colour face
SOLID3D	Top view as symbol, if needed 3D body
SPLINE	*BeckerCAD* Spline
TEXT	Text
TOLERANCE	Individual geometry elements
TRACE	Lines
VERTEX	Invisible object class

Layer attributes

Name	**Layer name**
Line type	Layer line type see "Line type allocation for DXF/DWG import"
Colour	Layer colour see "Colour allocation for DXF/DWG import"
Display: On	Active: Yes
Display: Off	Active: No
Freeze: On	Active: No Visible: No
Freeze: Off	Active: Yes Visible: Yes
Lock: Yes	Active: No
Lock: No	Active: Yes

Element properties

Line type	Element line type see "Line type allocation for DXF/DWG import"
Colour	Element colour see "Colour allocation for DXF/DWG import"
Weight	Element weight see "Lineweight allocation for DXF/DWG import"

Line type allocation for DWG/DXF import

CADdy line type	ACAD line type
	CONTINUOUS
	DASHED
	DOT
	DASHDOT
	DIVIDE
	HIDDEN
	BORDER
	DOT2
	CENTRE
	PHANTOM

Colour allocation for DWG/DXF import

BeckerCAD converts all colours according to the ACI standard.

The DXF/DWG format describes a spectrum of 255 different colours, numbered from 1 to 255.
In order to allocate a colour to a number, use the “AutoCAD Color Index” (ACI standard). This contains the RGB values of the different colour numbers.

Special characters

Special characters which are contained in text are retained up to the following which have to be converted specially.

%%d = °

%%c = Ø

%%p = ±

%%% = %

%%o = Text overlined

%%u = Text underlined

Important information:

Partial drawings will be converted without recognising the individual scale to get an identical appearance in the DXF/DWG format.

The DXF-/DWG format only supports specific digits for BLOCK and LAYER names. Please use only the following digits:

Characters (A-Z)
Numbers (0-9)
(-)
(_)
($)

Please recognise that *BeckerCAD will support upper and lower case characters*.
The DXF/DWG format does only support upper case characters. Make sure your layers and symbols are named only with upper case characters.

In the DXF/DWG, the format for the length of the layer and symbol names is limited to 31 digits.

Embedded DXF-/DWG-Files

Drawings from DXF or DWG files can be embedded data reduced with the menu command ***Insert, Embed DXF/DWG-File***.

The DXF / DWG file is then embedded into the drawing originally from the source file. Unlike to the import of DXF / DWG files the single elements cannot be edited; but the endpoints of lines can be accessed to create new geometry.

When exporting to DXF / DWG the embedded drawing is joined with the new geometry objects into a „normal“ object oriented drawing, so that importing into *BeckerCAD* or another CAD system creates a complete drawing.

,  Embedded DXF / DWG files can be processed using the command ***Information, Edit Object***:

-  In the line ***name*** the embedded file can be exchanged against a newer or a different DXF / DWG file using the button ***File open***.

- When changing the ***reference point*** the embedded file is moved inside the drawing.
- By editing the ***scale*** factors the size of the embedded drawing is changed.
- The embedded drawing is rotated, if the values in the field ***angle*** are changed.
- The option ***draw frame*** should be activated to display the real limits of the embedded drawing.
- With the option ***best possible accuracy*** it can be defined, if all the original objects should be read from the source file. If the option is deactivated, some of the objects will be substituted by equivalent *BeckerCAD* objects to improve the performance.

Export drawings

The menu command ***File, Export*** can be used to save objects of the current partial drawing as a file in *CADdy* PIC format and the objects of the current drawing in DXF/DWG format.

Export of SVG files

Use the ***File***, ***Export*** menu command to save all or only selected objects of the current drawing in SVG format.

1. If you do not want to export the entire drawing, but only certain objects from the drawing, first select these objects.

2. Select the menu command ***File***, ***Export***, ***SVG file***.

3. In the File name box, specify the drive, folder, and name of the file you want to export.

You can do this by making an entry or by clicking the button  and then perform the file selection.

4. If you have previously selected objects in the drawing and only want to export these objects, activate the ***export selection list*** option.

5. Start the export by clicking the ***OK*** button.

Export CADdy PIC files

In order to save the content of the current partial drawing in *CADdy* PIC format, select the menu command ***File, Export, CADdy PIC File***.

1. In the file selection dialog box, determine the drive, folder and file name for the *CADdy* drawing. These must correspond to DOS conventions.
2. Start the export of the file with *OK*.

Due to the various object structures in both programs, only the "normal" 2D objects can be transferred as *CADdy* drawing objects.

INF and LAY files are not generated when exporting.

Export DXF/DWG files

With the version *BeckerCAD 12* a new DWG/DXF converter has been integrated. This converter can be selected directly from the menu command ***File, Export, DXF/DWG File*** ....

In case the data transfer with the new converter is not satisfactory, you can start the old 32-bit converter via a user defined symbol bar. It is located in the ***General CAD Functionality***, ***2D Import/Export*** folder.

The settings for exporting to DXF and DWG files are identical for both converters.

In order to save the contents of the selected drawing or the previously selected objects as a DXF file or a DWG file you must select the ***File, Export, DXF/DWG File*** command.

1. If the drawing consists of multiple partial drawings with different scales, select the partial drawing with the main scale for the DXF / DWG file first.
2. To export not the complete drawing, but only parts if it, select the objects to be exported first.
3. Select the command ***File***, ***Export***, ***DXF/DWG File***.

   (In the old converter, the 64-bit version is exported via the ***DWG/DXF Service***, a temporarily displayed window, which is closed after handover.)
4. In the ***file name*** text box enter the drive, folder and name of the file to be exported.
   This can be entered in the text box or be searched for using the ***Browse*** command button.

5. In the dialog box, determine the ***output format*** DXF or DWG.
6. In the ***AutoCAD Version*** field, determine in which version the file should saved.
7. Select whether the *dimensions* are to be entered into the target file ***as BLOCK*** or ***as DIMENSION***.
8. The scale of the current partial drawing is valid if multiple partial drawings with different scales are contained in this drawing. Select the partial drawing for the valid scale.
9. If objects have been selected in the drawing before to be exported, activate the option ***Export Selection List***.
10. Start the export of the file with ***OK***.

Technical information (for the old converter only)

The information given immediately below will be taken into account during export procedures into2D DWG-and 2D DXF formats:

Object types dealt with

Arc	ARC
Symbol	BLOCK
Full circle	CIRCLE
Dimension	DIMENSION
Ellipse	ELLIPSE (R13, R14) POLYLINE/VERTEX (R12)
Line	LINE
Polygon	LINE
Text (multi line)	MTEXT (R13, R14) TEXT (R12)
Point	POINT
Spline	POLYLINE(R13, R14) POLYLINE/VERTEX (R12)
Colour surface	SOLID
Spline	SPLINE (R13, R14) POLYLINE/VERTEX (R12)
Text (single line)	TEXT)

Not supported:

Auxiliary infinite lines, line widths, grid data

Layer attributes

Description	Layer name

Line type	Layer line type see "Line type allocation for DXF/DWG export"
Colour	Layer colour see "Colour allocation for DXF/DWG export"
Active: Yes	Lock: Yes
Active: No	Lock: No
Visible: Yes	Display: Yes
Visible: No	Display: Non

Element properties

Line type	**Element line type see "Line type allocation for DXF/DWG export"**
Colour	Element colour see "Colour allocation for DXF/DWG export"

Line type allocation for DWG/DXF export

***BeckerCAD* line type**	**ACAD line type**
———	CONTINUOUS
-------	DASHED
....	DOT
–·–·–	DASHDOT
—··—··	DIVIDE
———-	HIDDEN
--·--·-	BORDER
· · · · · · ·	DOT2
—–—	CENTER
—--—	PHANTOM

Colour allocation for DXF/DWG export

BeckerCAD converts all colours according to the ACI standard.

The DXF/DWG format describes a spectrum of 255 different colours, numbered from 1 to 255.
In order to allocate a colour to a number, use the “AutoCAD Color Index” (ACI standard). This contains the RGB values of the different

colour numbers. *BeckerCAD* uses the colours from the Windows colour palette. As a rule, this contains more than 255 colours.

Therefore a DXF/DWG colour number must be selected which is similar to the *BeckerCAD* colour.

Special characters
DWG/DXF formats permit a maximum length of 31 characters for names used for layers and blocks. In addition, the only characters permitted are A-Z, 0-9, -,_ and no others.
It is important that these restrictions are adhered to for naming the layers and the symbols.

Special characters in text will be retained, with the exception of the following ones that will be converted separately.

_G %%d = °

_D %%c = Ø

_+ %%p = ±

Please note:

Partial drawings are converted without taking the respective scale into account so that an identical representation is formed in DXF/DWG format.

The DXF/DWG format only allows specific characters for BLOCK and LAYER names. Therefore, please only use the following characters for your layer and symbol names:

Characters (A-Z)
Numbers (0-9)
The special character hyphen (-)
The special character underline (_)
Dollar sign ($)

BeckerCAD differentiates between smaller letters and capitals. Since only capitals are managed in DXF/DWG format, this should be taken into account when allocation layer and symbol names.

The names of layers and symbols cannot be longer than 31 characters.
Please take note of this limitation for layer and symbol descriptions.

Export WMF files

In order to save the content of the active partial drawing in Windows Metafile format, select the menu command ***File, Export, WMF File***.

1. In the dialog box, determine the drive, folder and file name.

2. Start the export of the file with *OK*.

Export of EMF files

Windows Enhanced Metafile (EMF) is a further development of the Microsoft® Windows Metafile-Grafic file format (WMF). It enhances the scalable vector graphics with the possibilitiy to use pixel graphics as filling.

To print a drawing in the EMF format, follow these steps:

1. Select the menu command ***Process***, ***Clipboard***, ***Plot to Clipboard***.
2. Select a printer driver, as described under Define Settings for print and define the desired settings.
3. Activate the option ***Output to file***.
4. Click the button  and define the file name and the folder fort he EMF file.
5. Click the button
   Print & Close → The drawing is printed; the dialogue window is closed.
   Print → The drawing is printed.

Copy 2D data in clipboard

If the complete drawing or only the visible sections of the current 2D drawing window are to be copied in the clipboard, in order to accept this display in other application programs, proceed as follows:

1. Select the menu command ***Process, Clipboard***.
2. Select whether the data is to be saved as ***Bitmap*** (pixel) graphics or in the form of (scalable) ***Vector Data***. To use the settings in the ***Print*** dialog window use the option ***Print to Clipboard***.
3. Here you have all the options from the ***Print*** dialog window and any printer driver can be selected to influence the quality of the output.

To print to the clipboard use the menu command ***Process***, ***Clipboard***, ***Print to Clipboard*** or use the key combination ***Ctrl+C***.

With the key command the current printer settings are used.

Another application can then be change to and the data inserted from the clipboard using ***Process, Insert*** or the key combination Ctrl + V.

Please note:

If vector files inserted in documents used in other programs are scaled, this will also affect line widths.
Occasionally you will have to define larger values for line widths in the 2D drawing window than those to be used for printing.

5 3D Commands

5.1 Model units

The menu command ***Settings, Units*** can be used to determine how the coordinates and length values determined by yourself are to be interpreted within the program.

The ACIS core contained in the program manages the solids generated in the 3D area.
Since these work with the **fixed unit mm**, all other units in the 3D area are recalculated into mm.

Example:

If cm is determined as the model unit, the values - length X = 10, length Y = 20 and height = 12 entered in the Status dialog box, result in the dimensions 100 mm, 200 mm, 120 mm.

5.2 Coordinate system

3D commands such as, generating a fixed solid block or rotating such a solid, require entries for the positioning of solids or for determining a point on the rotation axis.

Unlike with a 2D drawing, these points are not on one plane, but in “three-dimensional space“. The definition of points in three-dimensional space require an additional coordinate value.

In order to reach the "correct" results, **orientation in three-dimensional space** is important, i.e. knowledge of the position of the three coordinate axes to one another as well as the direction of rotation axes and the respective direction of rotation.

A **global coordinate system (GCS)** is also available in the 3D-window and - if defined - a **local coordinate system (LCS)**.

Orientation in three-dimensional space

Right hand rules:

1. **Axes directions**
   The axes of the Cartesian coordinate system are vertical to one another. An overview of the directions of the axes can be seen using the right hand:

On your right hand, spread your thumb and index finger so that you are in the plane of the hand face. Hold the middle finger so that it is vertical to the hand face.

If the thumb indicates the direction of the X axis displayed on the screen and the index finger indicates the direction of the Y axis, then the middle finger indicates the direction of the Z axis.

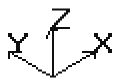
2. **Rotation angle**
   For rotations about an axis, an overview concerning the direction of a rotation can also be seen using the right hand:

   On the right hand, if the thumb indicates the direction of the rotation axis, the fingers which are curved towards the palm of the hand indicate the positive rotation direction. Therefore, negative rotations require input of negative values.

   If the rotation takes place in a plane about a point, rotation takes place about a vertical axis on the face (see Axes directions).

3D global coordinate system (GCS)

BeckerCAD uses a global coordinate system for the 3D-window.
The **origin** (zero point) of the GCS, which the coordinate entries refer to, always lies in the common origin of the three standard workplanes (WP), as long as their position is not modified or erased.
The axes directions of the GCS correspond to those which are displayed by a coordinate axis, if the *Standard XY* WP is active:



Activating the GCS

In the drop-down list box of the main toolbar, activate **Absolute 3D** .

The position of the origin is displayed as a marker whilst generating and transforming objects.

3D local coordinate system (LCS)

In the 3D-window, you can define an LCS as well as the global coordinate system.

This toolbar button can be used to determine or modify the position of the origin for the LCS. This point can be defined as the spatial point. The LCS axes have the same directions as those of the GCS.

Activating the LCS

In the drop-down list box of the main toolbar, activate **Relative 3D**.

If no origin is defined for the LCS, it will be identical to that of the GCS.

Cartesian coordinates

At present, only Cartesian coordinates can be worked with in the 3D-window.

Axes running at right angles to each other form the Cartesian coordinate system.

Each point in three-dimensional space is determined by a coordinate triple, i.e. the coordinate pair, which defines a point in the plane in a 2D drawing, is enhanced by a further value which defines the vertical distance of the spatial point from the point in the plane.

The point (5,3,-4) is therefore 5 units away from the origin on the X axis, 3 units on the Y axis and 4 units in the direction of the negative Z axis.

Specify 2D/3D mode

When working in the 3D view window, you can use two icons in the main toolbar to determine modes for **selecting objects** as well as for **defining points**. Both modes can be active or just one of the two.

In a 2D drawing window, this selection is not available. The 2D mode is always active.

2D mode

2D mode active

If this button is "pressed" (green), the 2D mode is active, which means, if the 3D mode is inactive at the same time:

Selection
If selection is made using the cursor, only 2D objects can be selected.

Snap Menu
The 2D Snap Menu is displayed.

Points can only be snapped on 2D objects in the current workplane.

2D mode inactive

If the button is not pressed down (red), points cannot be snapped on 2D objects.
Likewise 2D objects cannot be selected with the cursor.
When commands that have an effect solely on 2D objects are used, the objects can still be selected.

Please note:

If the 2D and 3D mode are active, the following rules apply for the commands ***Fixed Solid, Swept Solid along Z axis*** and ***Rotational Solid***:
When you are dragging a window, 2D line objects and 2D faces have priority when being determined.
If objects are identified individually with the cursor, 3D faces are determined with priority but 2D objects can also be selected.

3D mode

3D mode active

If this button is "pressed" (green), the 3D mode is active, which means, if the 2D mode is inactive at the same time:

Selection
If the cursor is used for selection, you can select only 3D objects – i.e. solids and axes of solids.

Snap Menu
The 3D Snap Menu is displayed.
Points can be defined anywhere in the current workplane or in 3D space.

3D mode inactive

If this button is not pressed down (red), you cannot select 3D objects with the cursor: nor can you snap points on 3D objects with the cursor.
When commands that have an effect solely on 3D objects, are active the objects can still be selected.

Please note:

If the 2D and 3D mode are active, the following rules apply for the commands ***Fixed Solid, Swept Solid along Z axis*** and ***Rotational Solid***:
When you are dragging a window, 2D line objects and 2D faces have priority when being determined.
If objects are identified individually with the cursor, 3D faces are determined with priority but 2D objects can also be selected.

5.3 Define points

In the 3D-window of *BeckerCAD* when defining points which are required for generating and processing solids, a difference is made between **projection points** and "true" **spatial points**:

At present, the generation and processing of solids using "true" spatial points, which refer to a fixed three-dimensional coordinate system, is combined with a lot of calculation work and requires a high degree of thought concerning three-dimensional space.

For this reason, in *BeckerCAD*, points in three-dimensional space are usually defined using workplanes. Any number of workplanes can be defined at any position and can be permanent or temporary, i.e. points are either determined directly in such a WP or they are defined by projecting points on 3D objects in the WP.

Define spatial points

"True" spatial points cannot be used when solids are being generated. They are accessible only with the following commands:

- Definition and transformation of workplanes
  This also applies to temporary workplanes that are generated when the command used is ***Transform, Reposition***.
- Definition of a 3D reference point for the LCS
- Definition of the position and direction of a light source

In order to be able to determine the required spatial points by entering X, Y and Z coordinate values in the Status dialog box, the 3D mode must be activated in the main toolbar.

The coordinates entered refer to the GCS activated using 

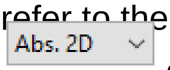
or to the LCS activated using 

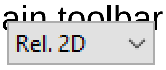
i.e. the origin and direction of the axes are independent of those of the current WP.

Define projection points

In *BeckerCAD*, the generation and transformation of solids in the 3D view window, with exception of the ***Transform, Reposition Solid*** command, on the basis of a permanent or temporarily defined WP.

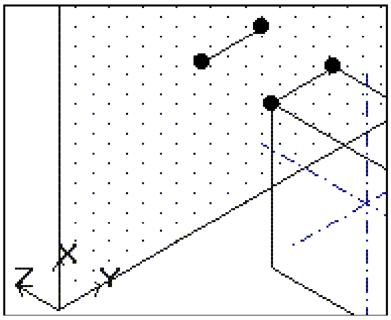
As in 2D drawings, the points required for this, e.g. the corner points of the block base or the endpoints of a line, can be specified directly in the current WP. This can be carried out using the **Cursor**, with or without active **drawing grid** in the WP, or using the **Snap Menu** commands.

If you wish to enter coordinate values which refer to the current WP, on the main toolbar, activate the 2D GCS using 

or the 2D LCS using 

.

The points required can also be defined in the WP by, e.g. snapping the endpoint of a solid edge. The projection of this in the current WP defines the required point, e.g.:



3D Snap Menu

Various Snap Menus are offered in the 3D-window - depending on the mode set in the main toolbar:

- If only the 2D mode is active, the 2D Snap Menu is displayed.
- If only the 3D mode is active, the 3D Snap Menu is displayed.
- If the 2D mode and the 3D mode are active, a Snap Menu made up of both the 2D and 3D Snap Menus is displayed.

If the 3D mode is active, the following commands are additionally available to the 2D Snap Menu commands - these can be used to snap points on 3D objects.

Intersect solid face/axis

Determines the intersection of a face (solid face) and a solid axis.

1. In the Snap Menu, select the command ***Intersect Face/Axis***.
2. Identify the face.
   If solid edges are being displayed, identify two of the edges instead which border the face.
3. Identify the axis.

Intersect line/plane

Determines the intersection of a 2D-/3D-line, an edge or a 3D-axis and a plane (surface, work plane).

1. In the Snap Menu, select the command ***Intersect Line/Plane***.

2. Identify a 2D line, 3D line, edge or 3D axis.
3. Identify a planar surface or the frame of a workplane.

Face/Centre of gravity (G, g)

Determines the centre of gravity of a flat solid face or a 2D object of type 2D face.

1. Identify the 2D object of type 2D face or, with shaded representation of solids, identify the solid face.

   If, for cylinders and cones, the edge is identified which borders the base, the base is also clearly identified as the required face. Otherwise, identify one of the other edges which borders the solid face.

   For cylinders and cones it is sufficient to identify the edge which borders the base or the top surface in order to determine the respective centre of gravity of this face.

Please note:

If this command is used when defining a WP, no further points are required in order to determine the position of the WP:
The WP lies in the same spatial plane as the solid face. The defined centre of gravity is the origin and its Z axis points away from the solid.

Snap Centre of Gravity of Solids (Ctrl+Shift+g)

Snaps the centre of gravity of a single Solid in 3D space.

1. Select the option ***Centre of Gravity*** in the point definition menu.
2. Identify a Solid.

Face/edge/axis

This command is only offered after loading the ***Define Workplane*** command.

If this command is loaded, lines, circles, solid axes, straight and circular solid edges and shaded solid faces can be identified.

Possible procedure:

Line or straight solid edge

1. Select the ***Object*** command and identify a line or a straight solid edge.

2. Define an additional point (Snap Menu) in order to determine the position of the WP.

Circle or circular solid edge
If a circle or a circular solid edge is identified, the position of the WP is completely defined by the circle face.

Solid axis
If a solid axis is identified, the position of the WP is completely defined, since the WP is vertical to the solid axis.
When you are identifying, the nearest endpoint of the solid axis is the origin of the WP.

Solid face
If a solid face is identified, the position of the WP is completely defined since the WP is vertical to the solid face.
The point selected on the solid surface when identifying is the origin of the WP.

3D origin

The 3D origin is only active as the snap point if absolute 3D or relative 3D are selected as coordinate references. Using workplane independent commands such as, Define WP or Define 3D Axis, the 3D origin can be used to snap directly in 3D space the origin that has been defined as the absolute or relative origin. For all other commands, the respective origin is projected onto the active workplane.

5.4 Use workplanes

Workplanes: basic principles

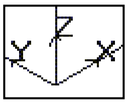
Points in three-dimensional space are usually determined by entering X, Y and Z coordinates, which give the distance from the origin (zero point) on all three axes of the coordinate system.

In most cases, when working with *BeckerCAD* to create solids, **workplanes (WP)** are used for the definition of points in three-dimensional space:

A WP is comparable to a 2D drawing face, which has its origin at any position in three-dimensional space and has any spatial position.

The origin of the currently selected WP lies in the point, which is identified by a coordinate axis.

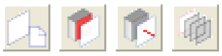
The direction of the axes is made more clearer with text, e.g.:



When using WPs, spatial points are defined by entering the X and Y coordinates in the selected WP.
As is the case for a 2D drawing, each WP has measurements which are represented by a grey frame on the screen.

Once the program is started, three **standard workplanes** are always defined, whose common origin is the origin of the GCS:

X/Y plane with the description *Standard XY*
Y/Z plane with the description *Standard YZ*
X/Z plane with the description *Standard XZ*

In addition to these, any number of WPs can be defined whilst working in *BeckerCAD*.
 For this, the commands ***Define Workplane,*** **Define WP using face/centre of gravity,** ***Define WP using face/edge/axis*** and ***Define Workplane/Workplane Set*** are available which can be loaded via the icons displayed here which can be found in the 3D toolbar.

WPs can be defined as permanent, or - with the commands for generating solids - can be used temporarily.

The definition of points whilst generating or transforming solids always takes place in the **current workplane** or they are projected in this.

WPs which are no longer required for the definition of points, can be erased. However, the three standard WPs should be retained for orientation purposes. Lost standard WP´s can be recreated as described here.
All WP data is saved in templates and models.

Individual Display for Workplanes

A filling colour and a transparency can be defined for planes. This can be defined individually for ***active workplanes***, ***inactive workplanes*** and for the new ***cutting planes***.

The appropriate settings can be made in ***Settings***, ***Options*** in the tab ***Settings Graphic Windows***. The settings are stored when leaving the program.

Workplane sets

In some cases (e.g. when using the commands *Fixed Solid, Swept Solid, Transitional Solid, Profile Solid*) it is necessary to generate a number of WPs which have a defined position to each other and which can be manipulated together.
This is possible when using workplane sets.

Linking individual WPs to a WP set is carried out using their description: All WPs, whose description is made up of a basic name (which can also be empty) and a continuous number, form a WP set, e.g:
WP_XY1, WP_XY2, WP_XY10 or 112, 120, 150.

In order to define WP sets, individual WPs only need to be allocated the respective description. This can be carried out directly when generating WPs, but also by renaming existing WPs.

When using the command ***Define Workplane/Workplane Set***, a number of parallel or radial running WPs can be generated. After entering the basic name, a continual number is added to it and is therefore directly defined a WP set.

Workplanes and 2D objects

If 2D objects are generated within the 3D view window, this always takes place in the current WP. They are connected to this WP and are transformed if the workplane is transformed and are also erased if the WP is erased.

Within the 3D view window, 2D objects are used exclusively as design aids or as bordering objects for the generation of swept solids or rotational solids.

When working on 2D objects, the current WP is - as the active partial drawing in the drawing - the **active 2D section**.
Using the menu command ***Process, Erase Active 2D Section***, all 2D objects in the current WP can be erased without erasing the WP itself.

2D objects can be embedded in workplanes as well as in partial drawings.

Workplanes and solids

When generating solids, workplanes are basically used as planes, on which solids are designed.
Once these are generated, solids become independent of the WP which was used as the basis of their generation.

Define workplane

The command for the definition of a WP can be activated using this icon in the 3D toolbar.

The commands to create workplanes can be found in the 3D window in the 3D icon bar on the left side. They can be selected using the button ***Define WP´s***.

For the definition of permanent and temporary workplanes, one of the following procedures can be used.

If the WP is not in the correct position or is not the right size after being defined, it can be transformed directly or the dimensions can be modified using the commands offered in the 3D toolbar.

All WPs defined as permanent are saved in a template along with their Features (description, position, dimensions), i.e. they are available for each model which is generated on the basis of this template.

Define workplane using 3 points

The command for the definition of a workplane can be activated using this icon.

The commands to create workplanes can be found in the 3D window in the 3D icon bar on the left side. They can be selected using the button ***Define WP´s***.

If a WP is to be specified using 3 points, use the following procedure:

1. Specify the first point in the new WP.
This defines the position of the origin.

2. Specify the second point in the WP.
The connection from the first to the second point determines the direction of the X axis.

3. Specify the third point, which then defines the complete position of the WP.

To get the dialog window ***Edit Workplane*** at the end of the command press the ***Ctrl*** key while starting the command. In this dialog window a name can be specified for the workplane as well as another colour and another line type for the frames.

If the WP does not have the correct position or size after defining, this can be transformed directly using the commands in the 3D toolbar or the dimensions can be modified.

Please note:

If the 3D mode is activated using the  toolbar button in the main toolbar, the points can be defined as spatial points, e.g. enter all three coordinate values in the Status dialog box.
The coordinates refer to the GCS or the LCS - depending on your setting.

The name and the dimensions of the workplane can be modified using the command ***Edit Workplane Properties***, which can be loaded using the icon .

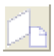
Define workplane using a solid face

Define WP using face/centre of gravity

The commands to create workplanes can be found in the 3D window in the 3D icon bar on the left side. They can be selected using the button  ***Define WP´s***.

1.  Click on the displayed icon in the 3D toolbar.
2. Identify a planar solid face of an existing solid.

Define WP using face/centre of gravity (Snap Menu)

1.  Click on the displayed icon in the 3D toolbar.
2. In the Snap Menu, select **Face/Centre of Gravity**.
3. Identify a planar solid face of an existing solid.

By default the size of the frame is related to the size of the identified face multiplied with a factor. The origin of the workplane is in the point of gravity of the identified face.

To get the dialog window ***Edit Workplane*** at the end of the command press the ***Ctrl*** key while starting the command. In this dialog window a name can be specified for the workplane as well as another colour and another line type for the frames.

Origin in pick point

1.  Click on the displayed icon in the 3D toolbar.
2. Identify a solid face of an existing solid.

To get the dialog window ***Edit Workplane*** at the end of the command press the ***Ctrl*** key while starting the command. In this dialog window a name can be specified for the workplane as well as another colour and another line type for the frames..

Origin in pick point (Snap Menu)

1. Click on the displayed icon in the 3D toolbar.
2. In the Snap Menu, select **Face/Edge/Axis**.
3. Identify a solid face of an existing solid.

To get the dialog window ***Edit Workplane*** at the end of the command press the ***Ctrl*** key while starting the command. In this dialog window a name can be specified for the workplane as well as another colour and another line type for the frames.

Please note:

Is an edge identified in this command to define the face, the selection is not completed. Choose another edge to clearly identify the face. If the identified edge belongs to the top face of a cylinder or a cone, the WP is positioned with the origin in the centre of the face. If the WP does not full fill the requirements according to position or size, this can be changed with the commands in the 3D toolbar.

The name and the dimensions of the workplane can be modified using the command ***Edit Workplane Properties***, which can be loaded using the icon .

Define workplane using a solid axis

The commands to create workplanes can be found in the 3D window in the 3D icon bar on the left side. They can be selected using the button ***Define WP´s***.

1. Click on the displayed icon in the 3D toolbar.
2. Identify a solid axis.
   The endpoint of the solid axis which is nearest to the identified position becomes the origin.

To get the dialog window ***Edit Workplane*** at the end of the command press the ***Ctrl*** key while starting the command. In this dialog window a name can be specified for the workplane as well as another colour and another line type for the frames.

or:

1. Click on the displayed icon in the 3D toolbar.
2. In the Snap Menu, select the ***Face/Edge/Axis*** command. Then identify a solid axis.

The endpoint of the solid axis which is nearest to the identified position becomes the origin.

If the WP does not full fill the requirements according to position or size, this can be changed with the commands in the 3D toolbar. To get the dialog window ***Edit Workplane*** at the end of the command press the ***Ctrl*** key while starting the command. In this dialog window a name can be specified for the workplane as well as another colour and another line type for the frames.

Please note:

The ***Face/Edge/Axis*** command can be used to define a WP by means of solid edges, lines and circles.

The name and the dimensions of the workplane can be modified using the command ***Edit Workplane Properties*** which can be loaded using the icon .

Define workplane using an existing workplane

The command for the definition of a workplane can be activated using this icon in the 3D toolbar. The commands to create workplanes can be found in the 3D window in the 3D icon bar on the left side. They can be selected using the button ***Define WP´s***.

If a WP is to be defined, which corresponds to an existing WP as far as the origin and axis direction is concerned (therefore is a "copy" of this WP), use the following procedure:

1. In the Snap Menu, select the ***Face/Axis/Edge*** command. Then identify the frame line of the active WP.

If the WP does not full fill the requirements according to position or size, this can be changed with the commands in the 3D toolbar. To get the dialog window ***Edit Workplane*** at the end of the command press the ***Ctrl*** key while starting the command. In this dialog window a name can be specified for the workplane as well as another colour and another line type for the frames.

Please note:

The name and the dimensions of the workplane can be modified using the command ***Edit Workplane Properties***, which can be loaded using the icon .

Workplane in Screen Plane

Using the command ***WP in Screen Plane*** a new workplane is created parallel to the screen plane. The current view in the 3D window defines the orientation of the new WP. Very often this is used to create texts in the 3D window, which should be parallel to the view in this window.

To define a WP orientated as the screen plane, please follow these steps:

1. Define the desired view in the 3D window using the commands to rotate and zoom the view.

2. Click the displayed icon in the 3D toolbar. (in the flyout ***Define WP***)

The commands to create WP´s are located in the 3D toolbar on the very left position. They can be selected using the button ***Define WP***.

3. Define the origin with a point definition.

Please note:

The name and the size of the WP can be edited later on using the command ***Edit WP Properties*** in the 3D toolbar.

Restore Standard Workplanes

The three standard workplanes allows a good orientation in the GKS and in the LKS and should therefore not be deleted. If they were deleted, maybe by accident, the can be reinstalled using the command ***Restore Standard WP´s***.

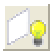
In the 3D-top toolbar on the outer left select the icon bar *Define WP´s* and in this icon bar the command ***Restore Standard WP´s***.

Activate workplane

Solids are always generated with reference to the current WP.
In the same way, 2D objects can only be generated in the current WP in the 3D-window.

In order to select a defined workplane, one of the following procedures can be used:

1. In the 3D toolbar, open the  drop-down list box by clicking on the arrow.
2. From the list, select the required WP by clicking on the respective name.

1. In the 3D toolbar, click on the  icon.
2. Identify a frame line or a 2D object of the WP required.

When saving a model or template, the program "recognises" the current WP, i.e. it will become the current WP again if the model or template is reopened.

Select workplane

For the Transform, Process and Erase commands concerning workplanes, individual as well as a number of workplanes can be selected. The following options are available.

Please note that for the editing of workplanes only the special commands in the upper front of the 3D window are available:

- **Select individual WP**
  Identify a frame line of the required WP.
- **Select a number of WPs using a window**
  Position the cursor on an endpoint of an imaginary rectangle. Press the left mouse button and keep it pressed whilst you drag the window with the cursor. The window definition is finished once the left mouse button is released.
  All WPs whose frame lines project into the window, are accepted in the selection.
- **Select a number of WPs by collecting**
  Keep the *Ctrl key* pressed and identify the frame lines of the required WPs.
  In order to execute the previously selected command with the selected WPs, confirm using the left mouse button. Otherwise abort using the right mouse button.
- **Select all WPs of a WP set**
  By keeping the *Shift key* pressed, identify the frame line of one of the WPs that belongs to a WP set.

Modify workplane properties

In order to modify the properties (dimensions, colour, line type) of workplanes, proceed as follows:

1. Load the command ***Edit Workplane Properties*** using the 3D icon bar.
2. Accept the required WP(s) into the selection.
   If the description is to be modified, the selection should only contain one WP.
3. A dialog box is then displayed in which the *colour* and *line type* for all selected WPs can be specified for the representation of the frame as well as the ***frame size X*** and ***Y***.
   If the coordinate display and input is not to refer to one of the corners, enter the distances to the origin in the ***dist. origin X*** and ***Y*** text boxes.
   The position of the displayed coordinate cross corresponds to the position of the origin.

   If only one WP is selected, a new description can be entered in the first text box.

Please note:

Please keep in mind, that workplanes created with a link to a face will change accordingly to the face.

Transform workplanes

Since WPs often do not have the correct position and axes orientation during definition, use one of the following procedures in order to modify the WP to meet your requirements.

The commands to transform workplanes can be found in the 3D window in the 3D icon bar. They can be selected using the button ***Transform WP´s.***

Please note:

In a WP, any existing 2D objects are also moved.

The three standard WPs should not be transformed since the orientation in the GCS and LCS would become more difficult.

Move workplane

The command can be loaded using this icon in the 3D toolbar.

The commands to transform workplanes can be found in the 3D window in the 3D icon bar. They can be selected using the button ***Transform WP´s.***

If WPs are to be moved within "themselves", proceed as follows:

1. Select the WP(s) which is to be moved.
If a number of WPs are selected, the move is defined in the WP which is first in the selection.

2. Define the starting point of the move vector.

3. Specify the move dynamically or by defining an endpoint.

Please note:

In a WP, existing 2D objects are also moved.

The three standard WPs should not be transformed since the orientation in the GCS and LCS would become more difficult.

Rotate workplane about point

 The command can be loaded using this icon in the 3D toolbar. The commands to transform workplanes can be found in the 3D window in the 3D icon bar. They can be selected using the button ***Transform WP´s.***

If WPs are to be rotated in "themselves", use the following procedure:

1. Select the WP(s) which is to be rotated.
If a number of WPs are selected, the rotation is defined in the WP which is first in the selection.

2. Define the midpoint of the rotation.

3. Specify a point as the reference point for the rotation.

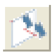
4. Specify the rotation angle dynamically, by defining the target position of the reference point or by entering the value in the Status dialog box.

Please note:

In a WP, existing 2D objects are also moved.

The three standard WPs should not be transformed since the orientation in the GCS and LCS would become more difficult.

Move workplane along Z axis

 The command can be loaded using this icon in the 3D toolbar. The commands to transform workplanes can be found in the 3D window in

the 3D icon bar. They can be selected using the button ***Transform WP´s.***

If WPs are to be moved parallel to the Z axis (modify the height), use the following procedure:

1. Select the WP(s) which is to be moved.
If a number of WPs are selected, the move is defined along the Z-axis of the WP which is first in the selection.

2. Specify the position of the WP dynamically with the cursor or by entering the value for the distance to the original position in the ***Height 1*** text box of the Status dialog box.
Positive values result in positioning in the direction of the positive Z axis, negative values in the opposite direction.

Invert workplane

The command can be loaded using this icon in the 3D toolbar. The commands to transform workplanes can be found in the 3D window in the 3D icon bar. They can be selected using the button ***Transform WP´s.***

If WPs are to be inverted (the direction of the Z-axis is to be reversed), select this command.

Rotate workplane about axis

The command can be loaded using this icon in the 3D toolbar. The commands to transform workplanes can be found in the 3D window in the 3D icon bar. They can be selected using the button ***Transform WP´s.***

If WPs are to be rotated about any axis in three-dimensional space, use the following procedure:

1. Select the WP(s) which are to be rotated.

2. Define the first point of the rotation axis.

3. Define the second point of the rotation axis.

4. Specify a point as the reference point for the rotation.

5. Specify the rotation angle dynamically or enter the value in the Status dialog box.

Please note:

If the 3D mode is activated using the toolbar button in the main toolbar, when defining the points on the rotation axis, all three coordinate values can be entered in the Status dialog box. The coordinates refer to the GCS or LCS - depending on the setting.

In a WP, existing 2D objects are also moved.

The three standard WPs should not be transformed since the orientation in the GCS and LCS would become more difficult.

Erase workplane

The command can be loaded using this icon in the 3D toolbar.

This command can be used to erase one or more WPs.

Identify the WPs which are to be erased.

Please note:

If 2D objects exist on these WPs, they will be deleted along with the WPs.

If only one WP exists, this cannot be erased.

The three standard WPs should not be erased since the orientation in the GCS and LCS would become more difficult.

Use Workplanes as Cutting Planes

Work planes can be changed to clip planes. If a clip planes cuts solids, they are displayed in a section view.

The display of the sections can be set with the option ***clipped colour properties*** in the dialogue window ***3D Display*** (menu ***Settings***, ***3D Display*** in the tab ***Solid Display***)

In addition three standard cutting planes can be called each of them in the centre of all solids with different directions. A maximum of 6 cutting planes can be active at the same time, additional cutting planes will be displayed as non active cutting planes.

Cutting planes can be modified and transformed with the same commands as the workplanes, e.g. move along Z direction to “walk” through the model with the command: ***Move WP along Z*** or switch the cutting direction with the command : ***Invert WP***.

A standard cutting plane can be activated as follows:

1. Open the ***Model Explorer*** using the menu command ***View*** and select the tab ***Model Explorer***.

2. Start the context menu and select from the option ***Cutting Plane*** the standard cutting plane.

To switch an existing workplane to a cutting plane do the following:

1. Open the ***Model Explorer*** using the menu command ***View*** and select the tab ***Model Explorer***.

2. Place the cursor on the workplane and open the context menu. Select the option ***Cutting Plane***, ***Add and Activate***.***(Shift+Q)***.

To deactivate a cutting plane, so that the solid is not displayed in a sectioned manner any more, open the context menu on the cutting plane and select the option ***Cutting Plane***, ***Activate/Deactivate***.

To switch from a cutting plane to a workplane the command ***Cutting Plane***, ***Remove*** is used.

Icons in the Model Explorer:

 Active Cutting Plane

 Inactive Cutting Plane

5.5 3D Selection mode for Solids

3D commands are supported by the ***3D Selection Mode***. The 3D selection mode allows the user quick access to solids to be processed during construction. The 3D selection mode can be divided into the following points:

Highlighting Points, edges and Faces

The 3D selection mode supports the 3D commands by highlighting vertices of faces, edges of solids, 3D axes and faces when the cursor is moved over these. This visually illustrates what is being identified in the command.

Hide Solids

You may hide a solid using the command ***Hide Solids*** . You may hide the rest with the help of the 3D selection mode or with the command  ***Start Hide Others Action***. You may show the hidden solids once again one by one. An alternative is to hide all of the currently visible solids in the graphic window with the press of a button and at the same time switch the solid that you would like to process to transparent.

Switch Solid to Transparent

You can switch a solid to transparent in the 3D selection mode to identify concealed vertices, edges or faces. These are highlighted when you move the cursor over them, making identification clearer.

General Settings for the ***3D Selection Mode*** can be carried out via the menu ***Settings, 3D Selection Mode***.

Settings for the 3D Selection Mode

With the help of the ***Selection Mode***, faces, edges and vertices of faces are highlighted in various 3D commands to make identification in the command clearer. In addition to this, you may temporarily hide solids or switch a solid to transparent to identify concealed points, lines or faces as well.

The settings for the selection mode are carried out via the menu ***Settings, 3D Selection Mode***. The settings you make here are automatically saved when you exit the program.

Highlight Colour: In commands that support the selection mode, points, lines and faces are highlighted in the colour selected here when the cursor is moved over them.

You can activate the commands in the selection mode which hide or switch solids to transparent in one of the 3 following ways:

activate hover mode: If this option is active, then the selection mode is generally active in the supported 3D commands.
This setting can be combined with the following:

only with CTRL key: If you activate this option, the selection mode is only active when you hold down the Ctrl key and do not move the mouse during the ***activation time*** (in ms).

activation time: After the amount of time defined here, the hover mode engages if the mouse is not moved in the previously activated 3D command.

support keyboard: If you have activated this option, then you can control the selection mode in the supported 3D commands using the keys.

support mouse: Activate this option if want to control the selection mode in the supported 3D commands in a combination with the mouse and the keyboard.

Please be aware: The collection command while holding down the Ctrl key is not available in some of the 3D commands if you have activated the selection mode with support mouse.

time to refresh: An edge of a solid is highlighted when you move the mouse over it and you may move over the faces bordering the edge. A new edge is defined as a reference edge when the mouse is moved over it only after the time to refresh defined here. You can then move over the bordering faces from the reference edge.

Max. activation distance

This defines the maximum distance betwenn object and cursor from which it will be highlighted.

Activation speed

This defines the time in seconds, before the object will be highlighted with the cursor on top.

edge length of snap box: You determine the size of the snap box that is used for highlighting with this value (value in pixels). However, the snap box displayed will not be adjusted to this value. Small values allow for the selection of small details: Selecting a narrow face between two edges for example. Small values, however, also lead to sensitive behaviour, i.e. the highlighted geometry switches quickly.
The standard value for this setting is between 3-6 pixels.

line width: Define the value here in pixels for the line width in which the edges should be highlighted.

Apply 3D Selection Mode

With the help of the selection mode, faces, edges, 3D axes and vertices of faces are highlighted in most of the 3D commands to make identification clearer.
In addition to this, you may temporarily hide solids or switch a respective solid to transparent in order to be able to identify concealed points, edges of solids or faces in the command as well. You can activate the 3D selection mode commands which hide the visibility of concealing solids or switch solids to transparent using the following options:

<u>**Integrated into the 3D commands**</u>

Hover Mode

With **key commands**

With **support mouse and keyboard**

<u>**Separate 3D Command**</u>

/ **Start Hide Action**

/ **Start Hide Others Action**

You can carry out settings for the 3D selection mode and activate or deactivate the individual activation options integrated in the 3D commands via the menu command ***Settings, 3D Selection Mode***.

<u>**General Procedure:**</u>

After you have started the 3D command, move the mouse into the graphic area. If faces, edges or vertices of individual solids are highlighted when the mouse is moved over them, then the commands for hiding solids or switching to transparent are supported in the 3D selection mode.

- ***Switch Solid to Transparent***
  Move the mouse over the solid that you would like switch to transparent.
  - Do not move the mouse when hover mode is activated (if necessary, hold the Ctrl key – depending on the selected settings).
  - Press the ***d*** key when support keyboard is activated.
  - Press the left mouse button while holding the ***Ctrl*** key when support mouse is activated.

  Move the mouse over an edge or vertex. If you move over a bordering face it is highlighted. You can come to the next faces bordering this edge or the vertex from another edge or from a vertex of this face.

- ***Switch Solid to Transparent and Hide all Others***
  Move the mouse over the solid to be processed.
  - If the hover mode is activated press the ***Shift key*** and do not move the mouse (if necessary, hold the ***Ctrl key*** – depending on the general settings).
  - Press the ***f*** key when the support keyboard is activated.
    Please note:
    By using the key combination ***Shift+f*** (= F) the transparency of the solid can be toggled.
  - Press the left mouse button while holding the ***Ctrl and Shift key*** when support mouse is activated.

Move the mouse over an edge or vertex. If you move over a bordering face it is highlighted. You can come to the next faces bordering this edge or the vertex from another edge or from a vertex of this face.

- ***Hide all Others***
  Move the mouse on top of a solid.
  - If keyboard support is activated enter the key ***H*** *(= Shift+h)*.
  -  Start the ***Start Hide Others Action*** and identify the solid, which should remain visible.
- ***Hide Solids Individually***
  Move the mouse over the solid that you would like switch to transparent.
  - Press the ***h*** key when the support keyboard is activated.
  - Press the centre mouse button while holding the ***Ctrl key*** when support mouse is activated.
  -  You can identify the solid with the separate command ***Start Hide Action***.

    Repeat the procedure for any number of additional solids.

    If solids have been hidden by one of the options, this is made apparent by a modified display of the command button ***Start Hide Action***:
     No solids have been hidden.

     Solids have been hidden.

    Pressing the command button shows all of the previously hidden solids once again.
    Hold the ***Shift key*** and press the button if you would like to retrospectively hide further solids.
- ***Show Solids Individually***
  This option is only available if solids were previously hidden individually.
  - Press the ***u*** key when the support keyboard is activated.
  - Press the right mouse button while holding the ***Ctrl key*** when support mouse is activated.

    Repeat the procedure until all solids are visible again.
- ***Show All Solids***
  This option is only available if solids were previously hidden.

- Press the ***Start Hide Action*** command.
- Press the ***a*** key when the support keyboard is activated.
- Press the right mouse button while holding the ***Ctrl*** and ***Shift key*** when support mouse is activated.

Please be aware: The selection mode cannot be used per snap point (point definition menu) during the assignment of point constraints.
Only faces, lines, and vertices can be identified in transparent mode. They are highlighted when the cursor is moved over them.

3D Selection Mode Activation Options

The 3D selection mode is directly integrated into the 3D commands. In addition to this, it may be started separately via the commands ***Start Hide Action*** and ***Start Hide Others Action***. The following options for activating the 3D selection mode are available to you:

Hover Mode: If you have activated this option in the settings, then the selection mode is generally active in the supported 3D commands. If the pointer does not move from a solid during the selected ***activation time***, the selection mode is then active and the solid becomes transparent. Holding the ***Shift key*** in addition to this hides all of the other solids currently visible in the graphic area at the same time.
If the additional option ***only with CTRL key*** is activated under ***Settings, 3D Selection Mode***, the selection mode will then be active only after pressing the ***Ctrl key*** and after the ***activation time*** has passed.

support keyboard: If you have activated this option in the settings, then you may activate the 3D selection mode in the supported 3D commands with the following keys:

h	➔	The solid the cursor is pointing to is hidden temporarily.
H	➔	The solid the cursor is pointing remains visible. All other bodies in the visible graphics area are hidden.
u	➔	The solid hidden last is visible again.
a	➔	All previously hidden solids are visible again.

d	➔	The solid the cursor is pointing to becomes transparent. Previously concealed points, lines and faces may be identified.
f	➔	The solid the cursor is pointing to becomes transparent. Previously concealed points, lines and faces may be identified. All other solids positioned in the visible graphic area are hidden.
		With the key combination **F** (=Shift+f) the transparency of the solid can be toggled.
y	➔	Exit transparent mode.

support mouse: If you have activated this option in the settings, then you can control the selection mode in the supported 3D commands in a combination with the mouse and the keyboard.

Ctrl + left MB*: The solid the cursor is pointing to becomes transparent. Previously concealed points, lines and faces may be identified.

Ctrl + Shift + left MB*: The solid the cursor is pointing to becomes transparent. Previously concealed points, lines and faces may be identified. All other solids positioned in the visible graphic area are hidden.

Ctrl + middle MB*: The solid the cursor is pointing to is hidden.

Ctrl + right MB*: The solid hidden last is visible again.

Ctrl + Shift + right MB*: All previously hidden solids are visible again.

right MB*: Exit transparent mode or end command.

(MB* = mouse button)

Please be aware: The collection command while holding the Ctrl key is not available in some of the 3D commands if you have activated the selection mode with support mouse.

Start Hide Action

 The ***Start Hide Action*** command is started via the button shown above. It is part of the 3D selection mode and serves to temporarily hide solids in the 3D area.

After you have started the command, identify the solids that you would like to hide successively. The status *invisible* will remain valid even when leaving the command but it is not stored with the model or in the UNDO list.

If solids have been hidden this way or by one of the other options described above in the 3D selection mode, this is made apparent by a modified display of the command button ***Start Hide Action***:

 No solids are hidden.

 Solids have been hidden.

Pressing the command button shows all of the previously hidden solids once again.
Hold the ***Shift key*** and press the button if you would like to retrospectively hide further solids.

Please note:

Alternatively solids can be faded out by using the **h** key. More key commands for the 3D Selection Mode.
The options in 3D selection mode, including the command ***Start Hide Action***, serve to temporarily hide interfering solids in 3D commands. Use the ***Groups*** options in the ***Model Explorer*** to switch solids to invisible; also permanently, even after a model has been saved and closed.

Start Hide Others Action

 To get access to the command ***Start Hide Others Action*** open the fly out menu by clicking the arrow beside the command ***Start Hide Action***. If the command ***Start Hide Action*** was used last, it is offered directly.

,  The solid identified in the command ***Start Hide Others Action*** remains visible, all other solids are faded out. By clicking the same icon again all solids are faded in again. The status *invisible* will remain valid even when leaving the command but it is not stored with the model or in the UNDO list.
By clicking the icon on the right side all solids are faded in again.

If solids are faded out using one of the described commands, the icon to ***Start Hide Others Action*** is displayed differently:

 No solids are faded out.

 Solids are faded out.

By clicking the lower icon all solids which were faded out can be faded in again.

Please note:

Alternatively solids can be faded out in the active 3D Selection Mode with the key combination **H (**= Shift+h). More key commands for the 3D Selection Mode.
Opposite to the key command **f** the solid is not displayed in a transparent mode when using the key combination **H** (= Shift+h).

5.6 Generate and process fixed solid

 The commands for generating fixed solids can be loaded after clicking on this icon.

Fixed Solids are those solids which can be generated using the ***Fixed Solid*** CAD Menu commands. All fixed solid dimensions are determined when they are generated.

Primitive solids are fixed solids which are generated directly, e.g. blocks or cylinders.

Swept solids are fixed solids which consist of a closed two dimensional area or a flat partial area of a solid, by moving in the direction of the Z axis, along a path of 2D line objects or along the edges of other solids.

Rotational solids are fixed solids which consist of a closed two dimensional area or a flat face of a solid, by complete or partial rotating about an axis.

Transitional solids are fixed solids, which exist in 3D space by the connection of two even faces, These faces can be 2D objects of type face or even solid faces.

When creating some of primitives and swept solids a ***wall thickness*** can be entered in the status dialog window to create a hollow solid, where the base and top face is opened.

When **generating** primitive solids and free-form solids, positioning always takes place with reference to the currently selected WP or to a temporary WP defined at the beginning of generation. This means that the base or the equator plane of such a solid lies either in the current or temporarily defined WP.
Swept solids and rotational solids are generated in reference to the WP in which the cross-sectional face or the sweep path can be found.
After generation, solids are independent of all WPs.

The 3D solids and their axes can be integrated into the structure of your model and drawn in the way described immediately below:

3D workplane, 3D group

The objects that are generated are saved in the **3DGroupStructure**. If you have used ***the Model Explorer*** to create **3D groups** in this structure and have specified one of them as your current 3D group, the objects will be saved in this group.
You can change the group to which objects belong at a later stage by using ***the Model Explorer***.

If a SAT file is imported, a new group is created with the same name as the SAT file, in which the solid is saved. If the SAT file contains a number of solids, a separate group is set up for each solid with the same name and consecutive number; these are structured under the main group.

Colour, material, visible, access

Just as for 2D objects which have been generated using the ***Draw 2D*** menu, solids are linked with the current drawing layer. The settings ***active*** and ***visible*** are taken over from this layer.
By deactivating these settings, solids can also be switched to invisible or inactive.
Sometimes 2D objects linked by the identical layer are to be either inactive or hidden and so you should, as far as possible not use the same layers for solids and for 2D objects.

The solid display as well as the layer can be modified using the menu command ***Process***, ***Object Display***. For this, start the command and identify the solid or solids whose display you wish to modify or which you wish to move to a different layer.

If solid axes are also to be invisible, this can be specified using the menu command ***Settings, 3D/2D Commands*** on the ***2D/3D axes*** option card.

The **processing** or modelling of fixed solids is carried out with the volume operations unite, subtract, difference or with the commands blend, chamfer or generating a hollow solid or using the commands from the menu ***Local Operations***. Since the solids to be processed must be identified, the ***3D-Mode*** must be active for this.

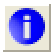
In order to obtain a better orientation in three-dimensional space when generating and processing solids and in order to be able to define points better and identify objects more precisely, a perspective **view** should be used. Additional **3D view windows** can be opened in which the objects can be displayed from different views.

The **display** of each solid depends on the settings which were determined before generation using the ***Settings, 3D Display*** menu

command on the ***Solid display*** option card.
The display of individual or a number of solids can be modified using the ***Process, Object Display*** menu command. This also applies to the accuracy of the object display.

The menu command ***Insert, Model Views*** can be used to transfer individual or a number of **2D views** from solids into the 2D drawing window and form **sectional views** from this.

Please note:

> When creating solids from 2D faces, you may end up with inconsistent solids, depending on the shape and position of the 2D faces and the valid parameters, and when you try to work on them any further you may get a model with errors in it.
>
> It will take a considerable amount of time to check the solids and so this will not be done automatically.
>   For this reason with solids that that are not displayed "normally" you should use ***Information, Check Solid Data***, before you use these solids in any other model.

<u>Referenced Fixed Solids</u>

Under certain circumstances the ACIS (SAT) part of identical fixed solids are stored only once in the memory. This development is useful when designing big 3D assemblies to reduce the necessary memory.

Precondition for referenced fixed Solids is the usage of identical parts or copies of fixed solids. A geometrical search for the identity of solids, e.g. if the same solid is imported as SAT file several times, is not done.

In every process, except of transformations or the change of display properties, the reference is disconnected. In this cases the ACIS (SAT) part is copied to the processed solid and needs his own memory again.

 To select shared Solids, please start the command ***Select shared Solids*** from a <u>user defined Toolbar</u> or from a <u>user defined menu</u> and identify the respective Solid in the 3D window.

Specify settings for processing fixed solids

Using the menu command ***Settings, 3D/2D Commands***, the ***Solid*** option card can be used to determine presettings in order to process fixed solid.
All these presettings can be modified after the command is loaded.

The following four values are settings for the command ***Fixed Solid, Blend/Chamfer***:

blend radius
Defines the suggested value for the blend radius.

distance
Defines the suggested value for the right and left distance of an edge to be chamfered.

face indent
Defines the value, with which special corners should be processed, if all the edges which collide are selected.

select edges sequentially
If this setting is active, when picking individual edges, all edges are also selected which connect tangentially to this edge and which also have tangential transitions.
If this is not active when picking an edge, simply this edge is selected.

Hollow body

wall thickness
Defines the suggested value for the wall thickness when executing the command ***Fixed Solid, Hollow Solid***.
If the value is negative, the material in the inside of the solid is removed.
If the value is positive, material is added to the outside of the solid.

Local Operations

The settings made here for ***offset*** and ***taper angle*** relates to the commands in the ***Solid local Operations*** menu***.***

Specify settings for generating solid axes

When generating primitives (block, cylinder, etc.), these are allocated solid axes at the same time. For solids which are generated from 2D faces, this does not happen, except for rotational solids.
In the command Transform, Show Geo Points a 3D axis can be used to move a solid in the direction of the axis. Also workplanes can be defined using 3D axis.

Using the menu command ***Settings, 3D/2D Commands***, the ***2D/3D axes*** option card can be used to determine the settings for the generation of axes and also of 2D centre lines.

define axes with overhang
The length by which the solid axes is to project over the solid face can be entered here.

If this setting is not activated by the check box, the axes end at the solid surfaces.

The value determined here is also valid for 2D centre lines which have been allocated an overhang over their calculated endpoint.

layer for 2D axes
The name of the currently set layer for 2D centre lines is displayed here.
This layer can be defined using the ***Layer structure tree*** in the ***Model Explorer***.

display 3D axes
The colour and line type with which the solid axes are to be displayed can be specified here. If the settings are deactivated using the check box, the solid axes are not displayed.
These settings are not valid for 2D centre lines.

Please note:

 If you have to change the display of 3D axis within the 3D window quite often, you are able to switch the display using a separate icon in your user defined symbol bar, in your User Defined Menu or by a User defined key commands.
The option ***Display of 3D axis on/off*** can be found in the folder ***General CAD Commands, General 3D CAD Commands*** (Dialogue window for user defined toolbars).
If you would like to assign the command to a hotkey, the command name is: **`".op.me3d.show3DAxes"`**

Display settings for 2D centre lines
In this area the object related display settings for 2D centre lines can be defined. Please keep in mind that object related settings overlap the layer relation. To define the settings with a layer relationship, they must be defined in the ***Layer Manager*** inside the ***Model Explorer***.

Please note:

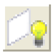
The display of the line type can be influenced using the menu command ***Settings, Options*** with the factor for the ***line pattern***.

Generate block

from ***BeckerCAD 3D***

Use the following procedure for generating a fixed solid block:

1.   Activate ***Fixed Solid, Block*** by clicking the (blue) symbols shown here.

2. , 
   If the face forming the base of the block is to lie inside one of the current WPs, you must activate whichever WP it is.

    If the base of the block lies in a temporary WP, activate this icon in the 3D toolbar and define the WP.
   It is removed again after quitting the command.

3. If you wish to create a hollow solid with a wall thickness, where the base and top face should be open, define a value for the ***wall thickness*** in the status dialog window. At the remaining faces material will be added from the outer faces to the inner direction when entering a negative value and material will be added outside when entering positive values.
   The default value for the wall thickness is 0. Within an active command a defined value will remain to create multiple solids with the same wall thickness.

4. Define the corner point of the base.
   Click the ***X*** key to create the base of a block by centre point.

5. If the edge of the base is not to run parallel to the coordinate axes, enter the angle under which the edge is to run in the *Angle* text box of the Status dialog box.

6. Define the edge lengths of the rectangular base by defining the diagonally opposite corners or by entering the values for ***Length X*** and ***Length Y***.
   If you have specified the size of an ***Angle***, the ***Length X*** will be measured off in that direction.
   The base area of a block can be defined by 3 points also. Define the first axis by 2 points and then the second axis by a third point.

7. If the block is to be drawn with an identical height on both sides of the WP, you must enter a numerical value in the box for ***Height 2*** first.
   The height specified here will be the amount by which the block will be extended in the opposite direction to the height specified in the following step.
   If the entry is deleted and the action confirmed by your pressing *Enter*, the height of the block on either side of the WP will be that specified in the next step.

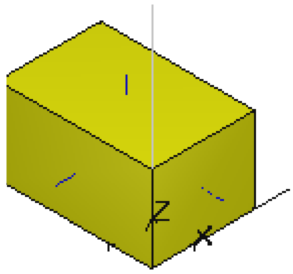
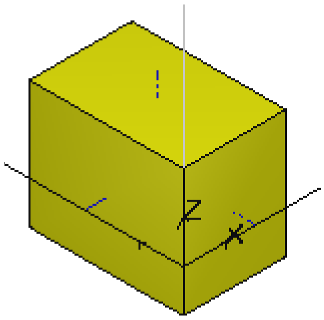
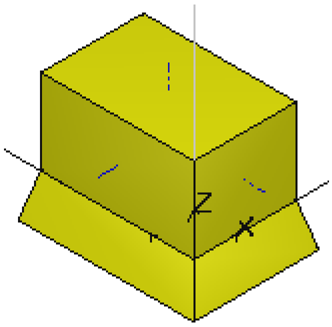
8. Specify the height of the block by snapping a point whose distance to the WP corresponds to the height or by entering a numerical value in the ***Height 1*** text box of the Status dialog box.
A positive height value generates a block "above" the WP, a negative one „below“.

Please note:

In step 6 you can insert a pyramid instead of a block. To do this type your entries in the ***Distance 2*** box or the ***Inclination 2*** box.

Examples:

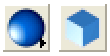
Height 1 = 100	Height 1 = 100	Height 1 = 100
Inclination 1 = 0	Inclination 1 = 0	Inclination 1 = 0
Height 2 = 0	Height 2 = 50	Height 2 = 50
Inclination 2 = 0	Inclination 2 = 0	Inclination 2 = 15

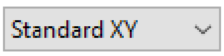




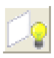
Generate pyramid

from ***BeckerCAD 3D***

Use the procedure described below to generate a four-sided fixed solid pyramid or a frustum of a pyramid:

1.  Activate ***Fixed Solid, Block*** by clicking the (blue) symbols shown here.

2. 

, 
If the base of the pyramid is to lie in one of the existing WPs, activate whichever of them is to be used.

 If the base of the pyramid is to lie in a temporary WP, activate the symbol shown here which can be found in the 3D toolbar and then define the WP.
It will be deleted as soon as you have finished with this command.

3. If you wish to create a hollow solid with a wall thickness, where the base and top face should be open, define a value for the ***wall***

thickness in the status dialog window. At the remaining faces material will be added from the outer faces to the inner direction when entering a negative value and material will be added outside when entering positive values.
The default value for the wall thickness is 0. Within an active command a defined value will remain to create multiple solids with the same wall thickness.

4. Next define the base for the pyramid, using the same method as for a block.

inclination and height

5. Enter the size of the angle in the ***Inclination 1*** box to specify the angle between all the lateral faces and the perpendicular.
A positive angle with a positive height will specify an outward inclination, and a negative an inward inclination.

6. Specify what height you want the pyramid to be by snapping a point that is at the same distance from the WP as the final height, or by typing a length in the ***Height 1*** box.
A height that is positive will result in a pyramid lying above the WP, and a negative height one that lies “beneath" the WP.

distance and height

5. Enter a numerical value in the ***Distance 1*** box to specify the distance separating all the edges of the top surface from the base. A distance that is positive will result in a top surface larger than the base.

6. Specify the height that you want the pyramid to be by snapping a point that is at the same distance from the WP as the final height, or by typing a length in the ***Height 1*** box.
A height that is positive will result in a pyramid lying above the WP, and a negative height one that lies “beneath" the WP.

Please note:

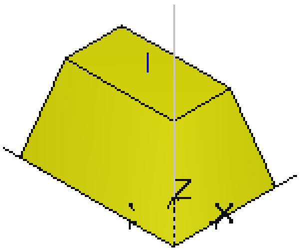
If the pyramid is to be extended by a block on the other side of the WP, you must precede step 5 by entering an appropriate height in the ***Height 2*** box.

If you want to add a second pyramid, type its height in the ***Height 2*** box and then enter a numerical value in the ***Inclination 2*** or the ***Distance 2*** box.

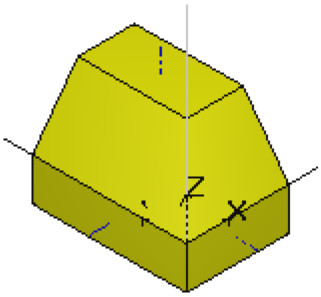
If you delete the entry in one of these boxes and press Enter to confirm, the same numerical value will apply to both sides of the WP.

Examples:

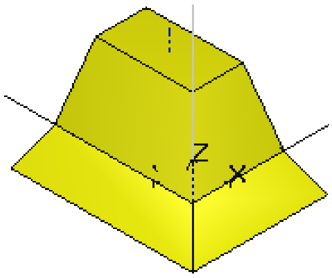
Height 1 = 100
Inclin. 1 = -15
Height 2 = 0
Inclin. 2 = 0



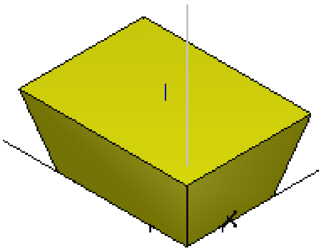
Height 1 = 100
Inclin. 1 = -15
Height 2 = 50
Inclin. 2 = 0



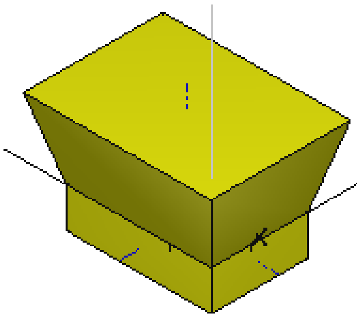
Height 1 = 100
Inclin. 1 = -15
Height 2 = 50
Inclin. 2 = 30



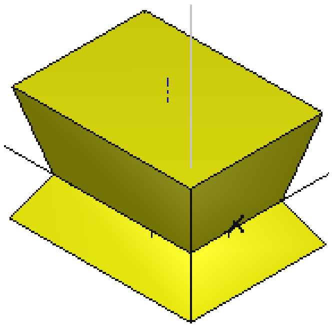
Height 1 = 100
Inclin. 1 = 15
Height 2 = 0
Inclin. 2 = 0



Height 1 = 100
Inclin. 1 = 15
Height 2 = 50
Inclin. 2 = 0



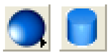
Height 1 = 100
Inclin. 1 = 15
Height 2 = 50
Inclin. 2 = 30

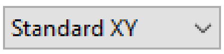


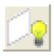
Generate cylinder

from ***BeckerCAD 3D***

Use the following procedure for generating a fixed solid cylinder:

1.  Activate ***Fixed Solid, Cylinder/Cone*** by clicking the (blue) symbols shown here.

2. 

, 

If the base of the cylinder is to lie in one of the existing WP, activate whichever of them is to be used.

 If the base of the cylinder is to lie in a temporary WP, activate the symbol shown here in the 3D toolbar and then define the WP. It will be deleted as soon as you have finished with this command.

3. If you wish to create a hollow solid with a wall thickness, where the base and top face should be open, define a value for the ***wall***

thickness in the status dialog window. At the remaining faces material will be added from the outer faces to the inner direction when entering a negative value and material will be added outside when entering positive values.
The default value for the wall thickness is 0. Within an active command a defined value will remain to create multiple solids with the same wall thickness.

4. Use the context dialog to specify whether the base of the solid is to be a ***full circle*** or an ***arc***.

5. In the same way decide whether you are going to draw the full circle (the arc) dynamically or by specifying ***3 points*** on the ***circumference***.

6. Specify the base to comply with your entries.
This is done by entering numerical values for ***Diameter 1***, ***Length***, ***Angle*** and the ***Open.angle*** inside the Status dialog box.

7. If the cylinder is to be drawn with an identical height on both sides of the WP, you must enter a numerical value in the box for ***Height 2***.
The height specified here will be the amount by which the cylinder will be extended in the opposite direction to the height specified in the following step.
If the entry is deleted and the action confirmed by your pressing *Enter*, the height of the cylinder on either side of the WP will be that specified in the next step.

8. Define the height of the cylinder.
Enter the value in the ***Height 1*** text box of the Status dialog box or determine a point whose distance to the WP corresponds to the height.
A positive height value generates a cylinder "above" the WP, a negative one „below“.

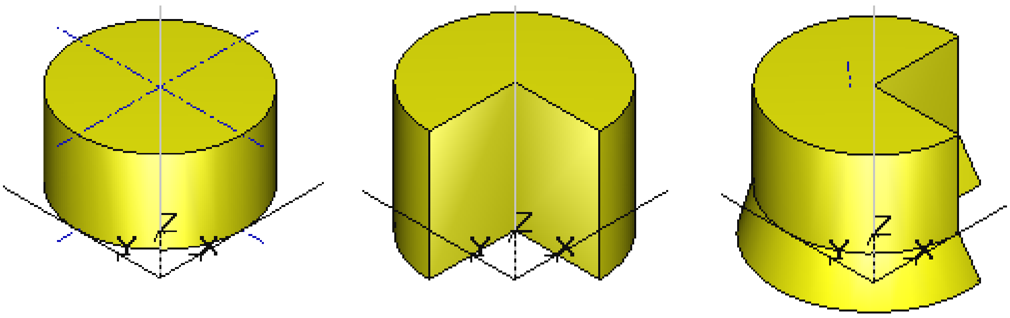
Please note:

If you are adding a cone (frustum of a cone) rather than a cylinder in step 6, you must enter the numerical values in the boxes for ***Diameter 2*** or ***Inclination 2***.

Examples:

Angle = 0	Angle = 270	Angle = 0
OpenAngle = 360	OpenAngle = 270	OpenAngle= 270
Height 1 = 100	Height 1 = 100	Height 1 = 100
Inclination 1 = 0	Inclination 1 = 0	Inclination 1 = 0

Height 2 = 0 Inclination 2 = 0 | Height 2 = 50 Inclination 2 = 0 | Height 2 = 50 Inclination 2 = 15



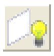
Generate cone (frustum)

from ***BeckerCAD 3D***

Use the following procedure for generating a fixed solid cone (frustum):

1.   Activate ***Fixed Solid, Cylinder/Cone*** by clicking the (blue) symbols shown here.

2. 

, 
If the face forming the base of the cone is to lie inside one of the current WPs, you must activate whichever WP it is.

 If the base of the cone is to lie in a temporary WP, activate this icon in the 3D toolbar and define the WP.
It will be deleted as soon as you have finished with this command.

3. If you wish to create a hollow solid with a wall thickness, where the base and top face should be open, define a value for the ***wall thickness*** in the status dialog window. At the remaining faces material will be added from the outer faces to the inner direction when entering a negative value and material will be added outside when entering positive values.
The default value for the wall thickness is 0. Within an active command a defined value will remain to create multiple solids with the same wall thickness.

4. Use the context dialog to specify whether the base of the solid is to be a ***full circle*** or an ***arc***.

5. In the same way decide whether you are going to draw the full circle (the arc) dynamically or by specifying ***3 points*** on the ***circumference***.

6. Specify the base to comply with your entries.
This is done by entering numerical values for ***Diameter 1***, ***Length***, ***Angle*** and the ***Open.angle*** inside the Status dialog box.

7. Select one of the following procedures in order to determine the further dimensions of the cone/frustum:

Diameter and height

8. In the ***Diameter 1*** text box of the Status dialog box, enter the radius of the top surface.
If the diameter is 0.0, a cone is formed, if larger a frustum is formed.

9. Specify the height of the cone.
Enter the value in the ***Height 1*** text box or determine a point whose distance to the WP corresponds to the height.
A positive height value generates a cone (frustum) "above" the WP, a negative one „below“.

Inclination and height

8. In the ***Inclination 1*** text box of the Status dialog box, enter the angle indicating the angle between the area of cylindrical surface and the perpendicular. A positive angle will enlarge the cone towards its top and a negative angle will reduce it.

9. Specify the height of the cone
Enter the value in the ***Height 1*** text box or determine a point whose distance to the WP corresponds to the height.
A positive height value generates a cone (frustum) "above" the WP, a negative one „below“.

Please note:

If you are adding a cylinder to the cone on the WP, you must enter a numerical value in the ***Height 2*** box to specify the height of this cylinder before completing step 8.

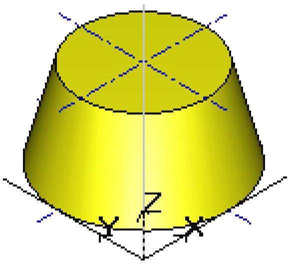
If you are adding a second cone to the one already completed, you must enter a numerical value for the new cone in the ***Height 2*** box and then specify ***Diameter 2*** or ***Inclination 2*** or the ***Distance 2***.

By deleting the numerical entries in tone of these boxes and confirming the action by pressing Enter you can draw a cone on either side of the WP with the same dimensions.

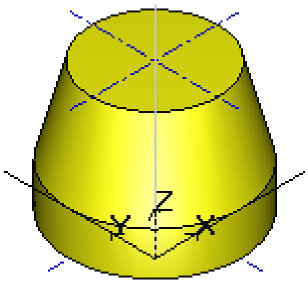
Examples:

Height 1 = 100 Inclin. 1 = -15	Height 1 = 100 Inclin. 1 = -15	Height 1 = 100 Inclin. 1 = -15

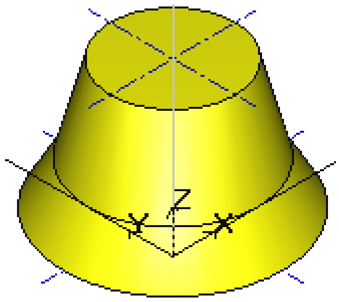
Height 2 = 0
Inclin. 2 = 0



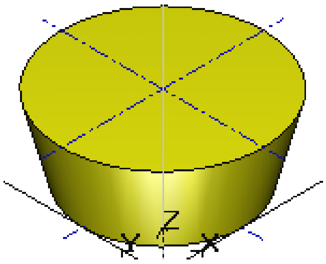
Height 2 = 50
Inclin. 2 = 0



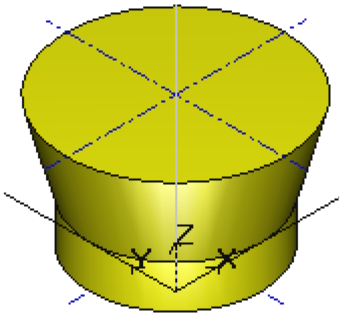
Height 2 = 50
Inclin. 2 = 30



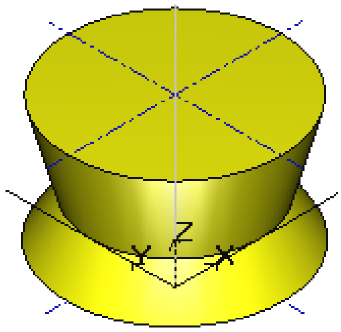
Height 1 = 100
Inclin. 1 = 15
Height 2 = 0
Inclin. 2 = 0



Height 1 = 100
Inclin. 1 = 15
Height 2 = 50
Inclin. 2 = 0



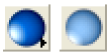
Height 1 = 100
Inclin. 1 = 15
Height 2 = 50
Inclin. 2 = 30



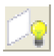
Generate sphere

from ***BeckerCAD 3D***

Use the following procedure for generating a fixed solid sphere or partial sphere:

1.  Activate ***Fixed Solid, Cylinder/Cone*** by clicking the (blue) symbols shown here.

2. 

, 
If the equator of the sphere is to lie inside one of the current WPs, you must activate whichever WP it is.

 if the face forming the equator of the sphere is to lie in a temporary WP, activate this icon in the 3D toolbar and define the WP.
It will be deleted as soon as you have finished with this command.

Solid sphere

3. Activate the ***Full circle*** option in the contextual menu.

4. Then specify in the same dialog box whether you are going to draw the circle ***dynamically*** or by specifying ***3 points*** on the circumference.
5. Specify the circle forming the face of the equator according to comply with your entries.
   This is done by entering the suitable ***Diameter 1*** in the Status dialog box.

Partial sphere with full circle as equator face

3. Activate the ***arc*** option in the contextual menu.
4. Select ***dynamic***.
5. Specify the midpoint of the circular face.
6. Specify the diameter by positioning a point on the circumference or by typing a numerical value for it in the ***Diameter 1*** box.
7. The procedure for specifying an arc is aborted by pressing the right mouse button.
   You have finished specifying the equator face as a full circle.

An arc will be displayed perpendicular to the face of the equator so that you can specify a sector of a sphere.

8. Specify the size of the opening angle of the segment of the sphere either by positioning a point or by typing the size of the angle in the ***Angle*** box.
   The size specified can be between –90° - 90°.

   If you want to generate half a sphere, your entry must be 0.
9. The opening angle for the segment is specified by positioning a point or by entering a numerical value in the ***Open.angle*** box.
   A positive value will result in the angle being drawn anti-clockwise beginning at the start angle.

   If the half sphere is to be generated above the face forming the equator, you must specify an angle of 90°. If the half sphere is to be generated beneath the face, you must specify an angle of –90°.

Partial sphere with circle segment as equator face

3. Activate the ***arc*** option in the contextual menu.
4. Select either of the options ***dynamically*** or ***3 points***.
5. Specify the arc according to the option you have just chosen.

An arc will be displayed perpendicular to the face forming the equator so that you can draw the segment of the sphere.

6. If you want to specify a segment in the plane of this arc as well, you must specify the start angle and opening angle.

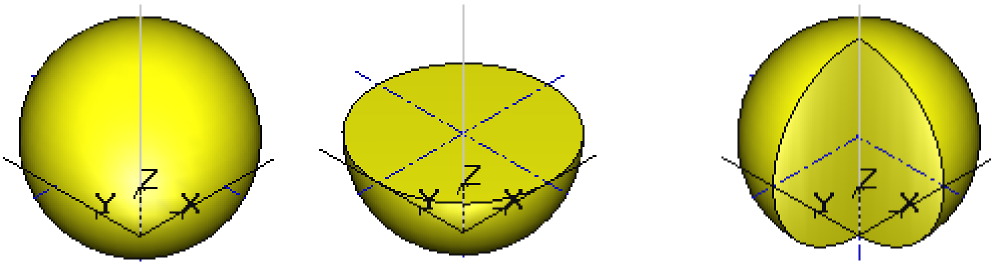
If you want to specify a complete circle, abort the procedure for defining a segment by pressing the right mouse button.

Please note:

Now the snap option ***middle*** may also be applied to the basic solids ***sphere*** and ***torus***. These snap points may be used just like faces and axes.

Examples:

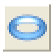
Sphere	part. sphere with circle as equator face	part. sphere with arc as equator face
	sector start angle = 270 sector open angle = 90	start angle = 270 open angle = 270

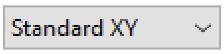


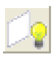
Generate torus

from ***BeckerCAD 3D***

Use the following procedure for generating a fixed solid torus or partial torus:

1.   Activate ***Fixed Solid, Torus*** by clicking the (blue) symbols shown here.

2. 

, 

If the face forming the equator of the torus is to lie in one of the existing WPs, you must activate whichever WP it is.

 if the face forming the equator of the torus is to lie in a temporary WP, activate this icon in the 3D toolbar and define the WP.
It will be deleted as soon as you have finished with this command.

3. If you wish to create a hollow solid with a wall thickness, where the base and top face should be open, define a value for the ***wall***

thickness in the status dialog window. At the remaining faces material will be added from the outer faces to the inner direction when entering a negative value and material will be added outside when entering positive values.
The default value for the wall thickness is 0. Within an active command a defined value will remain to create multiple solids with the same wall thickness.

4. In the Presettings for *circle definition* determine whether the torus is to be closed (*circle*) or open (*arc*).

5. In the same way, determine whether the design of the circle/arc is to be made *dynamic* or using *3 points*.

6. Enter the numerical values in accordance with the torus line.

7. In the Presettings for *circle definition* select whether the cross-section of the torus is to be a *circle* or *arc*.
The default will be the setting made in step 1.

8. Define the radius of the cross-section using the cursor or by making an entry in the ***Diameter 1*** text box of the Status dialog box.

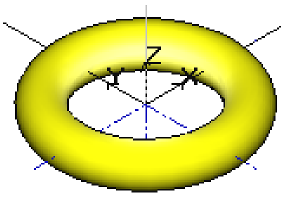
9. If the cross-section is to be an arc, define the starting angle and opening angle using the cursor, or enter the values in the ***Angle*** and ***Open.angle*** text boxes in the Status dialog box.

Please note:

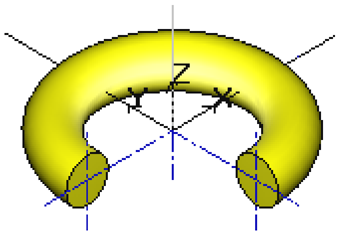
Now the snap option ***middle*** may also be applied to the basic solids ***sphere*** and ***torus***. These snap points may be used just like faces and axes.

Examples:

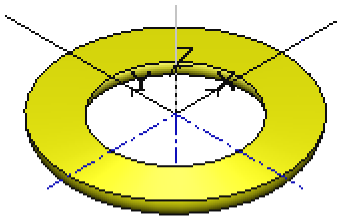
Start angle = 0
Open angle = 360



Start angle = 270
Open angle = 270



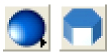
Intersection:
Start angle = 180
Open angle = 180



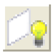
Generate prism

from ***BeckerCAD 3D***

Use the following procedure for generating a fixed solid prism:

1.  Activate ***Fixed Solid, Prism*** by clicking the (blue) symbols shown here.

2. 

, 
If the face forming the base of the prism is to lie in one of the existing WPs, you must activate whichever WP it is.

    If the face forming the base of the prism is to lie in a temporary WP, activate this icon in the 3D toolbar and define the WP.
   It will be deleted as soon as you have finished with this command.

3. If you wish to create a hollow solid with a wall thickness, where the base and top face should be open, define a value for the ***wall thickness*** in the status dialog window. At the remaining faces material will be added from the outer faces to the inner direction when entering a negative value and material will be added outside when entering positive values.
   The default value for the wall thickness is 0. Within an active command a defined value will remain to create multiple solids with the same wall thickness.

4. Specify the midpoint of the face forming the base.
   After the centre point is defined, using the hotkey ***X*** it can be switched between Incircle and Circumcircle.

5. Specify the number of corners by entering a number in ***Number***.

6. Specify how the edges are to be oriented by entering a number in the ***Angle*** box in the Status dialog box.

7. Specify the diameter of the circumcircle by positioning a point or by entering a numerical value in the ***Diameter 1*** box.

Prism with vertical lateral faces

8. If the altitude of the prism is to be identical on either side of the WP, first enter a numerical value in the ***Height 2*** box.
   This will lengthen the prism in the opposite direction to the altitude that is to be specified in the next step.
   If the number is deleted and you confirm the action by pressing *Enter*, the altitude of the prism on either side of the WP will be that specified in the next step.

9. Specify the altitude of the prism by snapping a point that lies at a distance from the WP corresponding to the altitude, or by entering a number in the ***Height 1*** box.
A positive number for the altitude will generate a prism "above" the WP, and a negative one "beneath" the WP.

Prism with inclined lateral faces

8. Enter a numerical value in the ***Inclination 1*** box to specify the angle by which the lateral faces of the prism are to diverge from the perpendicular.
A positive angle will generate a prism that increases in size towards the top, and a negative angle will generate a prism decreasing in size.

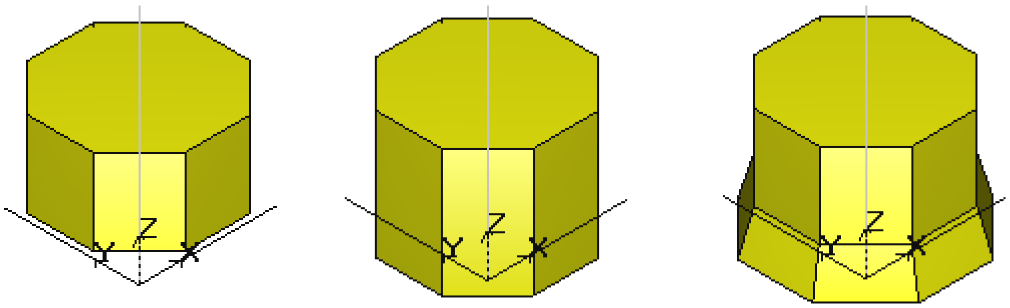
9. If the prism is to be of the identical altitude on either side of the WP, first enter a number in the ***Height 2*** box.
As a result of this entry the prism will increase in size away from the altitude that will have to be specified.
After you have deleted this entry and confirmed the action by pressing *Enter*, the altitude of the prism on either side of the WP will be that specified.

10. If the prism when extended is to have inclined lateral surfaces, enter an angle in the ***Inclination 2*** box.
If this entry is deleted, and you confirm the action by pressing *Enter*, the lateral faces will have the identical inclination on either side of the WP.

11. Specify the altitude of the prism by snapping a point at a distance from the WP corresponding to the altitude you want, or by entering a numerical value in the ***Height 1*** box.
A positive number will result in a prism "above" the WP and a negative one "below".

Examples:

Height 1 = 100	Height 1 = 100	Height 1 = 100
Inclination 1 = 0	Inclination 1 = 0	Inclination 1 = 0
Height 2 = 0	Height 2 = 50	Height 2 = 50
Inclination 2 = 0	Inclination 2 = 0	Inclination 2 = 15



Generate swept solid along Z axis

Swept solids along the Z axis are generated by moving a face along the vertical axis as far as a defined height. This face can be a 2D face in the current WP or a flat solid face.

When creating some of primitives and swept solids a wall thickness can be entered in the status dialog window to create a hollow solid, where the base and top face is opened.

Options found in a context dialog can be selected to determine the shape of the solid.
In this way you can create a solid with or without inclined lateral faces.
The solid can also be blended into the surface of another solid that has already been completed.

You will find a full description of how to generate swept solids of this type in the following two chapters.

Generate swept solid without inclination of lateral faces

Use the following procedure for generating a swept solid without inclination of the lateral faces:

1.  Start the command by clicking the icons shown here.

2. If you wish to create a hollow solid with a wall thickness, where the base and top face should be open, define a value for the ***wall thickness*** in the status dialog window. At the remaining faces material will be added from the outer faces to the inner direction when entering a negative value and material will be added outside when entering positive values.
The default value for the wall thickness is 0. Within an active

command a defined value will remain to create multiple solids with the same wall thickness.

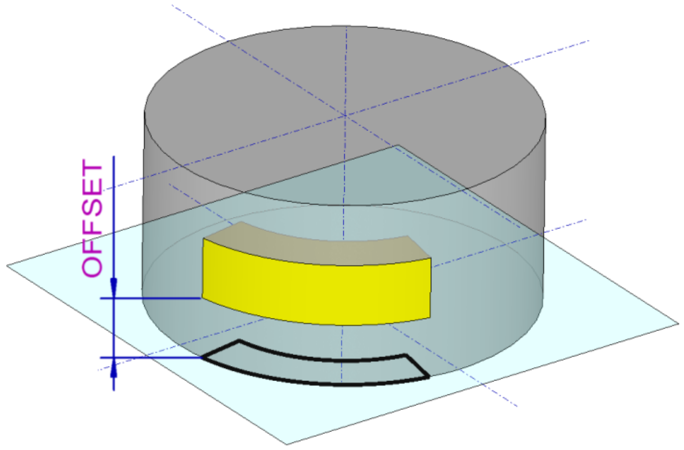
3. Using 2D faces

If no 2D face is defined, select the bordering 2D line objects. Define the 2D face to be moved by positioning the cursor in a completely bordered area and confirm with the left mouse button. If the Ctrl key is kept pressed, a number of areas can be defined.

If 2D faces already exist, identify them with the cursor or collect them by keeping the Ctrl key pressed.

Using flat solid faces

Identify the required flat solid face with the cursor, for non shaded representation, two edges which border this face.



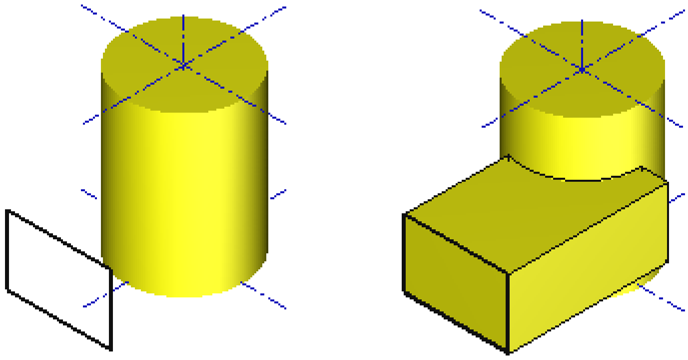
4. If the swept solid should be created with a distance to the active plane, please define an ***Offset*** in the status dialogue window for the distance between the base face of the swept solid and the 2D face or the planar surface.

5. Specify the height of the swept solid.
Type a value in the ***Height 1*** box, or define a point in the current WP to correspond to the height of the solid.
A positive value will result in a solid "above" the WP and a negative value one "below" the WP.

6. A context dialog will appear for you to select one of the ***Presettings***.

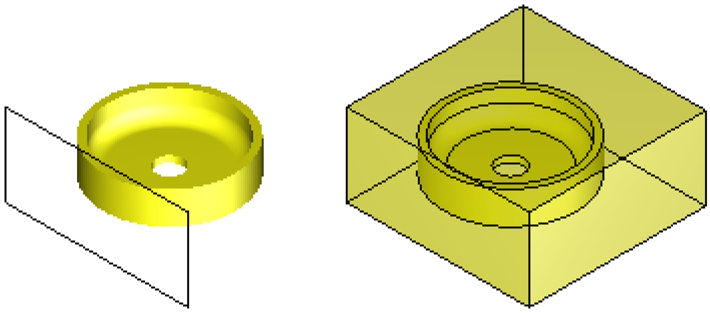
7. When you decide to generate the solid using the ***Height***, this is the only step needed to define the solid.
If you decide to use another method, identify the target solid.

Height + Trim
When you are using this method, the swept solid will be blended into the top surface of a solid that already exists, as shown here:



Height 1 will, in this case, determine the extension of the swept solid, if it runs, in part at least, close to the target solid.

This method can be used to generate the "negative" of a solid that already exists. This can only be done, if the swept solid completely encloses the target solid.



Height + Intersection
With this method the swept solid will be generated first. Afterwards an intersection will be formed between it and the target solid.

Height + Subtraction
With this method the swept solid will be generated first. Afterwards it will be subtracted from the target solid.

Height + Union
With this method the swept solid will be generated first. Afterwards a union will be formed between it and the target solid.

To Face + Subtract
Using this option the solid is created up to a planar face and then subtracted from this or any other solid.

To Face + Unite
Using this option the solid is created up to a planar face and then united with this or any other solid.

Please note:

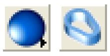
You may want to extend the swept solid to the other side of the

WP. This can be done by entering a value for ***Height 2*** before completing Step 2.

If the lateral faces of the swept solid or the extensions of them are to be inclined, use the procedure described immediately below in the next section.

Generate swept solid with inclined lateral faces

Use the following procedure for generating a swept solid with inclined lateral faces:

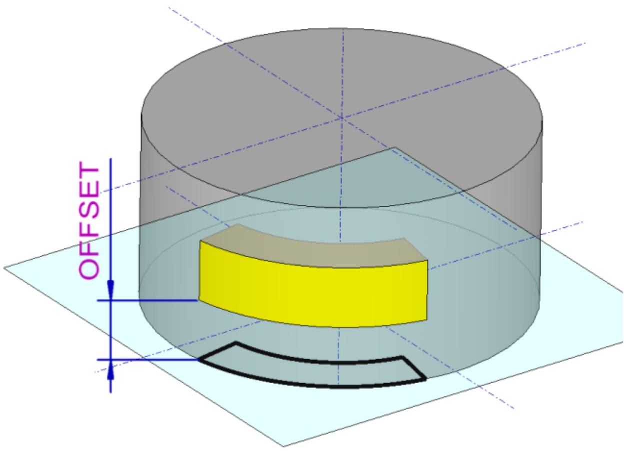
1.  Start the command by clicking the icons shown here.

2. If you wish to create a hollow solid with a wall thickness, where the base and top face should be open, define a value for the ***wall thickness*** in the status dialog window. At the remaining faces material will be added from the outer faces to the inner direction when entering a negative value and material will be added outside when entering positive values.
The default value for the wall thickness is 0. Within an active command a defined value will remain to create multiple solids with the same wall thickness.

3. **Using 2D faces**
If a 2D face has not been defined, first select the bounding 2D line objects.
Define the face that is to be moved by positioning the cursor inside a completely enclosed section and then clicking the left mouse button.
By keeping *Ctrl* pressed down at the same time you can collect several faces.

But if there are 2D faces already available, use the cursor to identify one of them, or collect them by keeping *Ctrl* pressed down at the same time.

Using flat solid faces
Identify one of the flat faces with the cursor, or, if the display is not shaded, two of the edges bounding the face.



4. If the swept solid should be created with a distance to the active plane, please define an ***Offset*** in the status dialogue window, for the distance between the base face of the swept solid and the 2D face or the planar surface.

Inclination and Height

5. Type an angle size in the ***Inclination 1*** box to specify the angle all the lateral faces are to deviate from the perpendicular.
   A positive angle with a positive height will result in an inclination outwards, and a negative angle an inclination inwards.

Distance and Height

5. Type a number in the ***Distance 1*** box to specify the distance between the edges of the top surface and the edges at the base.
   A positive distance will result in a top surface that is larger than the base.

6. Select one of the options offering ***contour parallel with inclination***.
   These options have an effect on the shape of the top surface that forms a parallel contour to the face previously selected; in other words, this defines the shape of the "sharp" corners on the original face of the parallel contour.
   Examples are to be found in the illustrations immediately below.

7. Select one of the options for ***degeneration with inclination***.
   These options will determine how the command will be executed, when lines in the parallel contour attain a value of 0 as a result of shortening.
   A description of the options is given below.

8. A context dialog will appear for you to select one of the ***generation types***. Depending on the method chosen, you can then identify the target solid.
   A description of the methods is given in the previous chapter.

9. Specify the height of the swept solid by snapping a point at the height from the WP equivalent to the height you want, or by typing a number in the ***Height 1*** box.
A positive value will result in a swept solid "above" the WP and a negative value one "below" the WP.

Please note:

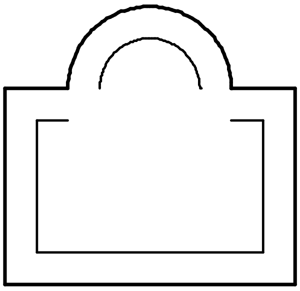
If you want to extend the swept solid with inclined lateral faces to the other side of the WP, first type the height of this extension in the ***Height 2*** box before continuing with Step 7.

If you want to extend the swept solid by the addition of a swept solid with inclined lateral faces, type the height in the ***Height 2*** box and then either ***Inclination 2*** or ***Distance 2***.

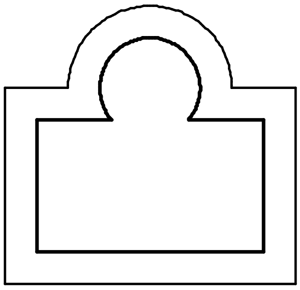
If you delete the entry in one of these fields and then press *Enter*, the swept solid will be the same size on both sides of the WP.

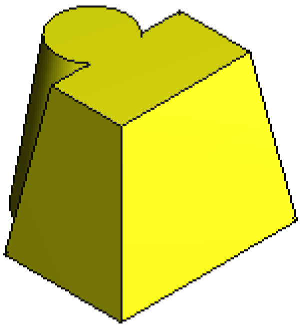
Contour parallel with inclination

The illustration immediately below is an example of a basic face. If you generate parallels to the boundary lines inside it, you can use the options to specify how the spaces between the parallels are to be closed:

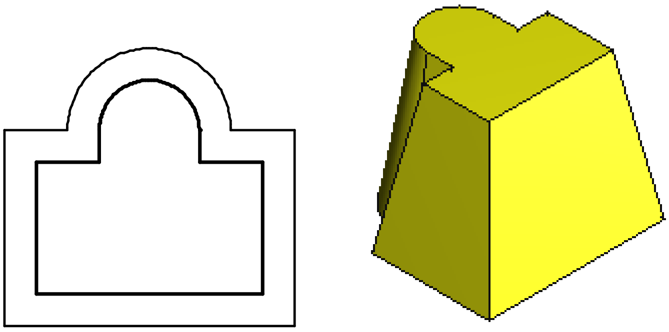


Extend lines
Curved line objects will be extended to conform with their geometric shape:



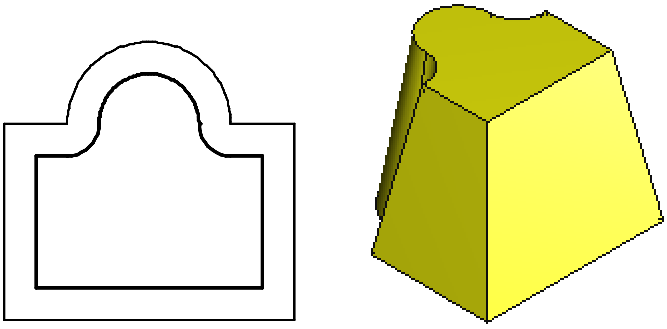


Insert lines
Curved line objects will be extended by tangential connecting lines:



Insert arcs

With the exterior contours an arc will be inserted with its centre forming the corner point between two adjacent line objects:



Degeneration with inclination

These options can be used to determine how the command is to be executed with degeneration of the top surface of the swept solid. Degeneration will result when a boundary line in the parallel contour determining the extent of the top surface has been shortened in such a way that it "disappears":

Do not check

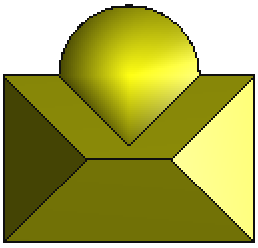
The effect of this option will be to exclude any degeneration check. This will accelerate the degeneration process appreciably.
The option can be used when the shape of the basic solid or the slight inclination is unlikely to result in any degeneration.

Do not generate solid

The effect of this option will be that you will get an error message during the degeneration procedure and no swept solid will be generated.

Generate solid fully

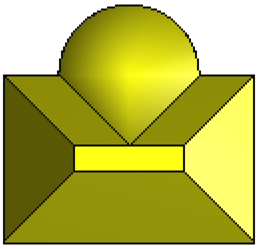
The effect of this option will be the solid will be generated as far as is possible without observing any degeneration, as shown here



It is important to remember that you could sometimes produce the wrong type of solid. In such a case use ***Information, Check Solid***, before you carry out any operations to produce the solid model.

Generate solid to 1st stage

The effect of this option will be that you will generate a swept solid that will not be changed further after you have reached the first stage of the degeneration process:



Generate swept solid along path

from ***BeckerCAD 3D pro***

,  Using the command ***Fixed Solid***, ***Swept Solid along Path*** a fixed solid can be defined by sweeping a face along a path.

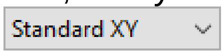
The face is defined as a closed 2D contour or a surface.

The path of the swept solid must be continuous and distinct, without intersections to other objects or branches. If these criterias are fulfilled the path of the swept solid can contain single objects or a collection of 2D and 3D curves, 3D polygons or even edges.

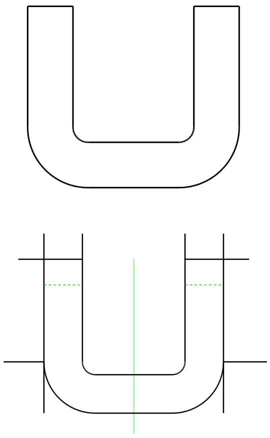
General Process

1. If you wish to create a hollow solid with a wall thickness, where the base and top face should be open, define a value for the ***wall thickness*** in the status dialog window. At the remaining faces material will be added from the outer faces to the inner direction when entering a negative value and material will be added outside when entering positive values.
The default value for the wall thickness is 0. Within an active command a defined value will remain to create multiple solids with the same wall thickness.

2. Select the path defining objects.

3. If the 2D elements defining the path and the 2D contour defining the section face or on different WP´s, now you cannot only switch between WP´s using the list box 

in the 3D toolbar but also starting the command  ***Activate WP*** in the 3D toolbar and then identify a frame line or a 2D object contained in this WP.



4. If the face for the solid is not yet defined as a 2D face select the 2D contour elements by collecting with the pressed ***CTRL*** key or by dragging a window. The selected elements should clearly define a closed contour. Overhangs, branches and intersections to other objects should be avoided.
   Define the face to be swept by clicking with the cursor in a closed area.

   If an existing 2D face or a surface should be used instead, just identify it with the cursor.

5. Define a reference point of the face. This point will follow the path.

6. A copy of the face with a temporarily work plane is transformed to the start point, so that the face it perpendicular to the path.
   If the position of the workplane and with it the position of the face should be changed, the commands to transform work planes can be used in this stage.

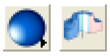
A swept solid can be created with additional options:

7. **Torsion**
   If the section face should be rotated around the path, enter the number of rotations in the field ***Torsion*** in the status dialogue window. This value is then aligned to the complete length of the path.

8. **Inclination**
   If the solid should be conical, enter a positive value in the field ***Inclination*** in the status dialogue window if the section face should be increased and a negative value if the section face should be decreased.
   If the path is closed, this option is not available.

9. Click the left mouse button to create the swept solid.

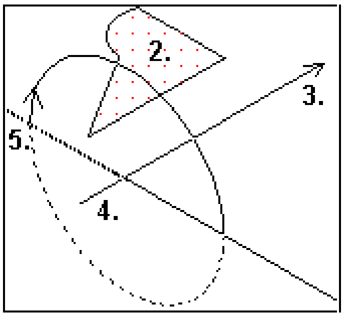
Generate rotational solid

 Alternatively, the command can be loaded using this icon.

This command can be used to generate fixed solids by rotating a 2D face by a defined axis in the current WP.
As the cross-sectional face for rotational solids, 2D faces bordering 2D line objects as well as flat faces of solids can be used.

If a flat solid face is to be used, a WP should be defined which lies in this face.

1. In the Presettings for ***Circle definition*** determine whether the rotational solid is to be closed (*full circle*) or open (*arc*).



2. **Using 2D line objects**

If no 2D face is defined, select the bordering 2D line objects by collecting with the *Ctrl key* pressed or by dragging a window. Define the face to be rotated by positioning the cursor in a completely bordered area and confirm with the left mouse button. If the *Ctrl key* is kept pressed, a number of areas can be defined.

If 2D faces already exist, identify them with the cursor or collect them by keeping the *Ctrl key* pressed.

Using flat solid faces

Identify the required flat solid face with the cursor, for non shaded representation, two edges which border this face.

3. The rotation axis can be defined by 2 points and also by identifying a line.
To define the rotation axis by 2 points start the point definition menu (middle mouse button) for both points or use the hotkeys to activate the appropriate snap option.
If an open rotational solid is to be generated, the first point determines the positive direction of the rotation axis and therefore the orientation of the rotation angle.
Define the second point of the rotation axis.
The axis cannot touch or cut the face to be rotated.

If a closed rotational solid is to be generated, all necessary parameters are defined.

Arc:

4. Specify a point outside the axis.
On this point, an arc is drawn about the endpoint of the axis, with which the angle value of the rotation is to be displayed.

5. Define the start angle for the rotational solid by entering a numerical value in the ***Angle*** text box of the Status dialog box or by defining a spatial point.
If the value for the angle is 0, the rotation begins in the current WP.

6. Specify the opening angle of the arc by entering a numerical value in the ***Open.angle*** text box in the Status dialog box or by defining a spatial point.

If the start and end angles are defined using two points, the rotation takes place from the first point to the second point.

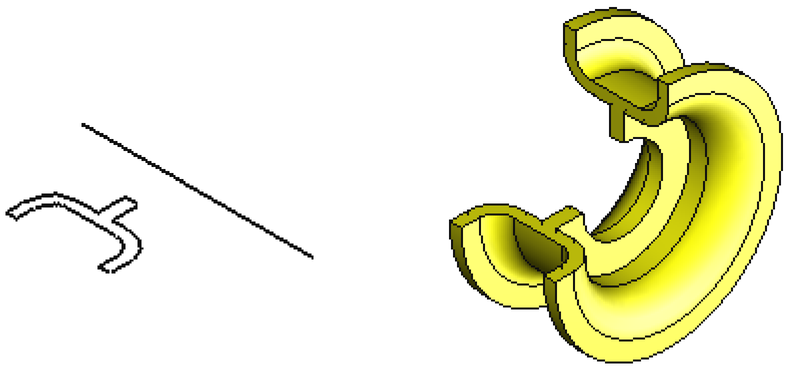
Please note:

If the 2D and 3D modes are active, 2D line objects, 2D faces and also solid faces can be selected.
The following rules apply for selecting with this command:

When you are dragging a window, only 2D line objects and 2D faces are selected. If only solids are in the window, a flat face of a solid is selected.

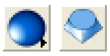
When you are identifying, all named objects can be selected, however solid faces are determined with priority.

Examples:



Generate transitional solid

from ***BeckerCAD 3D***

 Alternatively, the command can be loaded using this icon.

This command generates a fixed solid from two even faces, which originates from one face projecting into another.
The initial faces can be 2D objects of type face as well as flat solid faces.

The following restrictions exist for these faces:
They cannot lie in the same plane, they cannot border each other or have a common intersection line. They are also not to contain any "islands“.

There are two different methods of generating a transitional solid:

- automatic connection of the faces
- manual connection of the faces

Whichever method you use, you must take note of the following:

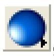
When both the 2D mode and the 3D mode are active during the procedure for selecting the faces, you can select 2D line objects, 2D faces, and also flat surfaces of solids.
When using this command to select, you must observe the following conditions:

When you are dragging a window, only the 2D line objects and the 2D faces will be selected.

When you are identifying anything, you can select all the named objects, but the program will. as a priority, find the flat surfaces of solids.

Options to Create Transitional Solids

from ***BeckerCAD 3D***

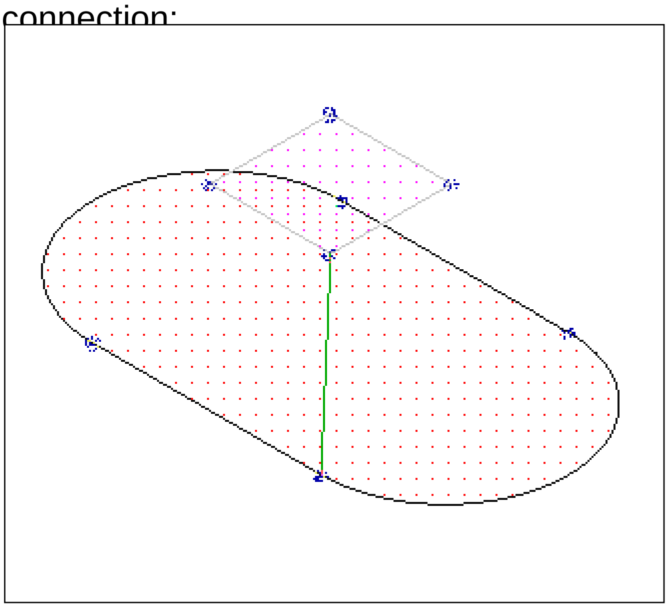
,  The command ***Fixed Solids***, ***Transitional Solids*** is started by clicking the displayed icons.

When creating transitional solids options can be set, to improve the quality. The popup menu ***Transitional Solid*** is offered after identifying two 2D faces or surfaces. Additional options can be set in the tab ***Parameters*** as described in the following:

Merge Coedges:

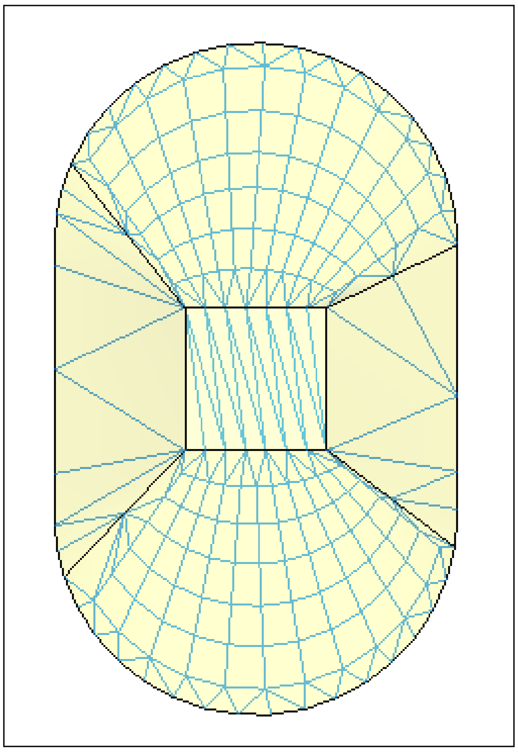
Tangent contours (e.g. line tangent to circle) are merged seamlessly. When edges are rounded, this may not result in the desired output and can be deactivated in this cases. The transition between the curves of the two faces will then be calculated separately.
The default for this option is **active**.

Example for a manual transitional solid with a point to point connection:

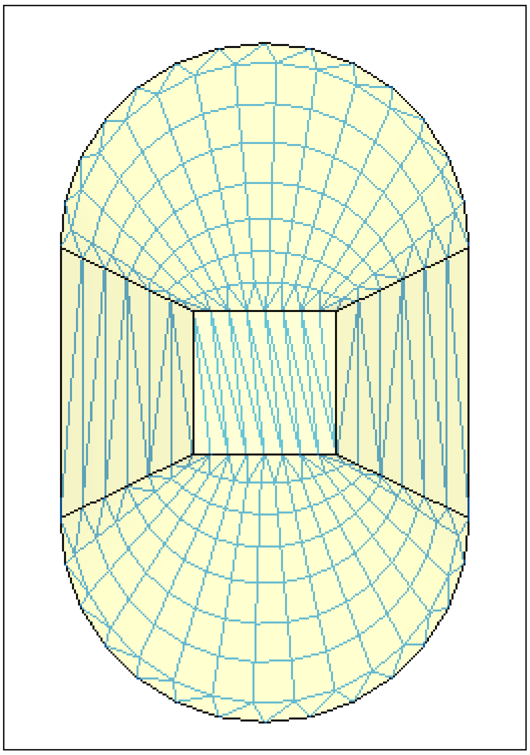


The option ***Merge Coedges*** is

active



inactive

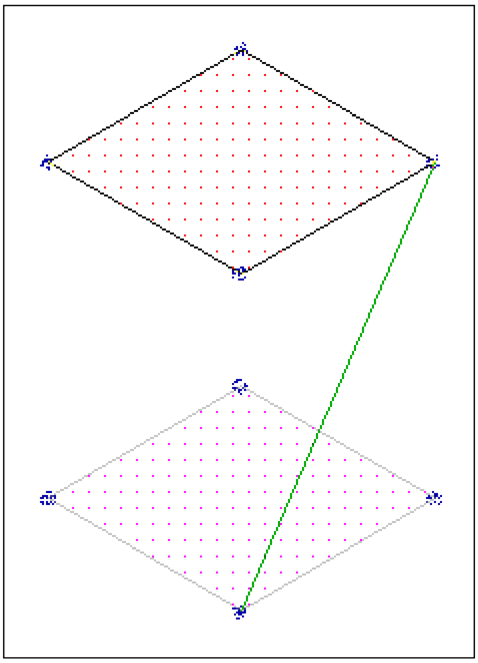


Match Vertices:

Fitting vertices are connected with priority. A distortion by manual connection will only affect partial. If necessary, additional points are added to a curve.

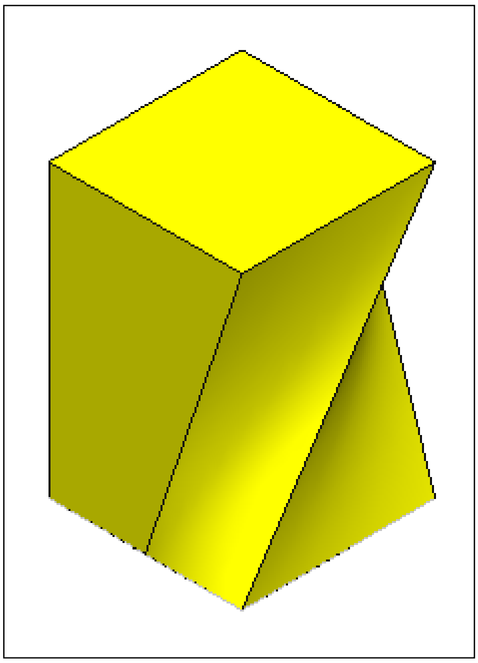
The default for this option is **active**.

Example for a manual transitional solid with a point to point connection:
(Condition: *Minimize Distortion* is deactivated)

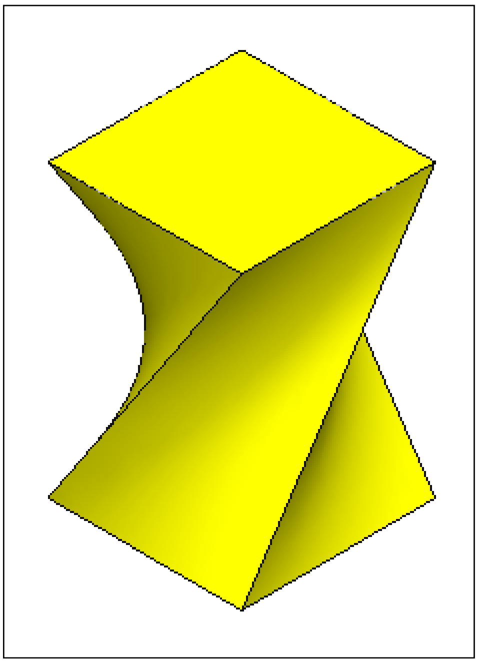


The option ***Match Vertices*** is

active



inactive

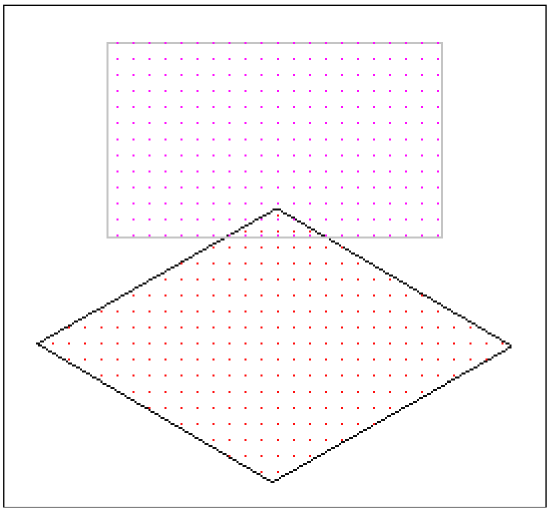


No new Twist Vertices:

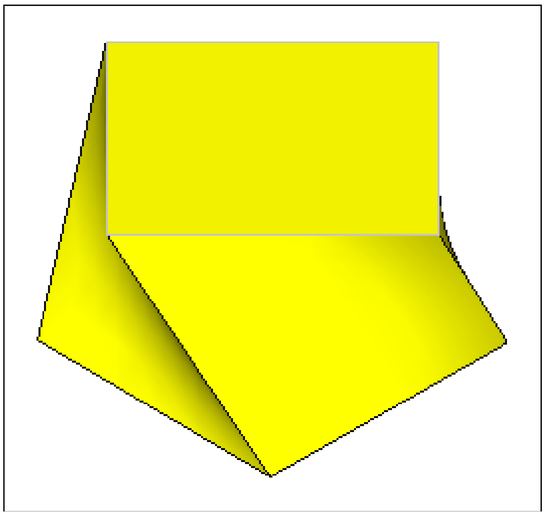
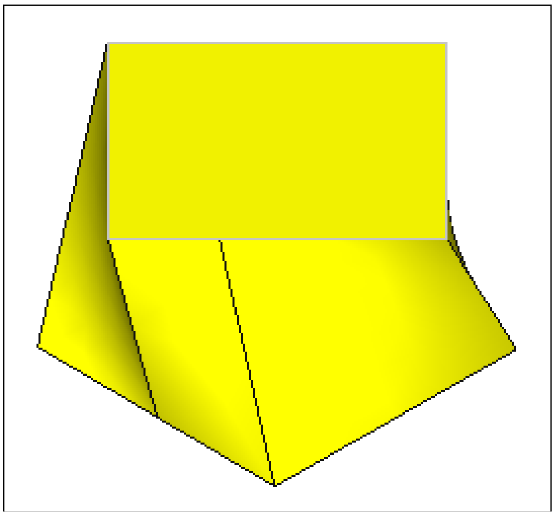
With the active option ***Minimize Distortion*** additional twist vertices

can be added, which can distort the result. With the option **No new Twist Vertices** the creation of additional twist vertices can be prevented.
The default for this option is **inactive**.

Example for an automatically created transitional solid with activated option ***Minimize Distortion***.



the option **No new Twist Vertices** is

active	**inactive**
	

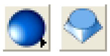
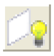
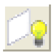
The optimal results are reached, when clicking the button ***Default Values*** in the popup window ***Transitional Solid*** in the tab ***Parameters***. Depending on the types of faces and the resulting transitional solid a different selection of parameters can deliver a better result.

Generate transitional solid automatically

from ***BeckerCAD 3D***

When you are generating a transitional solid automatically, the system will determine independently the corresponding (corner) points one the faces affected and will attempt to connect them with edges. ***Transitional Solids*** can be guided by lines using this option. The surface of the transitional solid will adapt to the guide lines. 2D lines can be used as guide lines. They have to be continuous without any sharp corner. Therefore single 2D lines (arcs, lines, splines) should be used as guide lines. The guide lines must start and end at the sketches or the edges.

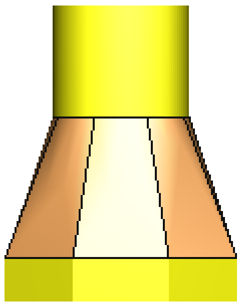
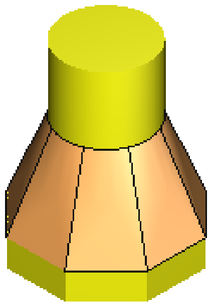
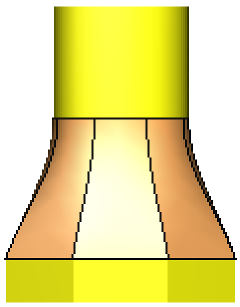
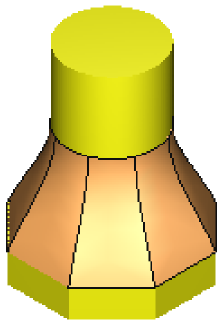
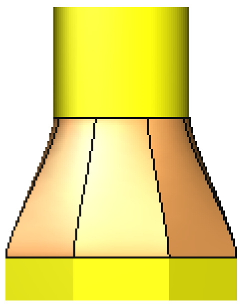
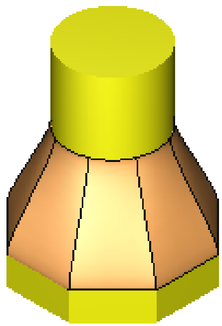
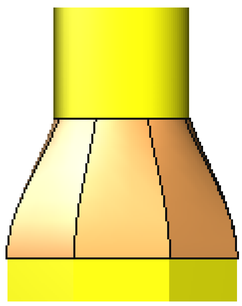
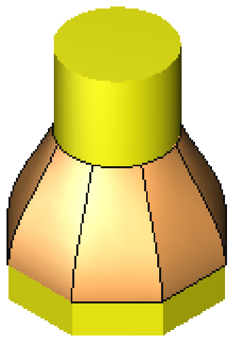
Use the following procedure to generate a transitional solid with automatic connection of the edges:

1.  Click the icons shown here in the 3D toolbar: ***Fixed Solid, Transitional solid***.
2.  If the first of the faces is a 2D face, activate the WP containing it.
3. Identify or specify the first face.
   You may have to change to a more suitable view first or use the 3D selection mode.
4.  If the second face is a 2D face, activate the WP containing it.
5. Identify or specify the second face.
   You may have to change to a more suitable view first or use the 3D selection mode.
6. The contextual dialog for specifying the method for connecting the faces will be displayed.
   Leave the option ***Automatic*** active.
7. If the transitional solid should follow guide lines, click the ***guide lines*** button. Identify previously created guide lines. Click the right mouse button to get to the next step in the command.
8. If the faces that have been identified are **solid faces**, two boxes in the ***tangential*** section will become active: the one on the left is for the first of the identified faces and the one on the right for the second face.
   When you are dealing with 2D faces, these boxes are not active.

   Use these boxes to indicate whether the transitional solid is to be given a linear or tangential connection to the surface:

If the entry is 0 will specify a linear connection, whereas 1 will specify the tangential type. If you wish, you can enter a numerical value between the two.

The following illustrations give you some examples of transitional solids generated between two faces of a solid to show the different ways of specifying tangential values:

Tangential	Result	
0 0		
1 0		
0.5 0.5		
0 1		

9. Activate the appropriate option in the tab Parameter, to get the desired result.

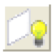
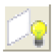
10. Start to generate the transitional solid by clicking ***Accept***.

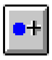
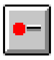
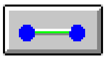
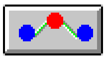
    As far as the geometry permits, the transitional solid will be generated. If it is impossible, an error message will be displayed.

Generate transitional solid manually

When a transitional solid is generated manually, you must first specify compulsory points on the perimeter of both faces. The by specifying a linear connection or triangular connection between the compulsory points on the two perimeters you can determine how the transition is generated.

Use the following method for generating a transitional solid that is to be completed manually by connecting points:

1.   Click the icons shown here in the 3D toolbar: ***Fixed Solid, Transitional solid***.
2.  If the first of the faces is a 2D face, activate the WP containing it.
3. Identify or specify the first face.
   You may have to change to a more suitable view first or use the 3D selection mode.
4.  If the second face is a 2D face, activate the WP containing it.
5. Identify or specify the second face.
   You may have to change to a more suitable view first or use the 3D selection mode.
6. The contextual dialog for specifying the method for connecting the faces will be displayed.
   Choose the option ***Manual***.
7. Specify the perimeter of the two faces by means of the **same number of points**. They will determine the transition from one area to the other.

   The two buttons  and  are for inserting extra points or for deleting ones not required. You will find information on how to do this in the following section.
8. Use one of the two buttons  or  to specify **the type of connection** between the points.
   information on how to do this in the section after the next.
9. Start generating the transitional solid by clicking ***Accept***.

As far as the geometry permits the transitional solid will be generated. Should this stage be impossible, an error message will be displayed.

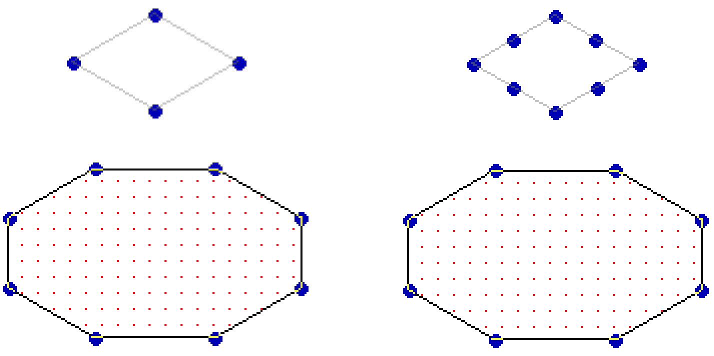
Please note:

You can increase the level of accuracy of how the transitional solid is displayed. Do this with the help of ***Process, Object - Display*** and by specifying parameters on the ***Solid facetting*** option card.

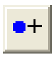
Transitional solid: add and remove points

After you have specified the two initial faces, the program will determine the endpoints of 2D straight lines or solid edges and will indicate which they are by markers. These points are compulsory and so cannot be deleted.
In the case of 2D lines and solid edges that are not straight the program will not determine the compulsory points.

You cannot complete a transitional solid unless there is **the identical number of points** on the lines forming the perimeters of the two faces. In the illustration on the left you would have to insert four additional compulsory points on the perimeter of the face. Then you would have the same number of compulsory points as shown in the illustration on the right:

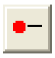


Inserting points:

1.  Activate the insert mode by clicking the button shown here.
2. Drag the cursor to the 2D line or the solid edge that you want to deal with.
   A marker will then appear on the object to indicate the position. The ***Position*** will also be shown in the Status dialog box by means of a ratio of between 0 and 1.
3. Type in a ratio specifying the position for the point you want to insert. This new point will be indicated by a marker.

Add more points on the perimeter of one of the faces by repeating Step 2 and Step 3.

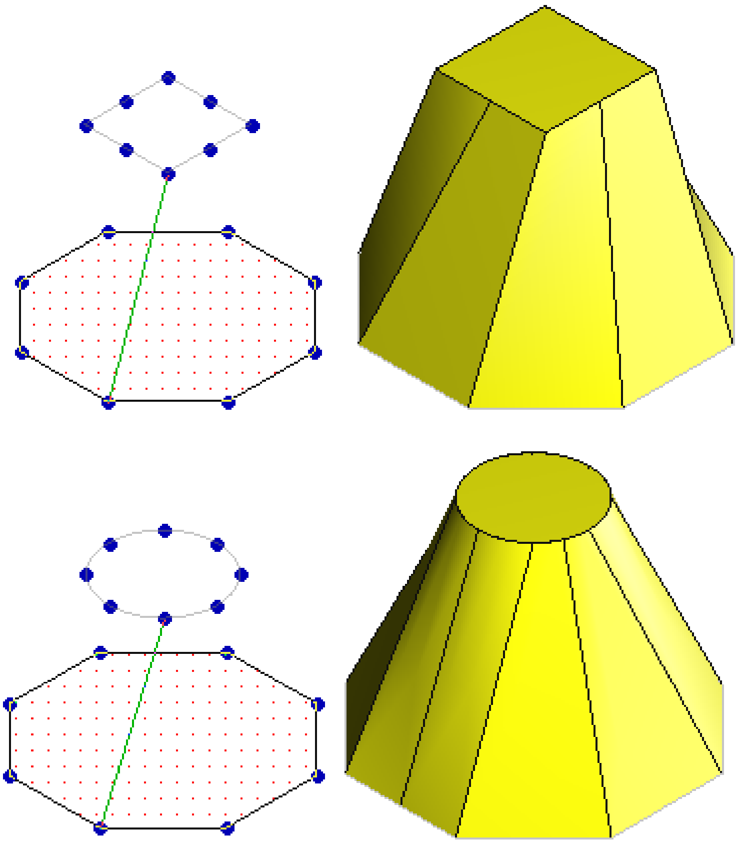
Deleting points:

1.  Click the button shown here.
2. Identify whichever point you want to delete.

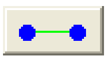
Transitional solid: define type of connection

The type of connection between points that is used will determine the way in which a transitional solid is generated:

Point-point connection:

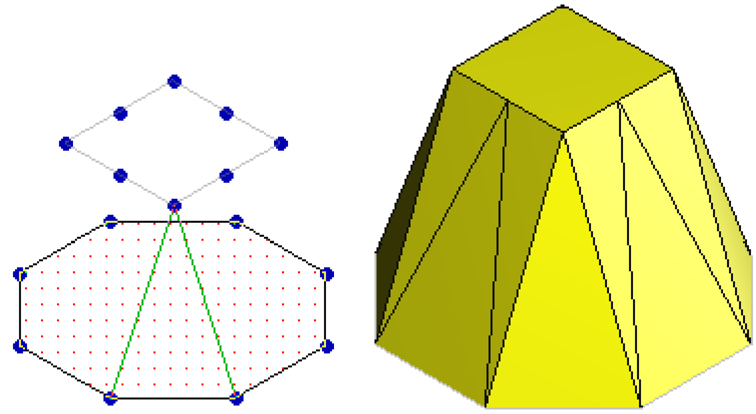


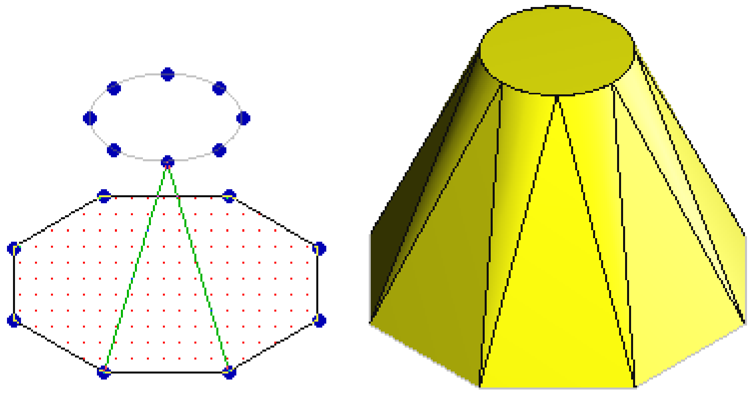
With this type of connection a point on the perimeter of one of the faces will be paired with a point on the perimeter of the other face. The program starts from this connection to determine the positions of the remaining compulsory points and connects them with solid edges.

1. Click the  button and then identify one of the compulsory points on one of the two perimeters.
2. Identify the compulsory point on the second perimeter to determine the way the connection is to be completed.

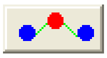
As far as the geometry permits the program will generate a transitional solid. If this step is impossible, an error message will be displayed.

Triangular connection





This procedure will produce a connection between two points on the perimeter of one of the faces with a corresponding point on the perimeter of the second face. Starting with this connection the program will then determine further triangular connections before generating the final transitional solid.

1. Click the  button and then identify one of the compulsory points on one of the perimeters. This point is the initial point for completing the triangular connection.
2. Identify a compulsory point on the second perimeter.
3. Identify a compulsory point on the first perimeter. This point serves as the endpoint of the triangular connection.

As far as the geometry permits the program will generate a transitional solid. If this step is impossible, an error message will be displayed.

Generate hollow solid

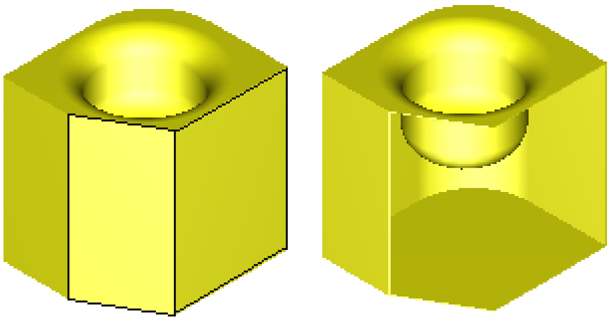
 Alternatively, the command can be loaded using this icon.

This command generated thin walled hollow solids from existing fixed solids:

Starting at the outer area of the solid, the material of the wall thickness is added towards the inside or towards the outside. If required, the solid can be "opened" at defined areas.

The suggested value for the wall thickness can be specified using the menu command ***Settings, 3D/2D Commands*** on the ***Solid*** option card.

1. Identify the solid from which a hollow solid is to be generated.
2. If a solid is to be "opened" in the area of one or more of its faces (n the illustration on the left the two enclosed faces), proceed as follows:



   Identify the faces to be removed or identify the faces which are to be retained and reverse the selection by pressing the  icon of the context dialog box.
3. In the ***Thickness*** text box, enter the value for the wall thickness for the remaining faces:
   With negative values, the walls are within the initial solid, with positive values, the walls are outside the solid.
4. Select the ***Preview*** button to get a preview of the resulting solid.
   - Click the ***right mouse button*** if you want to get back to the face selection.
   - Click the ***left mouse button*** or ***Accept*** to create the hollow body.

If the hollow solid cannot be generated, an appropriate error message will be displayed.

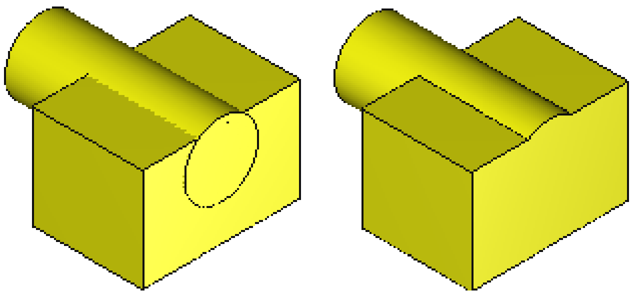
Please note:

The feasibility of this command depends on a number of factors: from the entered wall thickness and from the size and position of the solid faces to another.
Therefore, no generally valid statements can be made concerning the restrictions.

Unite fixed solids

 Alternatively, the command can be loaded using this icon.

This command can be used to unite a solid with one or more other solids, i.e. the resulting solid contains the entire volume of all individual solids e.g:



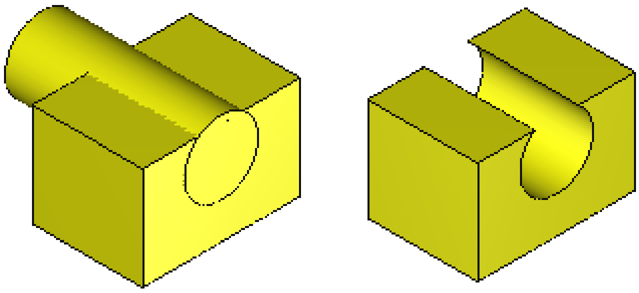
1. Identify the first solid.
2. Define one or more solids whose volume is to be united with that of the first solid, by selection using identification, dragging a window or by collection keeping the *Ctrl key* pressed.
   The selection can contain the solid identified in step 1.
   The 3D selection mode also supports you to make sure that you identify the correct solid.

The display characteristics of the new solid correspond to those of the first identified solid.

Subtract fixed solids

 Alternatively, the command can be loaded using this icon.

From a solid, this command subtracts the volume of one or more solids. This means that a "tool" volume is removed from a "work piece" e.g.:



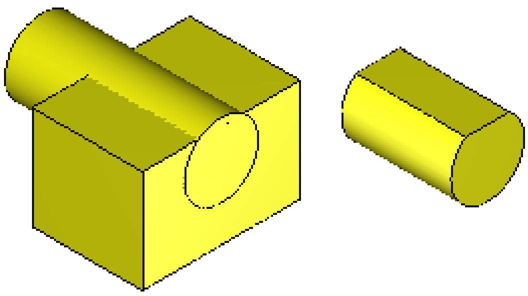
1. Identify the "work piece“ to be processed, therefore the solid from which volume is to be removed.
2. Define the "tool", i.e. one or more solids whose volume is to be removed from the first solid, by selection via identification, dragging a window or by collection by keeping the *Ctrl key* pressed.
   The selection can contain the solid identified in step 1.
   The 3D selection mode also supports you to make sure that you identify the correct solid.

The display characteristics of the resulting solid correspond to those of the first identified solid.

Form intersection of fixed solids

 Alternatively, the command can be loaded using this icon.

This command can be used to form the intersection of one fixed solid with one or more other fixed solids, i.e. the resulting fixed solid only contains the volume which is common to all respective solids, e.g.:



1. Identify the first solid.
2. Define one or more solids with which an intersection should be formed, by selection via identification, dragging a window or by collection by keeping the *Ctrl key* pressed.
   The selection can contain the solid identified in step 1.
   The 3D selection mode also supports you to make sure that you identify the correct solid.

The display characteristics of the resulting fixed solid correspond to those of the first identified fixed solid.

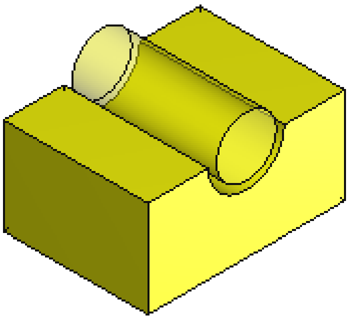
Subtracting Solids and obtaining tools

If you want to subtract two solids, without removing the "Tools" proceed as follows:

1. ,  Select the function ***Fixed solid, Subtraction/Keep tools***.
2. Identify the "workpiece" as the basic solid, i.e. the solid, from which you wish to subtract.
3. Identify the "workpiece" as the basic solid, i.e. the solid, which is to process the basic solid.

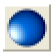
One or several "workpieces" are subtracted from another solid. The

"workpieces" will remain. Example:



Blend and chamfer fixed solids

from ***BeckerCAD 3D***

  Alternatively, the command for generating fixed solid blends and chamfers can be started using this icon

This command can be used to blend or chamfer fixed solids. This means that blended and chamfered areas are generated which connect sections of these solids. With this, material is added or removed.

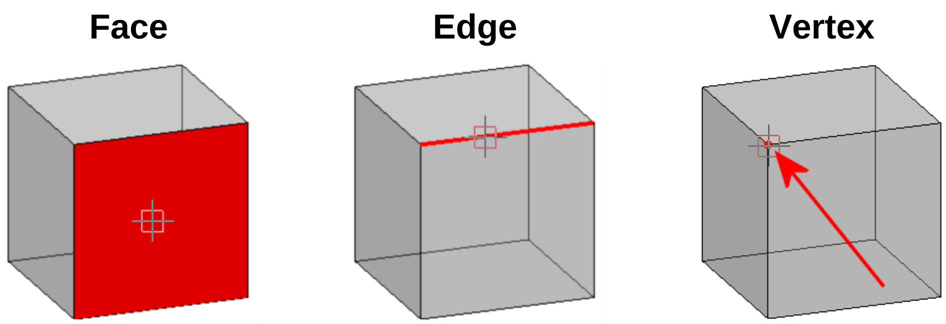
The command contains two different, separately usable execution variations which can be selected on both option cards:

Edges

The options on this option card can be used to **blend** and **chamfer** individual or a number of edges or an edge sequence of joining faces.

Entities

Every **Face**, **Edge** and **Vertex** of a solid is referred to as **Entity**.



The options on this option card can be used to **blend** two faces, two edges or a face and an edge.

With this kind of blending, the following steps are generally executed:

- Selection by the user of the section which is to connect the blend face.
- Determination by the user of the blending specific data, e.g. blending radius.
- If a blending radius with a predefined radius is to be inserted, e.g. between two faces, a sphere is positioned in a starting point so that it touches both faces at one contact point.
- The sphere is rolled along the face so that the contact to both faces is retained. A part of the cover developed in this way is inserted in the model as a blending face along the contact run.
- In order to retain a "closed" solid, the blending radius and, if necessary, the neighbouring faces are enhanced and blended together.

Please note:

If a blending can be generated with both execution variations, use the edge variant since it is faster and more stable.

Specify settings in order to blend and chamfer fixed solids

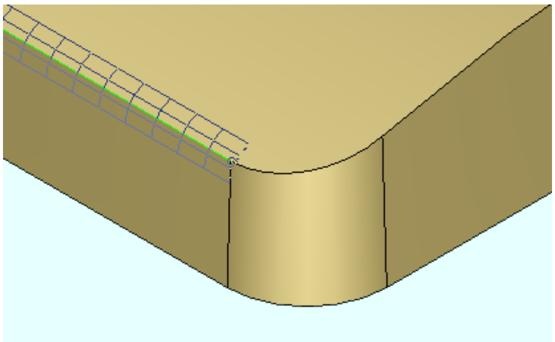
Using the menu command ***Settings, 3D/2D Commands***, the ***Solid*** option card can be used to determine presettings for the ***Fixed Solid, Blend/Chamfer*** command.
All these presettings can be modified after the command is loaded.

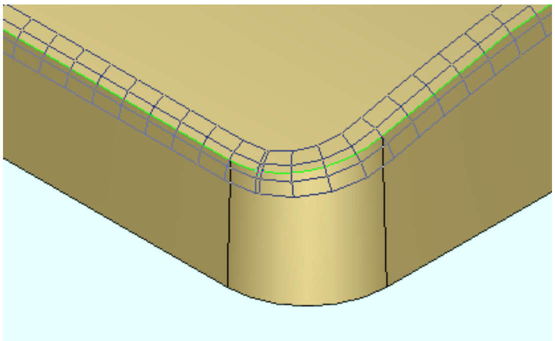
In the first three fields, values should be entered which are used in most cases for blends and chamfers, so that these values only need to be modified in the Status dialog box in seldom cases.

blend radius
Defines the suggested value for the blend radius.

distance
Defines the suggested value for the right and left distance of an edge to be chamfered.

face indent
Defines the value, with which special corners should be processed, if all the edges which collide are selected.
This entry only applies to the rounding or chamfering processes using the options on the ***Edges*** card.



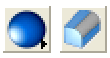


select edges sequentially

If this setting is active, when picking individual edges, all edges are also selected which connect tangentially to this edge and which also have tangential transitions.
If this is not active when picking an edge, simply this edge is selected.
This setting applies only for rounding or chamfering using the options on the ***Edges*** tab.

Blend fixed solids (edge)

from ***BeckerCAD 3D***

 Alternatively, the command can also be started using this icon.

This command can be used to insert blending faces between neighbouring faces of a fixed solid. The definition of the area which is to be blended is carried out by selecting individual edges or edge sequences.

General procedure:

1.  Select the ***blend*** button on the ***Edges*** by clicking on the icon shown.

2. The values set using the menu command ***Settings, 3D/2D Commands*** are then displayed in the ***Radius*** text box in the Status dialog box for the rounding radius.

   If required, enter another standard value for the radius.

3. In the dialog box, select the options for the selection of edges:

   face 
   The edges to be blended are selected by identifying solid faces. All edges bordering such a face should be blended.

   edge 
   The edges to be blended are selected individually or collected in an action list.

   sequential
   If this setting is active, when using the ***edge*** option, an edge sequence is selected:
   All edges are determined which connect tangential to one of the identified edges and also have tangential transitions within them.

4. According to the set options, determine the edges to be blended. Guidance on selecting edges is given in the following section.

5. If individual edges of those edges selected are to be blended completely with a radius which deviates from the predetermined standard radius, enter the required value in the Radius text box. Position this radius on the edges by positioning the cursor on each of these edges and pressing the left mouse button.

   If some individual edges of those selected are to be blended with different radii, define the ***Radius*** first for each of these edges and then the ***Position*** (see below).
   Guidance on specifying a radius and a position is given in the following section but one.

   All edges for which no radius is entered, are blended by the radius value which was displayed in the status dialogue window before clicking the ***Accept*** (Point 7.) button.

6. If all edges are selected which run together into one corner, additional face indentations can be defined for special processing of this corner.
   In order to do this, select each one of these edges one after the other, and according to the position of the corner on the edge, enter the radius value (for position 0 in the ***Indent 0*** text box, for position 1 in the ***Indent 1*** text box). The value for the indentation

must be larger than the radius so that the indentation can be generated.

7. The blend(s) are then carried out using the ***Accept*** command.

Notes concerning the selection of edges:

edge radio button active:

- Select an individual edge
  Selected by identifying with the cursor.
- Select a number of edges
  Selected by "collection", i.e. When you are identifying the edges (up to the last) keep the *Ctrl key* pressed.
  Collection is finished by releasing the *Ctrl key* and then identifying another edge.
- Whilst selecting edges, by pressing the *Shift key*, the current setting for ***sequential*** can be "outvoted" and the opposite mode can be activated.

face button active:

- Selects all edges, which border a face
  With a shaded display of the solid, identify the face or with edge display identify two of the edges which border the face.
- Selects all edges which border a number of faces
  The faces are selected by "collection", i.e. whilst identifying the faces or their edges (up to the last), the *Ctrl key* is kept pressed.
  Collection is finished by releasing the *Ctrl key* and then identifying a further face or one of their edges.

 When collecting edges or faces of the solid using the command for dynamic rotation or zoom, release the *Ctrl key* whilst doing so. Press again after the view is determined.

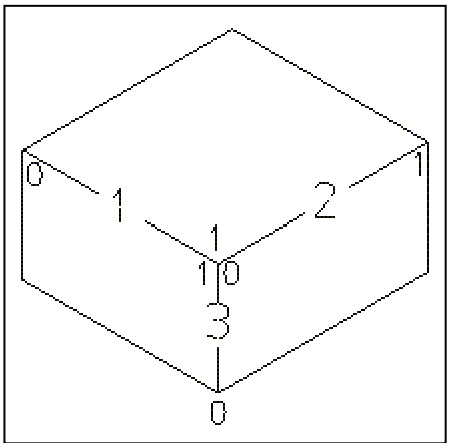
Notes concerning determination of position and radius

- If an edge (sequence) is to be blended with different radii, at least the radii on the endpoints of the edge must be defined (= position 0 and position 1).
- Enter the value for the radius <u>before</u> the value for the position is entered. Both values must be confirmed with *Enter*.
- For the radius - apart from the positions on the endpoints of the edges - only values larger than 0.0 are permitted.

- If the cursor is on an edge for which the position and blend radius are to be entered in the Status dialog box, by pressing the *Shift key*, the cursor is retained on the selected edge.

Blend fixed solids (edge) - examples

from ***BeckerCAD 3D***



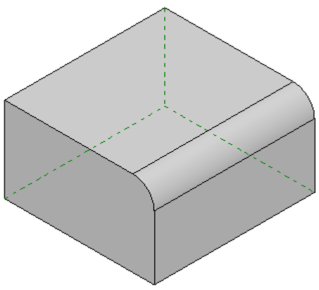
Origin solid with edges 1, 2, 3 and the positions 0 or 1 on the endpoints of each edge.
The value of the position on one edge endpoint is displayed in the ***Position*** text box of the Status dialog box.
It depends on the orientation of the face, so therefore does not always correspond to the entries in the following examples.

30 is taken as the preset value for the blend radius. This can be defined using ***Settings, 3D/2D Commands*** on the ***Solid*** option card.

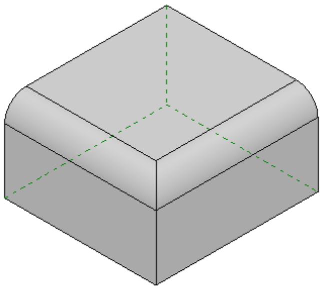
Always confirm the entries in the Status dialog box with Enter.

Possible results of blend:

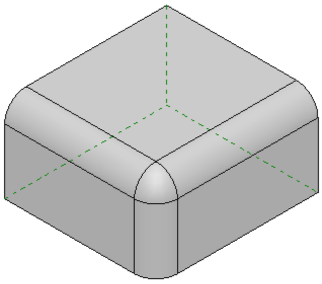
1. Enter *Radius* 10, identify edge 2, *Accept*



2. Enter *Radius* 10, collect edge 1 and 2, *Accept*

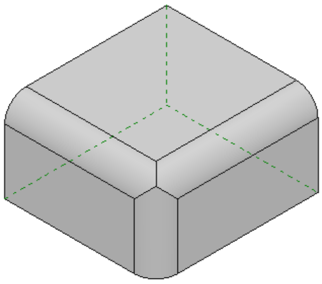


3. Enter *Radius* 10, collect edge 1, 2 and 3, *Accept*

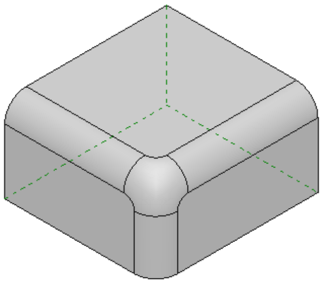


4. Enter *Radius* 10, collect edge 1 and 2, *Accept*.
Deactivate *sequential*, identify edge 3, *Accept*.

If the *sequential* radio button is active whilst identifying edge 3, the results are identical to example 3.

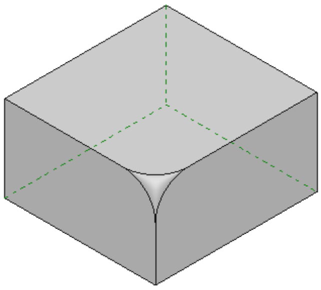


5. Enter *Radius* 10, collect edge 1, 2 and 3, on edges (1, 2, 3) for the common corners in the text boxes (*Indent 1*, *Indent 0*, *Indent 0*) enter the value 15.0, *Accept*.



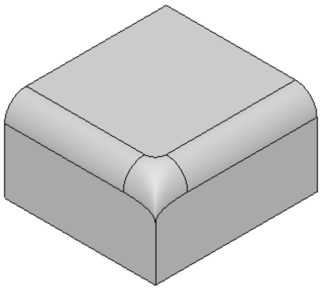
6. Enter *Radius* 0, collect edge 1, 2 and 3, position cursor on each edge and release the radius at any position with a mouse click. On edges (1, 2, 3) for the common corners in the text boxes

(*Indent 1*, *Indent 0*, *Indent 0*) enter the value 15.0, *Accept*.

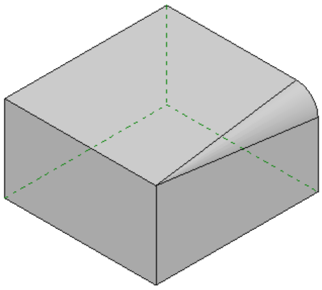


7. Enter *Radius* 10, collect edge 1, 2 and 3. Release on required position by clicking the mask on edge 1 and 2, enter *Radius* 0, release any on edge 3.

On edges (1, 2, 3) for the common corners in the text boxes (*Indent 1*, *Indent 0*, *Indent 0*) enter the value 15.0, *Accept*.

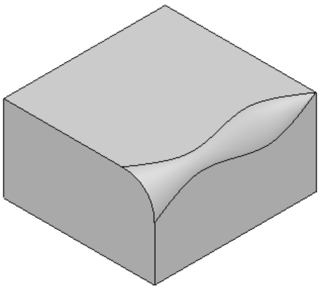


8. Identify edge 2, enter *Radius* 0, enter *Position 0*, enter *Radius* 10, enter *Position 1*, *Accept*.



9. Identify edge 2.
Enter *Radius* 15, enter *Position* 0,
enter *Radius* 5, enter *Position* 0.3,
enter *Radius* 10, enter *Position* 0.7,
enter *Radius* 0, enter *Position* 1,

Accept.



Blend fixed solids (face - face)

from ***BeckerCAD 3D***

This type of blending is generated by rolling a sphere along two faces of the same solid.
The tracks of their contact points with both faces, define the area within which the blending is carried out. The limit for blends, that they need joining faces is <u>not relevant</u> for the Entity-Entity blend.

General procedure:

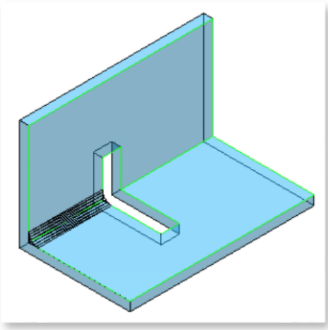
1.  In the displayed dialog box, activate the ***Identify faces*** setting in the ***Entities*** option card, by clicking on the icon shown.

2. With shaded representation, identify the two faces of a solid, between which the blending is to be inserted, with edge representation identify two of the edges which border these faces. With this, all faces are determined which connect tangentially to the identified faces.

3. In the ***Radius*** text box of the Status dialog box, enter the value for the radius. This value is valid constantly for the entire blending.

4. ,  By clicking on one of these icons, determine whether the blending faces inserted between the selected faces are to be convex or concave.

5.  If, from the previous entries, the position of the blending is not clearly defined, e.g. by interruption one of the contact tracks (see illustration), click on the displayed icon.
Then define a point near the section in which the blending is to be carried out.



6. The blending is bordered where one of the contact tracks meets a sharp edge.

    If the blending is to be enhanced over this border, therefore the sphere is rolled further, click on the displayed icon for the roll-on command.

    Then, using this icon, select that the roll-on command is to be set on an edge.

   Identify the edge on which the roll-on command should be valid.

7. If a further face is tangent to one of the faces, this face is also included in the process.

    If this is not required, define this neighbouring face as the border face, by setting a cap command on it. To do so, click on the displayed icon.

    Then select the displayed icon in order to be able to select a face.

   Identify the face for which the cap command is to be valid.

8. Click on the ***Accept*** command button in order to execute the blending.

Please note:

The feasibility of the command depends on a number of factors. For example, the blending radius can be selected too large or that of the faces bordering the blending face cannot be enhanced in order to generate a "closed" solid.
If the blending cannot be executed, an appropriate error message will be displayed.

Blend fixed solids (face - edge)

from ***BeckerCAD 3D***

This type of blending is generated by rolling a sphere with its outer face along a face and an edge of the same solid.
The edge and the contact track on the face defined the section within which the blending is to be executed.

General procedure:

1. In the displayed dialog box, activate the ***Identify faces*** setting in the ***Entities*** option card, by clicking on the icon shown.
2. With shaded representation, identify the face, with edge representation of two edges, identify those which border this face. With this, all faces are determined which connect tangentially to the identified faces.
3. Activate the *edge* option.
4. Identify the edge.
With this, all edges are determined which connect tangentially to the identified edge.
5. In the ***Radius*** text box of the Status dialog box, enter the value for the radius. This value is valid constantly for the entire blending. It must be larger than the distance between face and edge.
6. , By clicking on one of these icons, determine whether the blending face is to be convex or concave.
7. If, from the previous entries, the position of the blending is not clearly defined, e.g. by interrupting the contact track, click on the displayed icon.
Then define a point near the section in which the blending is to be carried out.
8. The blending on the edge is bordered by corner points, on which an edge connects without a tangent transition on the face where the contact track meets a sharp edge.

   If the blending is to be enhanced over this border, therefore the sphere is rolled further, click on the displayed icon for the roll-on command.

   Using the displayed icon, select that the roll-on command is to be set on a corner point (node).

Identify the corner point on which the roll-on command is to be valid.

7. If a further edge is tangent to one of the edges, this edge is also included in the process.

 If this is not required, define this neighbouring edge as the border edge, by setting a cap command on it. To do so, click on the displayed icon.

 Then select the displayed icon in order to be able to select an edge.

Identify the edge for which the cap command is to be valid.

8. Click on the ***Accept*** command button in order to execute the blending.

Please note:

The feasibility of the command depends on a number of factors. For example, the blending radius can be selected too small or that of the faces bordering the blending face cannot be enhanced in order to generate a "closed" solid.
If the blending cannot be executed, an appropriate error message will be displayed.

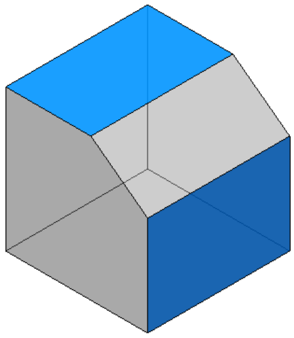
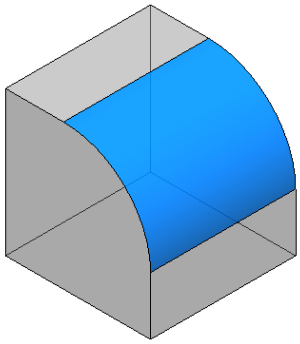
Blending of Fixed Solids (Entity - Entity) - Examples

from ***BeckerCAD 3D***

Below find some examples for the Entity-Entity blending, where the preferred edge blending cannot be used. Both examples illustrates, that the faces to be blended must not join, but they must be part of the same solid.

<u>Example Face / Face (convex)</u>

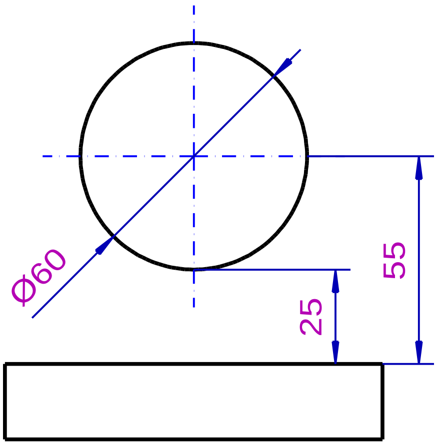
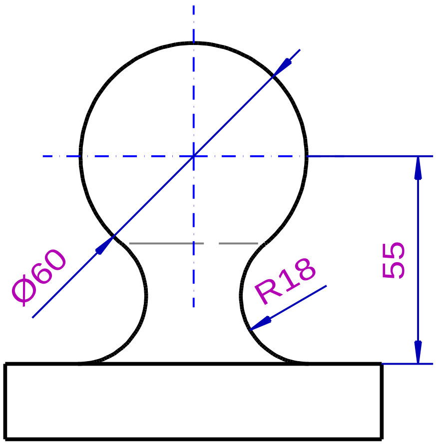
The two following examples illustrates, that the faces to be blended must not join, but they must be part of the same solid.

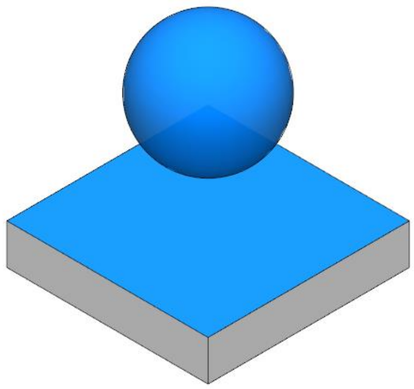
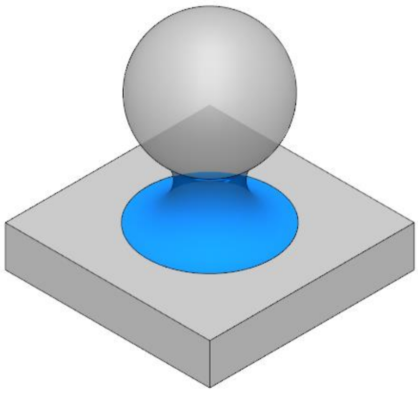
Original with chamfer 20x15mm	Blending Face / Face with 30mm radius
	

Proceedings:

1.  On the tab *Entities* select the option *Face Selection*.
2. Identify both marked faces of the solid in shaded display.
3. Enter in the field Radius in the status dialogue window: 30 ↲
4. Select in the dialogue window: Blending Convex
5. Accept

<u>Example Face / Face (concav)</u>

Original Block and Sphere united	Blending Face / Face with 18mm radius
	

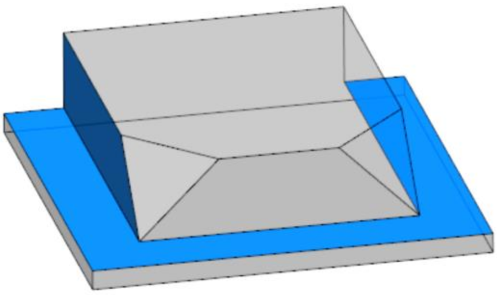
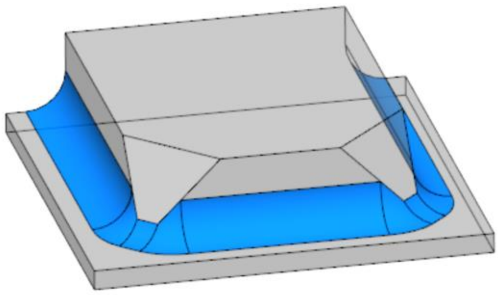
	

Proceedings:

1.  On the tab *Entities* select the option *Face Selection*.
2. Identify the top face of the block and the sphere face in shaded display.
3. Enter in the field Radius in the status dialogue window: 18 ↵
4.  Select in the dialogue window: *Blending Concav*
5. Accept

<u>Example with changing (Face/Face - Face/Edge)</u>

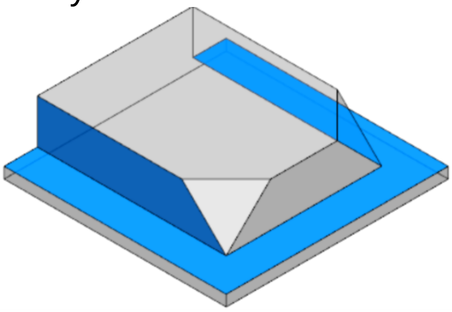
In this example, the Entity mode will change several times between Face/Face and Face/Edge. The blending will be created with a rolling sphere. If the sphere hits an edge, the blending would stop. Using a rollon command, the sphere runs across the edge to the next face and so on.

Original	Blending Face / Face with 10mm radius
	

Proceedings:

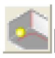
1.  On the tab *Entities* select the option *Face Selection*.

2. Identify the both marked faces in shaded display.



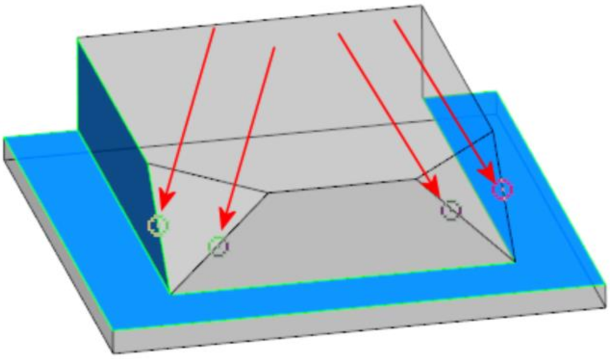
3. Enter in the field Radius in the status dialogue window: 10 ↲

4.  Select in the dialogue window: *Blending Concav*

5. Define *Rollon Command* for the sphere.

6.  Activate the option *Edge Selection*.

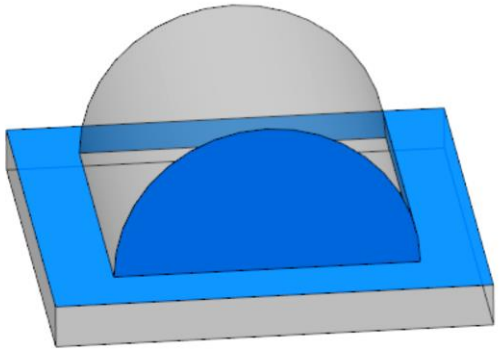
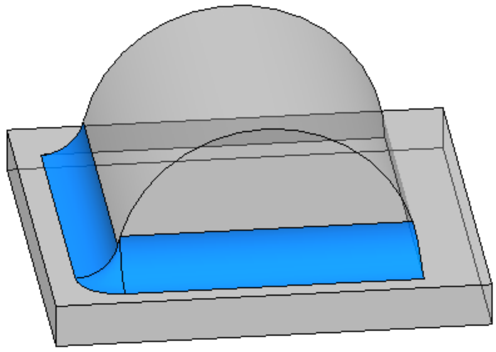
7. Identify the edges for the rollon command for the rolling sphere.



8. Accept

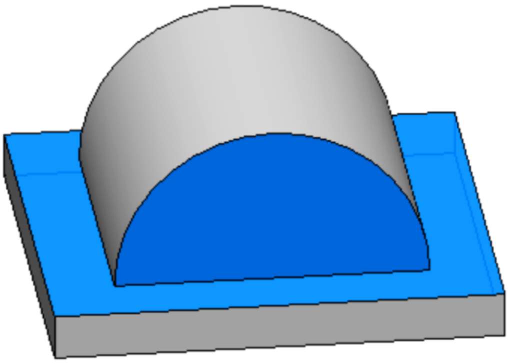
<u>Example for multiple commands on one single edge</u>

A circular edge hits the same face twice. The blending should proceed only on one part of the circular edge with a rollon command. The other part of the circular edge will get a cap command to stop the blending at the end of the edge.

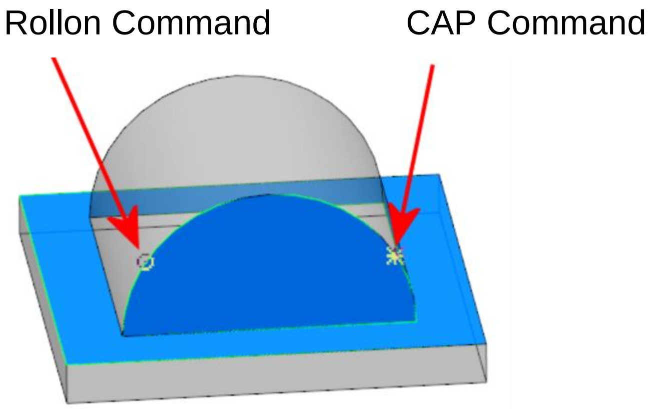
Original	Blending Face / Face with 10mm radius
	

Proceedings:

1.  On the tab *Entities* select the option *Face Selection*.
2. Identify both marked faces in shaded display.



3. Enter in the field Radius in the status dialogue window: 10 ↲
4.  Select in the dialogue window: *Blending Concav*
5.  Define *Rollon Command* for the sphere.
6.  Activate the option *Edge Selection*.
7. Identify the edges for the rollon command for the rolling sphere.



8.  Activate the *CAP command* in the dialogue window.
9. Identify the other side of the circular edge for the cap command. This will stop the rolling sphere not to pass this part of the edge.
10. Accept

Chamfer fixed solids

from ***BeckerCAD 3D***

  Alternatively, the command can be started using this icon.

This command can be used to insert chamfered faces between neighbouring faces of a fixed solid. The definition of the area which is to be chamfered is carried out by selecting individual edges or an edge sequence.

General procedure:

1.  In the displayed dialog box, select the ***chamfer*** button on the ***Edges*** option card, by clicking on the icon shown.
2. In the ***Left dist.*** and ***Right dist.*** text boxes, the values determined using the menu command for the distance of edges are displayed. The directions "right" and "left" can be specified, by looking from the edge position 0 along the respective edge.

   The chamfer value can be defined by two distances or by one distance and the corresponding angles.
3. In the dialog box, determine the options for the selection of edges:

   face 
   The edges to be chamfered are selected by identifying solid faces. All edges bordering such a face should be chamfered.

   edge 
   The edges to be chamfered are selected individually or collected in an action list.

   sequential
   If this setting is active, when using the ***edge*** option, an edge sequence is selected:
   All edges are determined which connect tangential to one of the identified edges and also have tangential transitions within them.
4. According to the settings, determine the edges to be chamfered. Guidance on selecting edges is given in the next section.

5. If individual edges of those edges selected are to be chamfered completely with values which deviate from the predetermined value, enter the required value in the ***Right dist.*** and ***Left dist.*** text boxes or by one distance and the corresponding angles. Position these values on the edges by positioning the cursor on each of these edges and pressing the left mouse button.
All edges, for which no values have been entered are blended by the values which were active when finishing edge selection.

Guidance on specifying distances and positions is given in the next section but one.

6. If all edges are selected which run together into one corner, additional face indentations can be defined for special processing of this corner.
In order to do this, select each one of these edges one after the other, and according to the position of the corner on the edge, enter the radius value (for position 0 in the ***Indent 0*** text box, for position 1 in the ***Indent 1*** text box).

7. The chamfer(s) are then carried out using the ***Accept*** command.

Notes concerning the selection of edges:

edge radio button active:

- Select an individual edge
  Selected by identifying with the cursor.

- Select a number of edges
  Selected by "collection", i.e. When you are identifying the edges (up to the last) keep the *Ctrl key* pressed.
  Collection is finished by releasing the *Ctrl key* and then identifying another edge.

- Whilst selecting edges, by pressing the *Shift key*, the current setting for ***sequential*** can be "outvoted" and the opposite mode can be activated.

face button active:

- Select all edges, which border a face
  With a shaded display of the solid, identify the face or with edge display identify two of the edges which border the face.

- Select all edges which border a number of areas
  The faces are selected by "collection", i.e. whilst identifying the faces or their edges (up to the last), the *Ctrl key* is kept pressed.
  Collection is finished by releasing the *Ctrl key* and then identifying a further face or one of their edges.

 When collecting edges or faces of the solid using the command for dynamic rotation or zoom, release the *Ctrl key* whilst doing so. Press again after the view is determined.

Notes concerning determination of position and distances

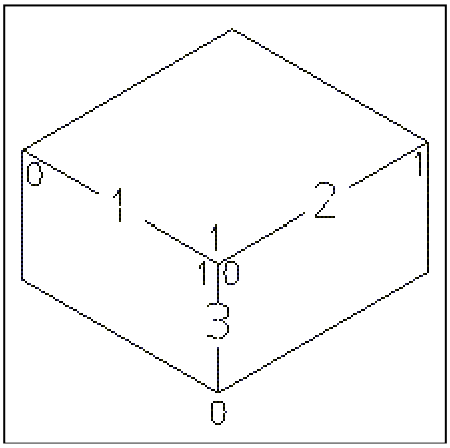
- If the cursor is positioned on an edge, for which the distances are to be entered in the Status dialog box, the cursor is retained on the selected edge by pressing the *Shift key*.
- On each edge, only constant chamfers can be executed. Changing chamfers along one edge are not possible.

Please note:

On one edge only a chamfer with one defined value can be defined. Changing chamfers cannot be defined.

Chamfer fixed solids - examples

from ***BeckerCAD 3D***

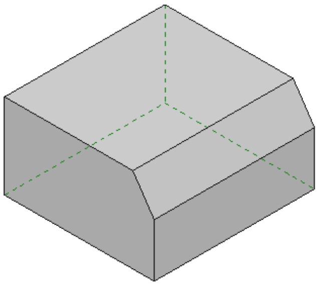


Origin solid with edges 1, 2, 3 and the positions 0 or 1 on the endpoints of each edge.
The value of the position on one edge endpoint is displayed in the *Position* text box of the Status dialog box.
It depends on the orientation of the face, so therefore does not always correspond to the entries in the following examples.

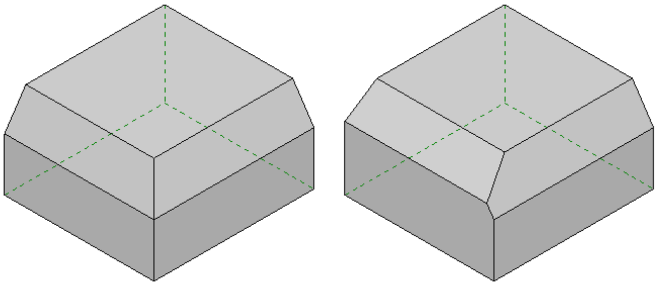
30 is taken as the present value for the chamfer radius. This can be defined using ***Settings, 3D/2D Commands*** on the ***Solid*** option card.

Always confirm the entries in the Status dialog box with *Enter*.

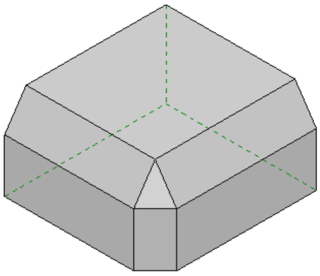
1. Identify edge 2, *Left dist.* 10, enter *Right dist.* 15, *Accept*.



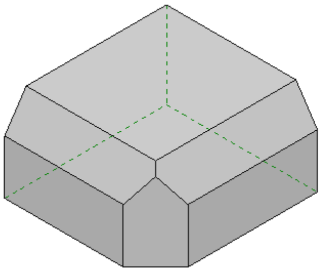
2. Collect from edge 1 and 2, enter *Left dist.* 10, *Right dist.* 15, *Accept*.
   Take note of the orientation from *Position 0* to *Position 1* with different distances. Exchange the distances as well if necessary. Otherwise, other results are reached.



3. Collect edges 1, 2 and 3, enter *Left dist.* 10, *Right dist.* 15, release on edges 1 and 2 by clicking the mouse.
   Enter *Left dist.* 10, *Right dist.* 10, release on edge 3 by clicking the mouse, *Accept*.

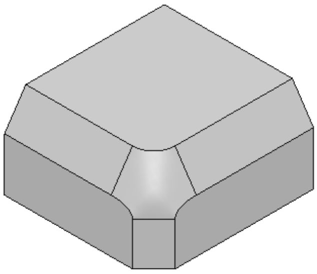


4. Collect edge 1 and 2, enter *Left dist.* 10, *Right dist.* 15, *Accept*.
   Enter *Distance* 15, identify edge 3, *Accept*.



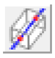
5. Collect edge 1, 2 and 3, *enter Left dist.* 10, *Right dist.* 15, release on edges 1 and 2 by clicking the mouse.

Enter *Left dist.* 10, *Right dist.* 10, release on edge 3 by clicking the mouse.
On edges (1, 2, 3) for the common corners enter the value 20.0 in the text boxes (*Indent 1*, *Indent 0*, *Indent 0*), *Accept*.



Please note:
A chamfer can only be created on an edge with a constant value.

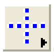
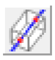
Allocate fixed solid with axes

 In order to allocate a fixed solid with axes, use the displayed icon in the 3D toolbar or the command *Solid Axes* in the CAD menu.

You can use solid axes to define permanent or temporary WPs by selecting the ***Face/Edge/Axis*** command in the Snap Menu and identifying the axis close to one of its endpoints.

You can also use the axis of a solid to orientate solids when executing ***Transform, Reposition*** or to move solids dynamically along an axis using ***Transform, Display and Process Geopoints***.

Define the axis of a solid in the following way:

1. Using ***Settings, 3D/2D Commands*** on the ***2D/3D axes*** option card, determine the settings for the projection, colour and line type of the axes.
2. ,  Activate the ***Centre Lines, Define 3D Axes*** check box.
3. In the ***Overhang*** text box of the Status dialog box, modify the suggested value by which the axes is to be projected over both endpoints.
4. Define the first point of the axis.
   The point is a spatial point, therefore is independent of the current WP.
5. Define the second point of the axis.

Axes generated by using points of a solid (endpoint of an edge, centre of gravity of a face, etc.) are linked with this solid like attributes, whereby the link is only valid for the current model and cannot be

transmitted with the solid to other models..
This means that whether solids or axes are selected, axes and solids are processed in the same way whilst being transformed.

Please note:

The endpoints of axes defined in this way are only on the visible endpoints if the projection has the value of 0. Otherwise they lie on the lines away from the visible edges on the value of the projection.

If no solid axes are to be displayed in the 3D window, activate the menu command ***Settings, 3D/2D Commands*** and change to the ***2D/3D axes*** option card.
Deactivate the ***display 3D axes*** check box.

The display of 3D axes can be processed using the menu command ***Process***, ***Object Display***. In the offered dialogue window ***Object Display*** the ***object colour***, a ***line type*** and a ***line width*** can be assigned ***object*** (grey) or ***layer oriented*** (colour).
In the field layer it is displayed, in which layer the 3D axis is contained. Using the iconl the layer selection dialogue window is offered, to move the 3D axis on another layer.
If the selected 3D axis have different colours, line types or line width, the respective fields displays the entry **<?>**.

5.7 Local Operations

from ***BeckerCAD 3D pro***

The icon bar ***Local Operations*** is offered by clicking the displayed icon.

It contains commands, to process fixed solids. The faces of solids are processed with the commands in this icon bar, e.g. move, taper and rotate faces.

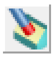
If a surface is identified after the start of the command, the command is executed dynamically, when no defaults are defined for the offset or the taper angle in ***Settings, 3D/2D Commands…***. Also a preview of the process is displayed, by a shaded body or edges. The settings for the preview can be made in ***Settings***, ***3D Display…***.

The default values for the ***local operations*** can be set in ***Settings***, ***3D / 2D Commands*** in the tab ***Solid***.

3D Surface Selection

from ***BeckerCAD 3D pro***

 After clicking the displayed icon, the icon bar ***Solid Local Operations*** is offered.

, 

It contains, among others, the displayed commands to move, rotate or delete the surfaces of fixed solids. After starting one of this commands or the command Solid Face Display or Transfer Solid Face Display the ***3D Face Selection*** dialogue window is offered, to define the settings for the selection of single or several surfaces.

Selection

process

The selection of the surfaces can be made for a single surface or, with the pressed ***Ctrl*** key, for several surfaces. After the selection of the last surface (release ***Ctrl*** key) the command to edit the surfaces starts immediately.

confirm

Several surfaces can be identified after each other. The selection command is finished by clicking the ***Accept*** button. Then the previously selected command is started.

Faces

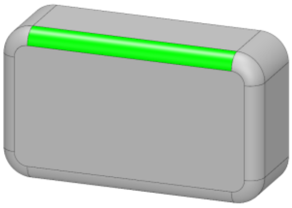
Here it can be defined if the selected surfaces should be ***added*** to the list of surfaces or ***removed*** from the list.

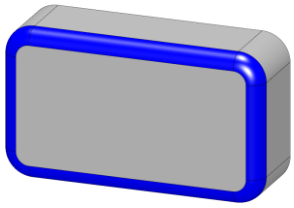
Selection mode

Within the selection mode a filter helps to identify the desired surfaces. You can toggle between the different filters during the selection by clicking the ***X*** key.

Single face

Identify single surfaces



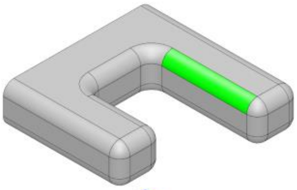


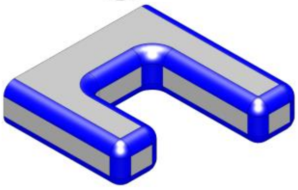
Blend sequence

When Identifying single blends also all other blends with identical radius are selected, which are tangentially connected to the identified one and to each other. The sequence must be unique without different branches.

Blend net

When Identifying single blends also all other blends with identical radius are selected which are connected to each other.



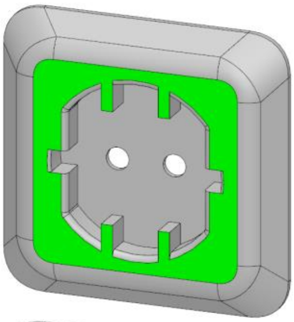


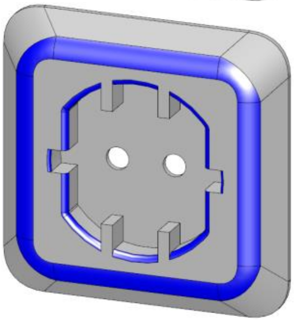
Adjacent blend

When Identifying single blends all connected blends are selected.

Protrusion

When Identifying a single surface all surfaces are selected which are connected convex to the selected surface and the connected ones. To finish the selection as the last surface of the protrusion also a concave surface can be selected.

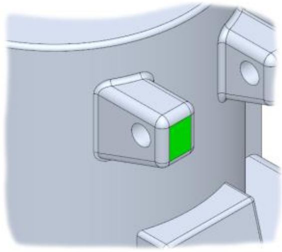


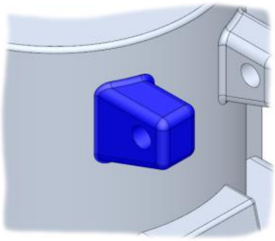


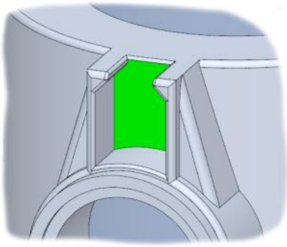
Depression

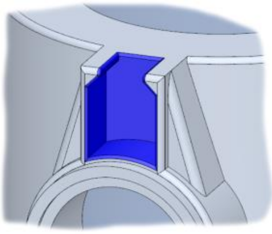
When Identifying a single surface all surfaces are selected which are connected concave to the selected surface and the connected ones. To finish the selection as the last surface of the depression also a convex surface can be selected.

All the selected surfaces within one command are listed in the lower part of the dialogue window. If you select one of the listed surfaces, it is highlighted in the graphic with a highlight color.
Click the ***Erase*** button to delete tagged surfaces from the list.









Pressing the Last selection button reloads the last selected faces into the dialog for editing.

Offset Face

from ***BeckerCAD 3D pro***

,  The command ***Local Operations***, ***Offset Face*** is started by clicking the displayed icons.

This command allows to define an offset to planar faces as well as to curved faces, so that the face will be moved.

1. For a dynamic preview activate a workplane, which is near to the face to be processed. The workplane should not have parallel alignment with the body surface, which you will first identify in the function.
2. Start the command by clicking the displayed icons.

3. If a default is defined and activated using the menu command ***Settings, 3D / 2D Commands*** in the tab ***Solid*** for the ***Offset*** this value is displayed in the field ***Offset*** in the status dialogue window.

   If a different value should be used, please enter. A positive value leads to an addition to the surface a negative value to an erosion.

   To get a dynamic preview of the process, please delete the value and confirm with Enter.

4. Identify one surface or collect surfaces with the pressed CTRL key.

5. If the value for the Offset is not 0, the selected surface(s) are moved if possible.

   If the value for the Offset is 0, it can be dynamically defined by moving the cursor or by entering a value in the field ***Offset*** in the status dialogue window.
   Please note: A dynamic preview cannot be displayed, if the first identified face is parallel to the current work plane.

6. Using the value defined, the ***Offset*** can be used for multiple faces.

Please note:

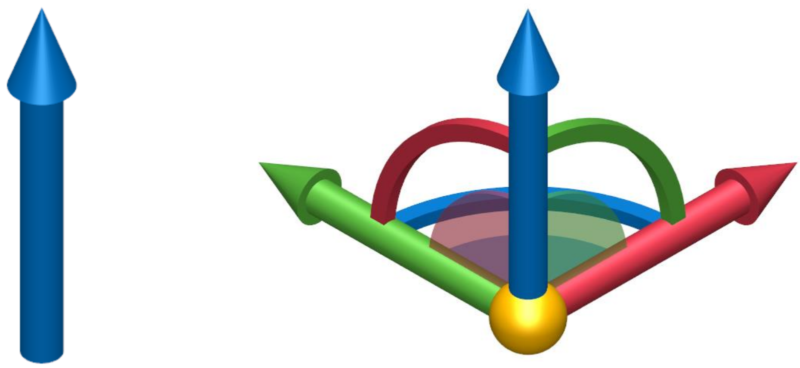
Settings for the display of the dynamic previews can be made using the menu command ***Settings***, ***3D Display …***.

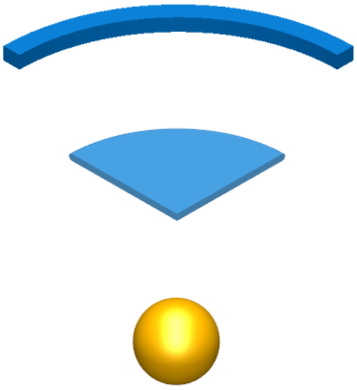
Transform Surfaces

from ***BeckerCAD 3D pro***

,  After clicking the displayed icon the command ***Solid Local Operations***, ***Transform Faces*** is started.

This command allows to move and/or rotate the surfaces of fixed solids. For the selection of the surfaces the ***3D Face Selection*** is used. A gadget is displayed on the last identified surface. Depending on the selected list of surfaces different transforming processes are offered by clicking on the different elements of the gadget.



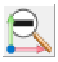
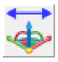


In the gadget click

- an arrow, to move the selection along the arrow.
- an arc, to rotate the selection around the arrow in the same color.
- a segment, to move the selection in this level.
- the sphere, to move the selection in the 3D space.

Dialog 3D Transformation / Key commands

When the surface selection is finished and the gadget appears, you can use either the buttons in the ***3D Transformation*** dialog or the following keyboard functions.

 	***Zoom in*** and ***out*** the gadget (Hotkey **+ / -**)
 	***Hide*** and ***show*** the gadget (Hotkey **Y**)
 	Move 3D gadget **with** selected surfaces (Taste **X**) Move 3D gadget **without** selected surfaces (Taste **X**)
B	***Enables*** / ***disables*** the previewing of the solid, e.g. to identify a vertex of the solid. If the preview is disabled the solid is displayed in the state before the current process.

Adjustable angle grid for rotation

You can use the menu command ***Settings***, ***3D/2D Commands*** on the ***Solid*** tab to set an ***angle grid for rotation*** for the 3D axis cross.

If this setting is activated, you can only rotate the 3D gadget after clicking on an arc at angles that are multiples of the entered value. This setting is ignored if the point definition menu is used to capture a point on an existing object, the status window is used to define a different angle, or the *Shift* key is held down.

General Method

1. Start the command by clicking the displayed icons.
2. Define the surfaces to be transformed using the ***3D Face Selection***.
3. The gadget appears on the last identified surface. Using the **+** or **-** key the gadget can be zoomed in or out.

   Move the gadget only

   To move the gadget to a defined point of the solid, without moving the selected surfaces, please process as follows:

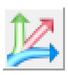
   3.1 By clicking the ***B*** key the dynamic preview is disabled. Only in this mode, points of the solid to be transformed can be identified.

   3.2  Click this button in the toolbar ***3D-Transformation*** or the ***X*** key to move the gadget without the selected surfaces.

   3.3 Identify an element of the gadget (arrow, arc, segment of sphere). Corresponding to the selected element the gadget can be moved or rotated.

   3.4 Define the new position of the gadget by point definition or by entering values in the status dialogue window..

   3.5 Click the ***B*** and ***X*** key again to enable the preview and to move the selected surfaces with the gadget again.
4. Identify an element of the gadget (arrow, arc, segment of sphere) according to your transforming needs.
5. Define the target position of the transformation by point definition or by entering values in the status dialogue window.

 **Align plane at 3D gadget parallel to surface**

This command allows to align a plane from the 3D gadget to a plane, cylindrical, spherical or conical surface parallel. You can also align a plane of the 3D gadget to a work plane. After you have started the command using the button shown, proceed as follows:

a) If necessary, enlarge the 3D axis cross by pressing the **+** key.

b)  At the 3D gadget, identify a colour on the *arrow*, *arc*, or segment of the *plane* you want to align.

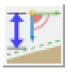
c) Identify a plane, cylindrical, spherical or conical surface to which the plane of the 3D gadget is to be aligned parallel. In the case of non-planar surfaces, the plane is aligned tangentially to the surface from the 3D gadget.

Alternatively, you can identify the frame of a work plane.

d) In the status window, the current distance between the plane of the 3D gadget and the identified surface is displayed and can be changed.

e) Finish the individual steps with the right mouse button.

Double-click the right mouse button to cancel the command.



Move along arrow with distance to surface

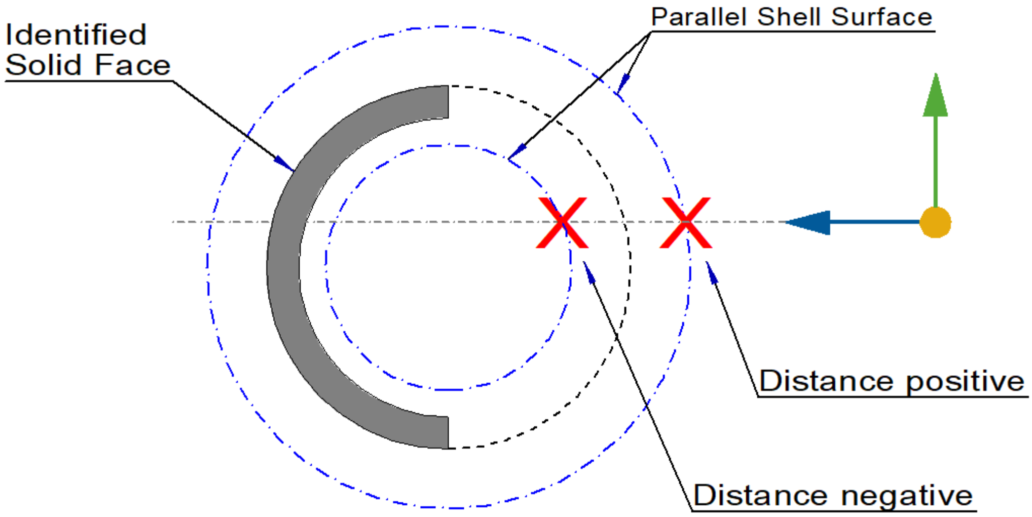
To move the point at which you have positioned the 3D gadget with a distance from a flat, cylindrical or conical surface, start the command using the button shown. The point is moved along an axis to be defined. You define the value for the distance in the status window.

a) If necessary, enlarge the 3D gadget by pressing the **+** key.

b)  At the 3D gadget, identify a colour on the *arrow*, *arc* or segment of the *plane* in whose direction you want to move the gadget.

c) Identify a flat, cylindrical, spherical or conical surface to which the distance is to be set.

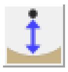
For non-planar surfaces, the intersection points (also for pitch circle bodies) are calculated over the surface of a full circle body. The axis to be moved can have no, one or two intersection points with the (parallel) lateral surface of the body. With two intersection points, the nearest point is determined in the direction of the arrow. A positive distance is always outside the body and a negative distance inside the body. Example picture

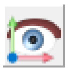


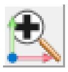
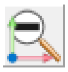
d) In the status window the current distance between the zero point of the 3D gadget and the identified surface is displayed and can be changed.

e) Finish the individual steps with the right mouse button.

Double-click the right mouse button to cancel the command.

 **Move with distance to point on surface**
With this command you move the 3D gadget along a surface and define the target point by cursor. In the status window you can define a distance to the body surface beforehand, visualized by a line to the 3D gadget.
This command is intended, for example, for the course of a hose line or a 3D line with a distance from any surface.

 **3D Gadget visible /**  **3D Gadget invisible (Y)**
Press this button or the **Y** key to display or hide the 3D gadget.

 **Enlarge 3D Gadget /**  **Shrink 3D Gadget**
To enlarge or reduce the display of the 3D gadget, use the buttons shown or the **+** and **–** keys on the keyboard.

6. Finish the transformation by clicking the right mouse button or the ***ESC*** key.

Please note:

Settings for the display of the dynamic preview can be made using the menu command ***Settings***, ***3D Display***.

Divide Surface by Curves

from ***BeckerCAD 3D pro***

,  After clicking the displayed icons the command ***Solid Local Operations***, ***Divide Surface by Curve*** is started.

Using this command 2D and 3D curves are projected to a surface and generate additional edges. The projection is done in the Z direction of the current coordinate system. The surface will be divided, if due to the projection partial surfaces are created. Otherwise only edges are created on the surface. The partial surfaces can be useful in further processes, for example in ***Offset Faces*** or ***Taper Faces***.

1. Activate a work plane to define the direction of the projection. In the case of 2D curves it will be their work plane; in the case of 3D curves it can be any work plane.
2. Start the command ***Divide Surface by curves*** by clicking the displayed icons.
3. Identify the surface.
4. Select 2D and / or 3D curves. Collect them with the pressed Ctrl key or select multiple by dragging a window.

Please note:

To erase the edges use the menu command ***Extras***, ***Simplify Solid Geometry***.

Divide Solid at Plane

from ***BeckerCAD 3D pro***

Fixed solids can be divided at a plane using the command ***Divide Solid at Plane*** in the ***Solid local operations*** icon bar.

Follow these steps to divide a solid:

1. ,  Start the command ***Solid local operations***, ***Divide Solid at Plane*** by clicking the displayed icons.
2. Identify the solid to be divided.
   Several solids can be selected by dragging a rectangle or with the pressed ***CTRL*** key. Release the ***CTRL*** key before identifying the last solid.
3. Define a divide plane. For this identify a planar face or the frame of a work plane. Alternatively click the right mouse button and define a plane by 3 points.

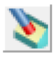
4. Identify the part of the solid which should remain as base solid. This part will contain the 3D axis.

Fixed solids at this point are divided.

Please note:
If solids are divided an additional solid is created. If the source solid is part of a group the new solid will also be part of this group.

Remove Surfaces of Fixed Solids

from ***BeckerCAD 3D pro***

,  After clicking the displayed icons the command ***Solid Local Operations***, ***Remove Faces*** is started.

Using this command single or multiple surfaces of fixed solids can be removed, if connected surfaces can close the gap. For example, blends or chamfers can be erased very easy. Also complex breakthroughs, protrusions and depressions can be erased using this command.

1. Start the command by clicking the displayed icons.

2. Select the desired surfaces using the ***3D Face Selection***.

3. If the surfaces can be erased the new state is displayed in a preview.
Confirm to erase the surfaces by clicking the left mouse button.

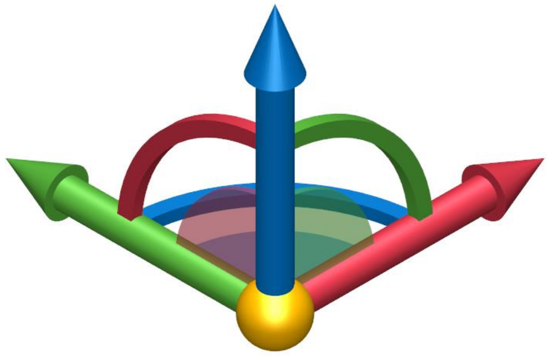
5.8 3D Gadget

The 3D gadget is used in various commands to move or rotate light sources, bodies, or body surfaces.

Within a respective command, when the 3D gadget appears, various transformation options are available to you, which you can execute by clicking on the areas of the 3D gadget described below.

In addition, the ***3D Transformation*** dialog opens, which provides you with further options for aligning the 3D gadget. These are described below.

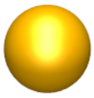
Transformation options

In the 3D gadget click on

- an **arrow**, to move the selection along the arrow direction.

- an **arc**, to rotate the selection around the arrow with the same colour.
- a **segment**, to move the selection in a plane.

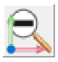


- the **sphere**, to move the selection in 3D space.

Define the target position by entering values in the status window or by defining points.

General process

1. Start any command where the 3D gadget can be used and follow the steps below.

2.   When the 3D axis cross appears, you can enlarge or reduce it with the **+ / -** keys or the symbols shown in the ***3D Transformation*** dialog.

3. **Moving the 3D gadget**

3.1  If you only want to reposition the 3D gadget without 3D object, click the button shown or press the button **X**.

3.2 Identify at the 3D gadget

- an arrow if you want to move along an axis

- a segment to move in a plane

- the ball, if you want to move the 3D gadget in 3D-space.

4. **Rotating the 3D gadget**

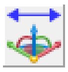
4.1 If you only want to rotate the 3D gadget without 3D object, click the button shown or press the button **X**.

4.2 Click on an arc in the 3D gadget.

5. Define the target position of the transformation via point definition or input in the status window.
6. Finish the transformation by pressing the right mouse button or the **ESC** key.

Please note:
You make settings for the display of the dynamic preview using the menu command ***Settings, 3D Display***.

3D Transformation Dialog

 **Move 3D Object /**  **Move 3D Gadget (X)**

This button is used to select whether the 3D gadget should be moved in combination with the 3D object or without. Make the selection before you identify an area at the 3D gadget using the button shown or the **X** key

 **Align plane at 3D gadget parallel to surface**

This command allows to align a plane from the 3D gadget to a plane, cylindrical, spherical or conical surface parallel. You can also align a plane of the 3D gadget to a work plane. After you have started the command using the button shown, proceed as follows:

1. If necessary, enlarge the 3D axis cross by pressing the **+** key.
2.  At the 3D gadget, identify a colour on the *arrow*, *arc*, or segment of the *plane* you want to align.
3. Identify a plane, cylindrical, spherical or conical surface to which the plane of the 3D gadget is to be aligned parallel. In the case of non-planar surfaces, the plane is aligned tangentially to the surface from the 3D gadget.

   Alternatively, you can identify the frame of a work plane.
4. In the status window the current distance between the plane of the 3D gadget and the identified surface is displayed and can be changed.
5. Finish the individual steps with the right mouse button.

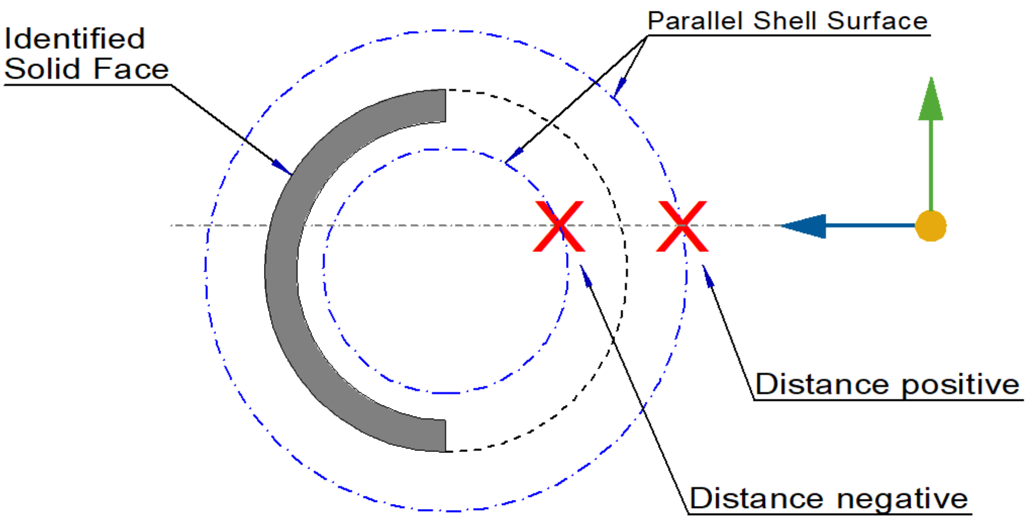
   Double-click the right mouse button to cancel the command.

 **Move along arrow with distance to surface**

To move the point at which you have positioned the 3D gadget with a distance from a flat, cylindrical or conical surface, start the command using the button shown. The point is moved along an axis to be defined. You define the value for the distance in the status window.

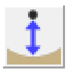
1. If necessary, enlarge the 3D gadget by pressing the **+** key.
2.  At the 3D gadget, identify a colour on the *arrow*, *arc* or segment of the *plane* in whose direction you want to move the gadget.
3. Identify a flat, cylindrical, spherical or conical surface to which the distance is to be set.

   For non-planar surfaces, the intersection points (also for pitch circle bodies) are calculated over the surface of a full circle body. The axis to be moved can have no, one or two intersection points with the (parallel) lateral surface of the body. With two intersection points, the nearest point is determined in the direction of the arrow. A positive distance is always outside the body and a negative distance inside the body. Example picture



4. In the status window the current distance between the zero point of the 3D gadget and the identified surface is displayed and can be changed.
5. Finish the individual steps with the right mouse button.

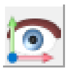
   Double-click the right mouse button to cancel the command.

 **Move with distance to point on surface**

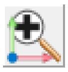
With this command you move the 3D gadget along a surface and define the target point by cursor. In the status window you can define a distance to the body surface beforehand, visualized by a

line to the 3D gadget.
This command is intended, for example, for the course of a hose line or a 3D line with a distance from any surface.

1. Within a running command, click on the symbol shown in the ***3D transformation*** dialog.
2. Identify a surface.
3. If necessary, set a value for the distance to the surface in the status window.
4. Move the cursor over the surface and determine the target point by mouse click.

 **3D Gadget visible /**  **3D Gadget invisible (Y)**

Press this button or the **Y** key to display or hide the 3D gadget.

 **Enlarge 3D Gadget /**  **Shrink 3D Gadget**

To enlarge or reduce the display of the 3D gadget, use the buttons shown or the **+** and **–** keys on the keyboard.

<u>Short cut keys</u>

In all commands using the 3D gadget the following short cut keys can be used.

+ / -	Enlarges or reduces the 3D gadget.
Y	Hiding/showing the 3D gadget.
X	Switch between 3D gadget **with** or **without** 3D-object / surface / geo-point transformation.
B	*Only in the command of transforming surfaces:* Switches the dynamic preview of the body **on/off** to snap points on the solid. If the preview is switched off, the original state is displayed before the current processing.

5.9 Transform solids

 Alternatively, the command for transforming fixed solids can be started using this icon

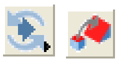
It contains the ***Reposition*** command which can be used to move solid in 3D space and, if necessary, align on an axis/edge of another solid. In addition, there are commands with which solids can be moved, rotted and mirrored in reference to the current workplane.

If the named commands are to be carried out without generating copies, fixed solids can be transformed if they have previously been interlinked.

The further command in the toolbox can be used to multiply solids and sort them with the same distances in the form of a longitudinal, matrix or circular spacing.

When you move, rotate or mirror a solid dynamically, a transparent gradation into 50% will be shown so that it is easier for you to specify the target point. The degree of transparency can be edited as required under **Settings / 3D Display**.

Reposition solids

 After clicking in the displayed icons, the ***Transform, Reposition Solid*** command is started.

This command makes it possible, as before, to move one or more solids in 3D space and, if necessary, align them on an axis, edge or face of another solid or workplane.

General procedure:

1. Start the command and identify the solid to be positioned.
2. In the context menu displayed, select one of the options in order to define the **initial context**.
   This step will specify a reference (a point, a face, a WP, an axis or an edge) that will be used to position the objects.
   If a curved face is defined as the initial context, the pick point is determined in this face.

   The initial context can even be on one of the objects that are going to be moved. But this is only optional.
3. A context menu will appear so that you can choose an option in order to define the **target context** (point, face or WP, axis or edge).

   In addition to the combination of the initial context and the target context you can also use other methods for positioning (as listed below in the table).

4. According to the option chosen, define the final context target point, or identify any face or WP or an axis or an edge formed by a straight line.
   After selecting a face as the target context you can also identify irregular faces.

5. **Context point**
   If a point has been specified as the initial or target context, the objects will be moved but not re-oriented.

   If the initial state and target context have been defined as a point, the spatial reorientation will be completed directly.

   If you have specified the initial context as a point and the target context differently, the objects will be moved in such a way that the point lies in the target context.
   Define a point to specify the position.

   If you have specified the target context as a point and the initial context state differently, the objects will be moved to the point.
   The program will generate a temporary WP at the position that has been specified. Using the commands available in the 3D toolbar for processing workplanes you can move, rotate or otherwise manipulate the WP and thus the objects it contains.

   Context face, axis/edge
   When the initial and target contexts are an axis/an edge, a face/a workplane, the objects will be moved and re-orientated.

   If you have identified a face that is not flat as the initial context, it will be positioned tangentially on a face specified as the target context by means of the snap point that has been determined earlier.

   The position is set by a point that you must specify.
   When the target context is a straight axis or edge, you can press the *Shift* key to move the solid over the axis/the edge.

   A temporary workplane will be generated at the position that has been defined. Using the commands available in the 3D toolbar for processing workplanes you can move, rotate or otherwise manipulate the WP and thus the objects it contains.

6. Once the correct position has been found, end the command by clicking either the left-hand or right-hand mouse button.
   Alternatively, you can repeat Step 2 and then specify a different initial context.
   Depending on the combination of initial and target context you will have a choice of how to position:

Initial context: point

Target cont.	Orientation	Move
Point	No	Initial context point to target point
Axis Edge	No	Initial context point to target axis/edge Position definable dynamically or with Snap Menu on axis/edge
Face/WP	No	Initial context point to target face/ WP. Position definable dynamically or with Snap Menu on face/WP

Initial context: face/WP

Target cont.	Orientation	Move
Point	No	Centre of gravity initial face or the WP origin to target point
Axis Edge	Initial face perpendicular to target axis or target edge	Centre of gravity initial face or the WP origin to target axis/edge Position definable dynamically or with Snap Menu on axis edge
Face/WP	Initial face coplanar to the target face or target WP	Centre of gravity initial face or WP origin to target face/WP Position definable dynamically or with Snap Menu on face/WP Temporary WP for work in hand

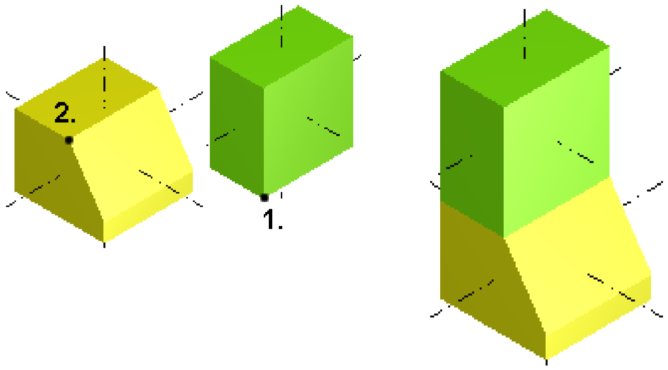
Initial context: axis/edge

Target cont.	Orientation	Move
Point	No	Identified endpoint original axis/edge to target point Temporary WP for work in hand
Axis Edge	Original axis or edge follows same direction as target axis or target edge	Identified endpoint original axis/edge to target axis/edge Position definable dynamically or with Snap Menu on axis/edge Temporary WP for work in hand
Face/WP	Original axis perpendicular to target face/target WP	Identified endpoint original axis/edge to target face/WP

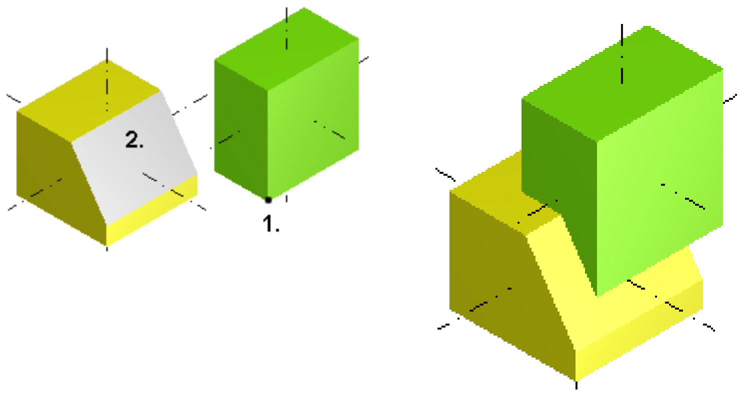
Original edge coplanar with target face/target WP	Position definable dynamically or with Snap Menu on face/WP
	Temporary WP for work in hand

Initial context: point

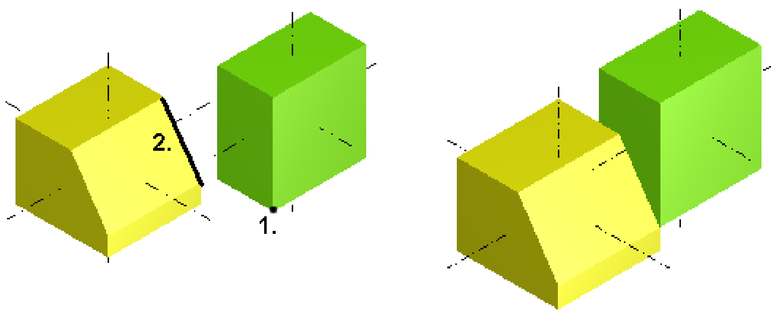
Target context: point



Target context: face

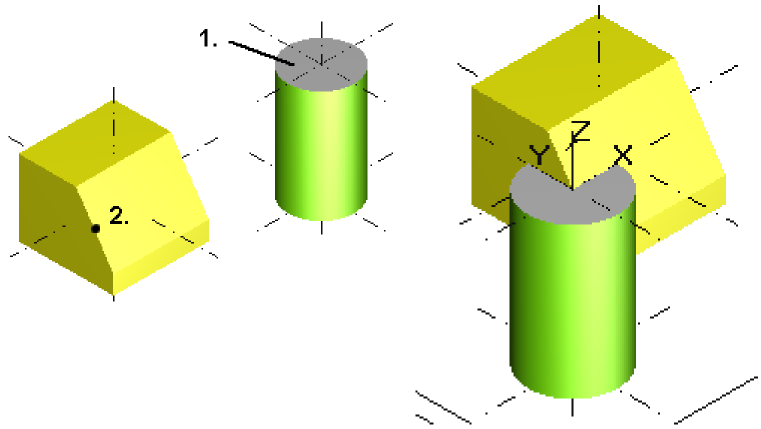


Target context: edge

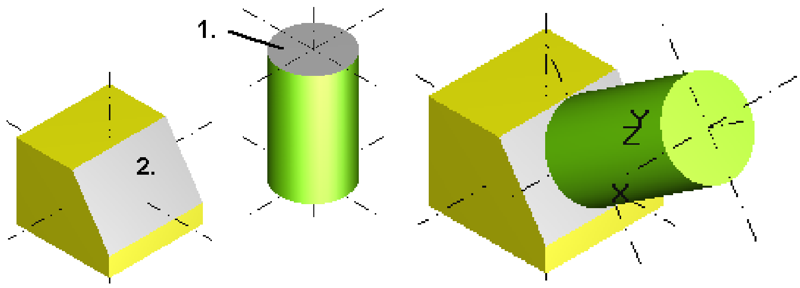


Initial context: face

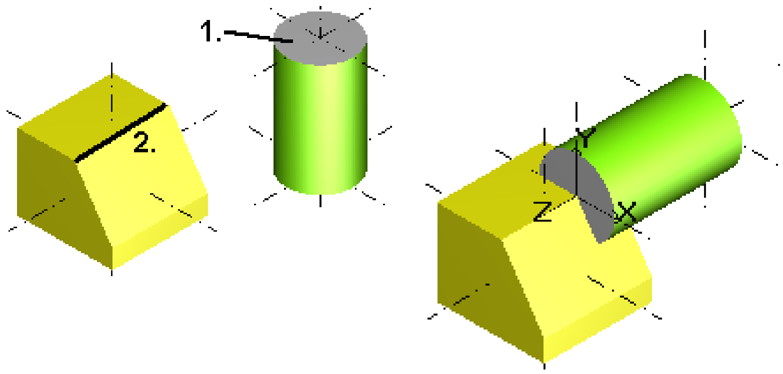
Target context: point



Target context: face

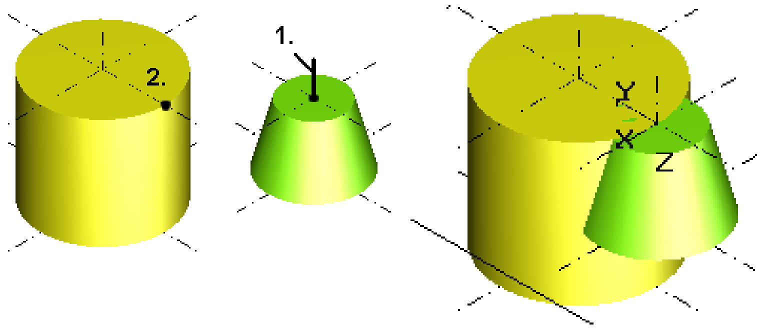


Target context: edge

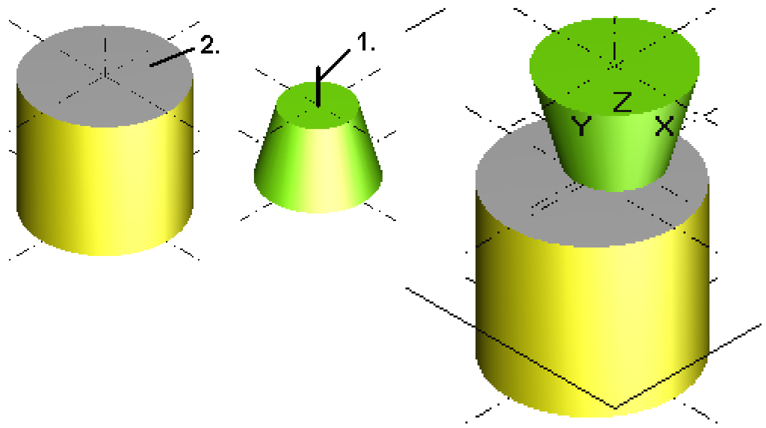


<u>**Initial context: axis**</u>

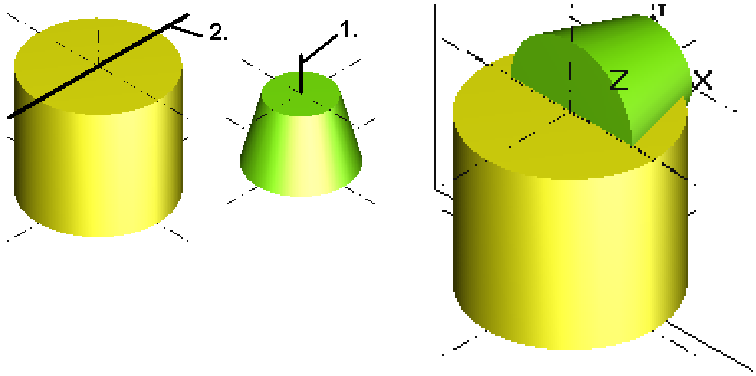
Target context: point



Target context: face



Target context: edge



Transform 3D Objects

from ***BeckerCAD 3D***

, After clicking the displayed icons the command ***Transform***, ***Transform 3D objects*** is started.

This command allows to move and / or rotate 3D objects in 3D space via the 3D gadget.

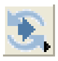
1. Identify the 3D object to be transformed. If you want to edit several objects, collect them while holding down the Ctrl key or select them by window. You can also copy the 3D objects to the selection list before calling this command.
2. Define the start point for the transformation (= origin of the 3D gadget).
3. Depending on the type of transformation, you identify an area at the 3D gadget. The possibilities are described in the chapter 3D Gadget.

Move solids

The two commands described in the following section can be used to move solids and 2D objects or their copies in the selected WP of the 3D window.

Such a move results in a complete modification of the position of the objects, and not simply a modification to the view.

Move solids

  Alternatively, the command can be loaded using this icon.

This command corresponds to the ***Transform, Move*** command for 2D objects.

This command can be used to move solids and 2D objects.

In order to carry out the command, the objects to be moved must be selected and the move vector must be defined. The following options are available:

Selection - action (dynamic move)

1. Take over the objects to be moved into the selection list **before** executing this command.
2. Specify the start point of the move vector (= reference point for the move).

   Please note:

   The start point of the moving vector must only be defined in the drawing / work plane if the vector is defined by the point definition menu.
3. Specify the target position of the reference point in the same way. If the cursor is used, the objects contained in the selection list are moved dynamically.

If the command is aborted, the objects will be redrawn at their original positions.

Move with action - selection

1. If the selection list is empty, specify the start point of the move vector (= reference point for the move).

   Please note:

   The start point of the moving vector must only be defined in the drawing / work plane if the vector is defined by the point definition menu.

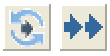
2. Specify the target position of the reference point in the same way.
3. Specify the object to be moved using object selection.

Please note:

If solids are to be moved in 3D space, use the command ***Transform, Reposition Solid***.

Using the menu ***Settings / Object Filter***, you can define the ***Copy mode for objects***. Depending on the settings made here, only objects can be copied or group structures can also be copied.

Move and copy solids

 Alternatively, the command can be loaded using this icon.

This command corresponds to the ***Transform, Move With Copy*** command for 2D objects.

This command can be used to generate and move copies of solids and 2D objects.

In order to carry out the command, the objects to be copied must be selected and the move vector must be defined.

The following possibilities are available:

Copy with selection - action (dynamic copy)

1. Take over the objects to be copied into the selection list **before** executing this command.
   After you have executed this command, copies of the objects included in the selection list will be made.

2. Specify the start point of the move vector (= reference point for the move).

   Please note:

   The start point of the moving vector must only be defined in the

drawing / work plane if the vector is defined by the point definition menu.

3. Specify the target position of the reference point for the copies in the same way.
   If the cursor is used, the objects that have been copied will be moved dynamically.

If the command is aborted, the copies will be deleted.

Copy with action - selection

1. If the selection list is empty, specify the start point of the move vector (= reference point for the move).

   Please note:
   The start point of the moving vector must only be defined in the drawing / work plane if the vector is defined by the point definition menu.

2. Specify the target position of the reference point of the copies in the same way.
3. Specify the object to be copied using object selection.

The copies of the objects will have the display attributes of the original object.

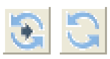
Please note:
Using the menu ***Settings / Object Filter***, you can define the ***Copy mode for objects***. Depending on the settings made here, only objects can be copied or group structures can also be copied.

Rotate solid

The commands for rotating can be used to rotate solids and 2D objects or their copies in the currently selected workplane.
The rotation axis runs parallel to the Z axis through the fixpoint to be defined.

A rotation results in an actual modification of the position of the objects, not simply a rotation of the view.

Rotate solid

 Alternatively, the command can be loaded using this icon.

This command corresponds to the ***Transform, Rotate*** command for 2D objects.

This command can be used to rotate solids and 2D objects.

In order to carry out the command the objects to be rotated must be defined and the rotation data specified. The following possibilities are available:

Rotate with selection - action (dynamic rotate)

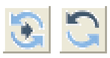
1. Take over the objects to be rotated into the selection list **before** executing this command.
2. Specify the fixpoint about which will be rotated.
3. In the same way, specify a point which is to act as the reference point for the rotation:
   The connection between the fixpoint and this point defines the origin from which the rotation angle starts.
4. Specify the rotation angle.
   This can be entered as a number in the ***Angle*** text box of the Status dialog box.
   It can also be specified by specifying the new angle position of the reference point.
   If the cursor is used, the objects will be rotated dynamically.

If the command is aborted, the objects will be redrawn at their original position.

Rotate with action - selection

1. If the selection list is empty, define the fixpoint about which should be rotated.
2. In the same way, specify a point which is to act as the reference point for the rotation:
   The connection between the fixpoint and this point defines the origin from which the rotation angle starts.
3. Specify the rotation angle.
   This can be entered as a number in the ***Angle*** text box of the Status dialog box.
   It can also be entered by specifying the new angle position of the reference point.
   If the cursor is used, the objects will be rotated dynamically.
4. Specify the objects to be rotated using object selection.

Rotate and copy solids

 Alternatively, the command can be loaded using this icon.

The command corresponds to the ***Transform, Rotate With Copy*** command for 2D objects.

This command can be used to generate and rotate copies of solids and 2D objects.

In order to carry out the command the objects to be rotated must be defined and the rotation data determined. The following possibilities are available:

Rotate with selection - action (dynamic rotate)

1. Take over the objects to be rotated into the selection list **before** executing this command.
After selecting this command, copies are generated of the objects contained in the selection list.

2. Specify the fixpoint about which the copies will be rotated.

3. In the same way, specify a point which is to act as the reference point for the rotation:
The connection between the fixpoint and this point defines the origin from which the rotation angle starts:

4. Specify the rotation angle about which the copies will be rotated.
This can be entered as a number in the ***Angle*** text box of the Status dialog box.
It can also be specified by entering the new angle position of the reference point.
If the cursor is used, the objects will be rotated dynamically.

If the command is aborted, the copies will be deleted.

Rotate with action - selection

1. If the selection list is empty, specify the fixpoint about which the copies should be rotated.

2. In the same way, specify a point which is to act as the reference point for the rotation:
The connection between the fixpoint and this point defines the origin from which the rotation angle starts.

3. Specify the rotation angle about which the copies are to be rotated.
This can be entered numerically in the ***Angle*** text box of the Status dialog box.
It can also be specified by defining the new angle position of the reference point.
If the cursor is used, the objects will be rotated dynamically.

4. Specify the objects to be rotated using object selection.

The copies of the objects will have the display attributes of the original object.

Please note:

Using the menu ***Settings / Object Filter***, you can define the ***Copy mode for objects***. Depending on the settings made here, only objects can be copied or group structures can also be copied.

Scale solid

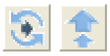
These command can be used to scale solids and 2D objects or their copies in the currently selected workplane.

The solids are scaled by the same factor starting at the fixpoint in all axes directions.

Please note:

Since solid data is managed with up to an exactness of 6 decimal places, solid scalings could become extremely imprecise if it is necessary to round up calculation results. This would then lead to inconsistencies in your model. This is particularly valid when enlarging solids.
Therefore, when using this command be aware not to enter too large or too small factors.

Scale solids

 Alternatively, the command can be loaded using this icon.

This command corresponds to the ***Transform, Scale*** command for 2D objects with the difference being that in the 3D window the same factor will apply to both axis orientations.

This command can be used to scale solids and 2D objects with the same factor in all axis directions.

Please note:

Since solid data is managed with an accuracy of 6 decimal places, solid scalings could not be precise if it is necessary to round up calculation results. This would then lead to inconsistencies in your model. This is particularly valid when enlarging solids.
Therefore, when using this command be aware not to enter too large or too small factors.

Before carrying out this command, the objects to be scaled must firstly be selected. The fixpoint as well as the factor must be defined for the scaling. The following possibilities are available:

Scale with selection - action (dynamic scale)

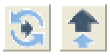
1. Take over the objects to be scaled into the selection list **before** loading the command.
2. Specify the fixpoint of the scale.
3. In the same way, specify the reference point for the scale. The connection from the fixpoint to this point defines a ray.
4. Specify the scale factor.
   The required value can be entered in the ***Factor XY*** text box of the Status dialog box.
   The length of the ray can also be specified by defining another point which corresponds to the scale factor.
   If the cursor is used, the copied objects are scaled dynamically.

If the command is aborted, the objects are redrawn in their original size and position.

Scale with action - selection

1. If the selection list is empty, define the fixpoint of the scale.
2. In the same way, determine a reference point for the scale. The connection from the fixpoint to this point defines a ray.
3. Specify the scale factor.
   The required value can be entered in the ***Factor XY*** text box of the Status dialog box.
   The length of the ray can also be specified by defining another point which corresponds to the scale factor.
4. Specify the objects to be scaled using object selection.

Scale and copy solids

 Alternatively, the command can be loaded using this icon.

The command corresponds to the ***Transform, Scale With Copy*** command for 2D objects with the difference being that in the 3D window the same factor will apply to both axis orientations.

This command can be used to generate copies of solids and 2D objects and scale in all axis directions.

Please note:

Since solid data is managed with up to an exactness of 6 decimal places, solid scalings could not be precise if it is necessary to round up calculation results. This would then lead to inconsistencies in your model. This is particularly valid when enlarging solids.

Therefore, when using this command be aware not to enter too large or too small factors.

Before carrying out this command, the objects to be copied must firstly be selected. The fixpoint as well as the factor must be defined for the scaling. The following possibilities are available:

Scale with selection - action (dynamic scale)

1. Take the objects to be copied over in the selection list **before** loading the command.
   After loading the command, copies of the objects contained in the selection list are generated.
2. Define the fixpoint of the scale.
3. In the same way, determine the reference point for the scale.
   The connection from the fixpoint to this point defines a ray.
4. Specify the scale factor.
   The required value can be entered in the ***Factor XY*** text box of the Status dialog box.
   The length of the ray can also be specified by defining another point which corresponds to the scale factor.
   If the cursor is used, the copied objects are scaled dynamically.

If the command is aborted, the copies will be deleted.

Scale with action - selection

1. If the selection list is empty, define the fixpoint of the scale.
2. In the same way, determine a reference point for the scale.
   The connection from the fixpoint to this point defines a ray.
3. Specify the scale factor.
   The required value can be entered in the ***Factor XY*** text box of the Status dialog box.
   The length of the ray can also be specified by defining another point which corresponds to the scale factor.
4. Specify the objects to be scaled using object selection.

The copies of the objects will have the display attributes of the original object.

Please note:

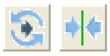
Using the menu ***Settings / Object Filter***, you can define the ***Copy mode for objects***. Depending on the settings made here, only objects can be copied or group structures can also be copied.

Mirror solid

These commands can be used to mirror solids and 2D objects or their copies in the currently selected workplane.

The mirror plane lies vertical to the WP. The position is determined by the mirror axis which is to be defined.

Mirror solid

 Alternatively, the command can be loaded using this icon.

The command corresponds to the ***Transform, Mirror*** command for 2D objects.

This command can be used to mirror solids and 2D objects.

In order to carry out the command, the object to be mirrored must be selected and the position of the mirror axis must be defined. The following possibilities are available:

Mirror with selection - action (dynamic mirror)

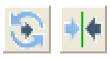
1. Take the objects to be mirrored over in the selection list **before** loading the command.
2. Define the first point of the mirror axis.
3. Define the complete position of the mirror axis:
   The required value can be entered in the ***Angle*** text box of the Status dialog box.
   It can also be specified by defining another point.
   If the cursor is used, the copied objects are mirrored dynamically.

If the command is aborted, the objects are redrawn at their original position.

Mirror with action - selection

1. If the selection list is empty, define the first point on the mirror axis.
2. Define the complete position of the mirror axis:
   The required value can be entered in the ***Angle*** text box of the Status dialog box.
   It can also be specified by defining another point.
3. Specify the object to be mirrored using object selection.

Mirror and copy solid

 Alternatively, the command can be loaded using this icon.

The command corresponds to the ***Transform, Mirror With Copy*** command for 2D objects.

This command can be used to generate and mirror copies of solids and 2D objects.

In order to carry out the command, the object to be mirrored must be selected and the position of the mirror axis must be defined. The following possibilities are available:

Mirror with selection - action (dynamic mirror)

1. Take the objects to be copied over in the selection list **before** loading the command.
After selecting this command, copies are generated of the objects included in the selection list.

2. Define the first point of the mirror axis.

3. Define the complete position of the mirror axis:
The required value can be entered in the ***Angle*** text box of the Status dialog box.
It can also be specified by defining another point.
If the cursor is used, the copied objects are mirrored dynamically.

If the command is aborted, the copies will be deleted.

Mirror with action - selection

1. If the selection list is empty, define the first point on the mirror axis.

2. Define the complete position of the mirror axis:
The required value can be entered in the ***Angle*** text box of the Status dialog box.
It can also be specified by defining another point.

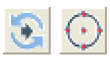
3. Specify the object to be mirrored using object selection.

The copies of the objects will have the display attributes of the original object.

Please note:

Using the menu ***Settings / Object Filter***, you can define the ***Copy mode for objects***. Depending on the settings made here, only objects can be copied or group structures can also be copied.

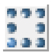
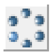
Move fixed solids along their axes

 Alternatively, the command can be loaded using this icon.

Solids which have been linked with solid axes during generation or later can be moved dynamically in the direction of these axes.

In order to do so, identify a solid axis and move the solid dynamically to the required position. Confirm the position with the left mouse button.

Multiply solid

, , , ,  If previously generated solids are to be multiplied and the copies allocated similarly, use the commands offered in the ***Transform*** toolbar: ***Multiply with sort in longitudinal*, *in wreath, in matrix* or *circular spacing***.

The following option is available in the 3D window after starting the commands:

Unite bodies

If this option is activated, the copies are combined with the original solid when you select the solid.
If this option is not active, each copy is created as a separate solid.

If a group is selected as object to be copied, new groups including their complete content are created on the same structure level.

The procedure corresponds to the respective commands for 2D objects and is described there.

Please note:

Using the menu ***Settings / Object Filter***, you can define the ***Copy mode for objects***. Depending on the settings made here, only objects can be copied or group structures can also be copied.

5.10 Control solid display

When generating solids, like 2D objects, they are linked to the current drawing layer. Out of these, only the settings ***active/inactive*** and ***visible/hidden*** are taken into consideration.

Please note:

You must remember not to use the same layers for solids and for 2D objects if at all possible. The reason is that any alterations to these properties will have an effect on the 2D objects and they might also be hidden whenever you hide the solids in the display.

The true display properties (colour, material, shading, display of edges) of the solids cannot be defined according to layers. This procedure has to be completed in relation to objects.
They can be specified while being generated by means of the ***Settings, 3D Display*** menu command. They can be changed by using ***Process, Object Display***.
You can even display separate faces of solids in different colours or material properties by using ***Process, Solid Face Display***.

The accuracy for calculating curved surfaces on shaded solids and displaying them can be altered by using the *Solid facetting* option card after activating the ***Process, Object Display*** command.

The lighting, which can also be specified using the menu command ***Settings, 3D Display***, always effects the entire 3D model.

Specify settings for solid display

When you are generating them, solids are directly allocated display attributes which can be set using the ***Settings, 3D Display*** command on the ***Solid Display*** option card.

The way in which one or more solids are displayed can be altered by means of ***Process, Object Display***.
The accuracy for calculating curved surfaces on shaded solids and displaying them can be altered by using the ***Solid facetting*** option card.

Although you can always specify different settings for each solid, this is not possible for the lighting for the model: in this case you will have to use ***Settings, 3D Display*** on the ***Lighting*** option card.

Option card: Solid display

Use this option card to indicate the display properties that will apply to the solids displayed immediately afterwards:

face display
You can choose whether or not solids are to be displayed and, if so, how the faces of the solids are to be displayed:

Without shading
Use this setting to suppress the display of the faces of the solids about to be generated.
If this is chosen as the type of display, you must click at least one of the options edges, silhouettes, etc. because the solids will not otherwise be visible and so, obviously, you will not be able to use the cursor to select any of them.

Flat shading
Solid faces - dependent on the settings determined in the ***Lighting*** option card - are displayed without shading, i.e. displayed without colour run.

Gouraud shading
Solid faces can be displayed with a colour run provided that

the option card for ***Lighting*** indicates both ***sunlight*** as well as ***light sources***.

Stereo lithography

This option is used as visual control of the settings for the export of files for stereo-lithography.
These settings can be specified using the menu command ***File, Export, Export Settings***.

Colour/Reflection as...

If ***Flat shading*** or ***Gouraud shading*** is active for the ***face display*** setting, the colour, transparency and/or material attributes can be set for face display:

1. In order to determine the colour, click on the colour box.
In the Colour dialog box which then appears, determine the colour selection and definition and confirm with ***OK***.

The colour can also be specified using the colour toolbar button at the top of the screen.

2. If the faces of the solids then generated are to be transparent, determine a numerical value for the transparency between 0 and 100.
It is very important to remember that a transparency of 100 will result in invisible faces.

The transparency can also be specified using the colour icon in the 3D toolbar.

3. If the solid is to be allocated material and reflection attributes, determine these in the dialog box.
User defined material properties can be defined in the tab ***Material Manager***.

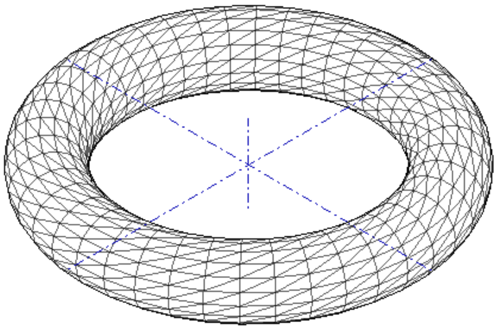
Density

Select a material from the list for all solids created afterwards and define a density. The density will be stored as property with the solids. To assign a density individually to a solid, use the command ***Process***, ***Object Display***.

Please note:

The list of materials can be extended. Open the file Density.cpa with an editor and extend the list.
The file can be found in the directory ...\Program\Bin.

Facetting Mode



Here you can set the precision with which shaded solids are to be presented with curved surfaces. It does not affect the accuracy of which the geometric data of the solids are managed.
Since the increase in accuracy means higher computational effort, you can specify your desired compromise between processing speed and accuracy of the representation itself with these settings.

Using the menu command ***Process***, ***Object Display*** you can enhance the display of surfaces in the ***Solid Facetting*** tab.

edges/silhouettes

In the middle section of the dialog you can determine whether and with which colours and line types, visible edges and hidden edges / visible silhouettes and hidden silhouettes are to be displayed. Silhouettes are, in fact, those lines that bound a view showing solid faces that are not flat.

1. In order to determine that edges are to be displayed, activate the check box in front of the respective line description.

2. Activate the colour button in order to load the Colour dialog box and determine the respective colour and colour definition. Confirm these settings with ***OK***.

3. Click on the list box to the right to determine which line type is required.

Further information concerning the display of edges and silhouettes *If the display of edges is activated, also this edges as well as the shaded solids are displayed based on the scene graph.*
Depending on the graphic board and graphic driver also silhouettes can be displayed based on the modern Shader technology. This can be activated in the area hardware support with the option Silhouettes.
The Shader programming has been developed to support up-to-date graphic boards with the advantage that the performance of the graphic boards can be used. The direct calculation on the graphic board speeds up the display of silhouettes, because they can be calculated simultaneously during a dynamic view.
Please note, that this performance and quality is very much depending on the hardware and the driver.
The following preconditions must be fulfilled for the new display of

edges and silhouettes:
- For silhouettes the graphic board must support a geometry shader, which is the case at least for DirectX 11. Otherwise the option silhouettes in the area hardware support is deactivated.
- Visible and hidden edges must be assigned to the same color and line type. Best conditions are given with the settings black and solid.
- Visible and hidden silhouettes must be assigned to the same color and line type.
- If transparency is bigger than 0 the display of visible and hidden edges / silhouettes must be activated.

clipped colour properties

Work planes can be changed to cutting planes. If a clip planes cuts solids, they are displayed in a section view. The display of the sections can be set with this option.
Further information's can be found here

render method

The display of all contained 3D objects can be displayed with different **Render Methods.** Select from the list of Render Methods:

Standard GL (Polygon)
Previous, non shader based lighting model. The calculation is based on polygons. Planar faces reflects the light completely.

Shader (Pixel)
Shader based lighting, which requires a graphic board designed for graphic programs. Calculation method is based on pixels, so that a spot light will result in a circular illumination also on a planar face.

Cartoon Shader
Pixel-Shader based lighting. To reach the best result there should only be one light source active in the scene and the edges and silhouettes of all solids should be activated.

Technical Illustration 1 & 2
Pixel-Shader based lighting for a non photorealistic Render method. The edges and silhouettes of all solids should be activated. The surroundings of objects and important edges are accentuated then. The face structure will get lost.

Transparency in dynamic actions

Define a value for the transparency in dynamic actions between 0 and 100.

The solid is displayed in this transparency during dynamic move or dynamic copy for example.

Transparency with Lowlight

To be sure to have the correct group active when creating new objects in the 3D window, all other objects, not contained in the active group, can be displayed in the transparency defined here. In the chapter Lowlight non active groups is described, how the lowlight for non active groups can be defined.

Display in Dynamic 3D Operations

Here you can choose if and how the edges and surfaces are to be displayed when creating and editing.

- **Display of edges using 2D lines**
  This option can be selected alternatively to the display by edges for presentation.
- **Shaded display of faces**
  In dynamic operations surfaces are shaded in the color of the solid - depending on the ***setting transparency with dynamic operations*** possibly also transparent.
- **Display of edges**
  This option can only be used in combination with the option ***Shaded display of faces***. For the presentation you can select a color from the color button.

  Facetting mode
  Here you can set the precision with which shaded solids are to be presented with curved surfaces.

3D graphic hardware support: Display of Solids with Antialiasing

For the display quality of solids the option for ***Antialiasing*** can be set, provided that the installed hardware and software supports this option.

This option reduces the aliasing effect, which appears if lines are not orthogonal to the screen. The higher the value the better the result on the screen. The setting is only valid when reloading the model or opening another 3D window.

Please be aware:
This option should only be used, if high quality hardware is used. The quality of any output is not effected, it effects only the display on the screen.

3D graphic hardware support: Silhouettes

When activated, silhouettes are calculated with the support of the Shader technology of up to date graphic boards, if the precondition are fulfilled. This opotion can only be activated, if the graphic board and the driver supports the Shader technology.

Please note:

The colour specified for solids as well as the degree of transparency will be saved in UIBECKER.SYS.

The display of the existing solids can be altered by using ***Process, Display object*** and ***Process, Solid Face Display***: these commands are described in the following sections.

Option Card Lighting

All solids in a model, displayed in a shaded view, are displayed with the assigned object colour or material property.

To get a more realistic 3D display, settings for the lighting of the model can be made.
The display quality is a combination of solid colour and material property and the settings for the lighting. Therefore at this stage, there cannot be given any suggestions how to get a specific illustration. So in the following only the different steps to define the settings and some relations can be explained.

The lighting in the 3D window now supports the complete **OpenGL** – light model.

The settings for the lighting can be made in ***Settings***, ***3D Display*** on the tab ***Lighting***.

This dialog is divided in three main areas; the list of light sources, the parameters of the selected light source and the general settings.

Spot and position light sources can be linked to 3D objects (e.g. solids, 3D graphics), so that when the 3D object is transformed, the light source also changes its position or orientation. This further development allows you to position light sources in e.g. lamps and headlights and create realistic scenes.
For directional light sources that do not appear from the observer's perspective (Camera column), you can link the beam direction with 3D objects.

List of Light Sources

In the list of the light sources new lights can be defined and existing lights can be edited or deleted. Basically three types of lights can be created:

Directional Light (N)

Directional lights are on the point a infinity. They shine with a parallel light in a defined direction.

Positional Light (Ctrl+N)

Positional lights are on a defined position in the model. From this point they shine in all directions.

Spot Light (Ctrl+Shift+N)

Spot lights are also on a defined position in the model, Opposite to the positional light they shine in a defined direction with a cone.

A new light can be defined as follows:

1. Open the dialogue window ***Settings***, ***3D Display*** and select the tab ***Lighting***.

2. In the list of light sources open the context menu with the right mouse button and select a type of light.

3. Activate the light by double-clicking the icon in the row ***Active***.

4. For **positional** and **spot lights** the position can be defined in the model, if in the row Camera this is deactivated .
If the camera is active , the position is defined by the camera; the light always shines from the position of the user.

5. To display a **positional-** or **spot light** by a sphere in the model, deactivate the option **Camera** and active the option **Visible**.

6. For every light individual settings can be made. These are described in the following.

In a model a maximum of 8 lights can be defined, but every additional light will stress the calculation capacity of your computer.
The optimum in regard to quality and computing power can be reached with two directional lights. This optimum can be set by selecting the option ***Create standard lights (Crtl+Shift+S)*** in the context menu.

In the new ***Connected*** column, light sources linked to solids are indicated by the symbol .
Double-click on this icon to highlight and focus the solids connected to the light source. Provided that the options ***activate colour highlight*** and ***centred and enlarged display*** are activated on the ***User Interface*** tab in the ***Model Explorer*** area.

When light sources are not linked, the view is centred on the light source.

Connect light source to 3D object

To connect an existing light source to a 3D object, proceed as follows:

1. In the list of light sources, select the light source to be linked, open the context menu in the line and choose ***Connect light***.
2. Move the mouse pointer from the dialog into the 3D window.
3. Activate a point snap and identify the 3D object to which the light source is to be connected. The snap point serves as a reference point for the light source.
4. Starting from the reference point, you can further align the light source via the 3D gadget.

   The beam direction of directional light sources for which the *Camera* option is deactivated runs along the blue arrow of the 3D gadget.
5. Right-click to exit the command.

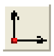
If you want to break the link between light source and 3D object, open the context menu on the light source and select the option ***Disconnect light***.

Current light source

In this area you have the possibility to position the light source marked in the list in 3D space and to influence the intensity.

Positional- and Spot light

If the Camera option is disabled, the position of positional lights and spotlights can be defined by entering the X-, Y- and Z coordinates.

Alternatively, after clicking the icon  ,the position can be defined in the model. Hereby the entries in the status dialogue window can be used as well as the point definition menu on the middle mouse button.

Directional light

If the Camera option is disabled, for the directional lights the direction can be defined by entering the X-, Y- and Z coordinates.

Alternatively, after clicking the icon  ,the direction can be defined by two points in the model. The status dialogue window can be used as well as the point definition menu on the middle mouse button. If points on existing geometry should be used, the 3D Cursor can be activated.

 Click this icon to change the position or orientation of the selected position or spot light source via the 3D gadget. For an infinite directional light source, set the direction of the parallel light beams. Decisive is the alignment of the blue arrow at the 3D gadget.
The possibilities are described in detail in the chapter 3D Gadget.

Light intensity

Set the intensity of the selected light source in a value range from 0 to 1 via input or slide control.

The quality of the display of 3D objects is dependent on the combination of solid colour or solid material and on the settings for the faceting of solids. We recommend to use the standard lights and the standard faceting.

Light shares

Lights are sending out three different light shares. These shares can be adjusted for every light. Hereby 3 shares are differentiated, which describes the different reflection of the light onto material:

Ambient Light

Light which is reflected from surfaces in all directions, independent from the position of the light, the surface and the viewer. The lightened surface is displayed nearly without any gradient in one colour. The solid is displayed without any structure.

Diffuse Light

Diffuse light emits in all directions. Because the light strength is depending on the angle of incidence, a gradient is displayed. This light creates the known structure of lightened models.

Specular Light

The specular reflection is mirrored on the surface of an object and creates a glance point. Hereby the solid must be assigned with a reflecting material.

Colour Model

The colour of each light can be defined by the colour icon or by defining it in one of the three following colour models:

RGB

The colour of the light can be defined by a slider or by entering a value in a scale from 0 to 255 for the colours **R**ed, **G**reen and **B**lue. The resulting colour is then calculated from this three basic colours.

HSV

The colour is calculated from the three components **H**ue, **S**aturation and **V**alue. Hue is defined in a scale from 0° to 360°, Saturation and Value is defined in %. If 100% is set for the saturation and the value, with the slider of the hue you can scroll through all the colours. Then 0° is for red, 120° is for green and 240° is for blue.

HSL

The HSL colour model is similar to the HSV colour model. The difference is the definition of the **L**ightness, which is defined from the middle. If the value is set between 51% and 100%, the colour is brighter, if the value is set between 50% and 0% the colour is darker.

More information's to the colour models can be found in technical literature or in the Internet, e.g. http://en.wikipedia.org/wiki/Colour_models .

As usually a light sends out white light, especially the HSV model is useful to define the light intensity in 3D space.

Distance Dependency

For the positional and the spot light a distance dependency can be defined. This defines the attenuation with the distance. The distance dependency can be defined by four parameters:

Constant Attenuation, is a distance independent attenuation.
Standard scale of the slider: 0 - 10
Standard value: 1

Linear Attenuation, an attenuation which increases proportional to the distance of the light.
Standard scale of the slider: 0 – 0.001
Standard value = 0

Quadratic Attenuation, an attenuation which increases quadratic to the distance of the light.
Standard scale of the slider: 0 – 0.00001
Standard value = 0

If in the entry field a value is entered which is outside of the scale (values possible from 0 to 100000), the scale will be adjusted automatically to this new value. To resize the value again, enter a value for the maximum and the a value for the minimum and press the ***Ctrl*** key while confirming the entry with ***Enter***. If the upper or lower boundary of the scale is changed will be decided by the current position of the slider. If he is in the left half, the lower boundary will be adjusted.

Settings for Spot Lights

For spot lights the following settings can be made:

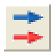
Light cone

The light cone is defined by the cone angle in a scale from 0° to 90°.

Exponent

The spot exponent define the attenuation of the light to the end of the cone in a scale from 0 to 128. If this value is 0, there is no attenuation.

Ray Direction of the Spot Light

The ray direction can be defined by X, Y and Z Coordinates starting from the origin or, after clicking the icon  , by the definition of two points in space.

<u>General Settings</u>

Global Light

With the global light the complete model can be displayed brighter without defining special lights or reflecting surfaces. If lights are activated, the value should not be to high, because otherwise the contrast will be to marginal.

Adapted Intensities

If this option is activated, all lights a normalized to the number of lights. All lights can deliver 100% intensity, but eliminates the risk to overexpose the model.
When deactivating this option, it must be taken care of that the summary of all lights do not overexpose the model.

Create new light source

After clicking on this icon, you can set the type and position of a new light source in the 3D window. By default, a spot light source is generated after starting the command. Press the **X** button to switch

to a directional light source or a position light source. The current type of light source is displayed in the message line.
If you position the light source via point snap on a 3D object, the light source is linked to the 3D object with regard to position and alignment. A linked light source is also deleted with the 3D object.
Alternatively, set the position for a light source via coordinates in the status window.
After you have positioned the light source, the 3D gadget appears, via which you can still change the position or orientation of the light source. The transformation options are described in the chapter 3D Gadget.

Global factor

Set the intensity of all light sources in a value range from 0 to 1 via input or slide control.

Option Card Material Manager

In addition to the delivered standard materials user defined materials can be created. Further informations can be found in the chapter ***Material Manager***, ***Create User defined Materials***.

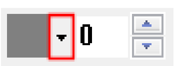
Materials

For the display of solids in 3D space colours, material properties or textures can be selected. Material properties are defined as constant colours together with lighting properties and can be assigned to solids as surface properties. You can select from the standard materials or create your own materials in the Material Manager.
The standard materials delivered with the program cannot be changed, but their values can be used to create own materials. Further informations can be found in the chapter ***Create User defined Materials***.

After the selection of a material, the button ***Define Solid Colour*** will display the diffuse part of the light of this material.

Select Material for New Solids

To create new solids already with a material, do the following:

1. Activate a 3D window.
2.  In the 3D symbol bar click to the marked icon to open the material list.
3. Select the desired material from the list.

4. Move the mouse cursor in the graphic area. The material list will be closed.

Newly created solids will get the material selected before. If a material is selected the colour selection icon is marked crossed. If the mouse cursor is moved upon the icon the current material is shown. When selecting a material with a transparency this value is displayed in the field ***Transparency of Solids***.
If in this icon a colour is displayed, the solids will be created in this colour.

Assign Materials to Existing Solids

To assign or edit a material on an existing body follow these steps:

1. Start the command ***Process***, ***Object Display***.
2. Identify a solid or collect several with the pressed ***Ctrl*** key or select several by dragging a window.
3. In the dialogue window ***Object Display*** in the tab ***Solid Display*** the material list can be found beside ***Colour / Reflection***.
4. If more settings should be done in the dialogue window click the button ***Accept***, otherwise click the button ***OK***.

Please note:

For the display properties of solids a list of RAL colours is available. This colours can be defined for new solids as well as for existing solids. The list shows the numbers and also the name of the colour. RAL colours are described by a diffuse light part.

Edit Material Display Properties

The way how lights will effect a shaded model can be edited by changing the material display properties. Material display properties are independent from the colour. Also the control elements in the dialogue window ***Object Display*** can be combined with the lights in a model to create various display effects.

Edit the Material Display Properties of a Solid

1. Start the command ***Process***, ***Object Display***.
2. Identify a solid.
3. If no material is assigned to the solid yet, select a material from the list in the dialogue window ***Object Display*** on the tab ***Solid Display***.
4. Click the button ***Accept***.

5. The material display properties can be defined in the tab Material:
   - **Shininess:** Defines the property of a surface, to reflect light.
   - **Transparency:** Defines the transparency of the material.
   - **Light Emission:** Describes, in which colour the light is reflected.
   - **Ambient Reflectance:** Describes the object colour and simulates the ambient lighting by the environment.
   - **Diffuse Reflectance:** Describes, who the light is reflected from the surface in all directions.

   Specular Reflectance: Describes, in which intensity and colour a solid reflects the light.

   Further informations concerning the different colour models (RGB, HSV and HSL) can be found here.
6. To take over the settings, click the button ***OK***.

User Defined Materials

In addition to the delivered materials, user defined materials can be created. To create a user defined material, follow these steps:

1. Start the menu command ***Settings***, ***3D Display*** and select the tab ***Material Manager***.
2. Click the button ***Create*** or start the context menu by clicking the right mouse button in the material list and select the command ***New Material*** (Ctrl+N). Enter a name for the new material.
3. To use the properties of an existing material, select the standard material from the list below the preview.
4. Define the material properties for the new material:
   - **Shininess:** Defines the property of a surface, to reflect light.
   - **Transparency:** Defines the transparency of the material.
   - **Light Emission:** Describes, in which colour the light is reflected.
   - **Ambient Reflectance:** Describes the object colour and simulates the ambient lighting by the environment.
   - **Diffuse Reflectance:** Describes, who the light is reflected from the surface in all directions.
   - **Specular Reflectance:** Describes, in which intensity and colour a solid reflects the light.

Further informations concerning the different colour models (RGB, HSV and HSL) can be found here.

5. To take over the settings, click the button ***OK***.

Get material from graphic

You can copy the object colour or material property assigned to a solid from the graphic and save it in the material list.

1. Select the menu command ***Settings, 3D Display*** and activate the ***Material Manager*** tab.
2. Click on the button ***Pick Material***.
3. In the 3D window identify a surface whose display properties are to be adopted.
4. Select ***New material*** in the context menu of the material list and click the ***F2*** key.
5. If desired, you can adjust the display properties of the new material.
6. Confirm the settings with ***OK*** or ***Accept***.

Please note:

User defined materials are stored in the USER directory in the file Material.DAT.

Process solid display

If the display attributes or the layer of one or more solids (fixed solid) is to later be modified, follow this procedure:

With selection list

1. Accept the solids to be processed in the selection list.
2. Load the menu command ***Process, Object Display***.

Without selection list

1. Load the menu command ***Process, Object Display***.
2. Accept the solids to be processed in the action list.

The rest of the procedure is the same in both cases:

The ***Object Display*** dialog box appears. The display properties of solids can be edited in the different tabs. Further informations on the different tabs can be found in the following chapters.

Solid Display

After identifying one or several solids using the command ***Process***, ***Object Display*** the dialogue window ***Object Display*** is offered. In the tab ***Solid Display*** it can defined in the same way as with the menu command ***Settings, 3D Display***, which display properties should be used for all previously selected solids.

In addition the following options are available:

Remove colours from solid faces
If this option is active, all solid faces which have an individual colour, Material Display or Texture allocated to them will be displayed in the Object display of the selected solid.

Layer
In this field, the layer on which the solid lies is displayed. Using the layer attributes, you can control whether a solid is active/not active or visible/hidden. If you wish to move the solid on to a new layer, proceed as follows:

1. Use the  icon at the end of the line to open layer selection.
2. From the list, select the layer on to which you wish to move the solid. If the layer does not yet exist, firstly set up a new layer in the layer manager.
3. Accept the modification with ***OK***.
4. You can use ***Accept*** to update the display of the solids that are selected at that moment without your having to close the dialog box first.
   The solids that have been selected are displayed in the selection colour and so a change of colour will not become effective until you have clicked *OK* to close the dialog box.

In addition to the ***Solid Display*** option card that corresponds broadly to the one for ***Settings, 3D Display***, you can also use the ***Solid faceting*** option card:

Solid faceting

After identifying one or several solids using the command ***Process***, ***Object Display*** the dialogue window ***Object Display*** is offered. Select the tab ***Solid Faceting***. This option card can be used to specify the accuracy for displaying shaded solids with curved surfaces. The setting will not have any effect on the accuracy used to manage the geometry data relating to the solids.

Any increase in the accuracy will need more time for the calculations to be completed and so you can use the settings here as an acceptable compromise between calculation time and display accuracy.

The settings that you choose will have an identical effect on all the solids no matter what size.

Mode

Select any of the modes offered here (***very fine, fine, standard, rough***), or select the alternative mode ***User defined***, if you want to specify your own parameters.
in this case you will be offered the parameters applicable to the mode most recently chosen.

user defined
If this faceting mode has been chosen, you can specify your own values for the parameters. They are divided into general faces and Spline faces that, for example, occur with free-form faces and with rounded areas with indentations.

The program will only observe the settings that have been activated by their respective boxes.

normal deviation

The value entered here will have an effect on the accuracy used to approximate the curved surfaces by means of level faces.
If the difference between the perpendiculars running to a level approximation face and the perpendiculars running to the curved solid surface is greater than the displayed value, the number of approximation faces and thus the accuracy level will increase.
The value entered here will have a direct effect whatever the size of the solid.

grid ratio

This entry will have a direct effect on the type of grid on which the lines will restrict the level approximation faces for the curved surfaces. The box must be active.

The entry here is a ratio which, depending on the accuracy, can be used to influence how regular rectangles or triangles are formed. The less the deviation from the standard value of 1, the greater the number of triangles that the program will compute for an approximation. In the case of a large deviation the number of triangles will increase.
The entry must be a positive number. An entry greater than 1 will have the same effect as values less than 1, if one of the entries is the reciprocal of the other, e.g. 0.1 and 10.

The value entered here will have a direct effect whatever the size of the solid.

surface deviation

This value will have an effect on the accuracy used to approximate curved surfaces by means of level faces.
Ii the spacing between a level approximated face and the curved solid surface is more than the value specified the number of approximated faces, and thus the accuracy, will increase.
The precise effect of the parameter will depend on the size of the solid. Moreover a considerable amount of time will be needed to complete the calculations.

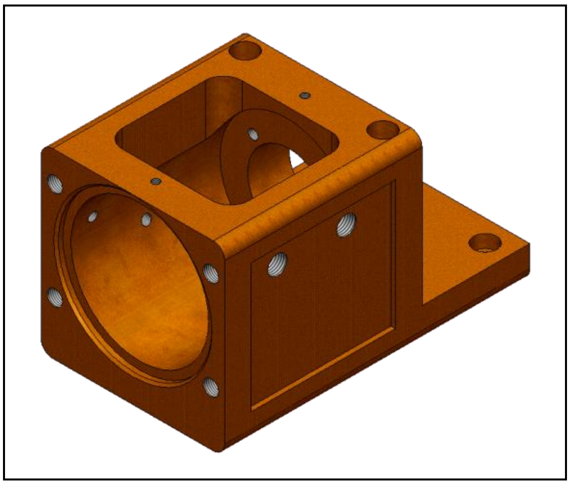
maximum edge length

This entry defines the maximum edge length of the level faces used to approximate an uneven face.
The precise effect of the parameter will depend on the size of the solid. Moreover a considerable amount of time will be needed to complete the calculations.

Material

After identifying one or several solids using the command ***Process***, ***Object Display*** the dialogue window ***Object Display*** is offered. In the tab ***Material Manager*** the material properties of the previously selected solids can be edited. Further informations can be found in the chapter Edit Material Display Properties.

Textures



Textures can be assigned to solids or single surfaces to get a realistic view. Any pixel image can be used as a texture. The supported formats are BMP, GIF, JPG, PNG, PPM and TIF. Textures used in a model are stored with the model, so that the model stays consistent when transporting to another desktop.
The realistic view can be supported by adding also material display properties.

Assign a Texture to a Solid or a Surface

1. To assign a texture to a solid start the menu command ***Process***, ***Object Display***.

To assign a texture to a surface start the menu command ***Process***, ***Solid Face Display***.

2. Identify the solid or the surface.

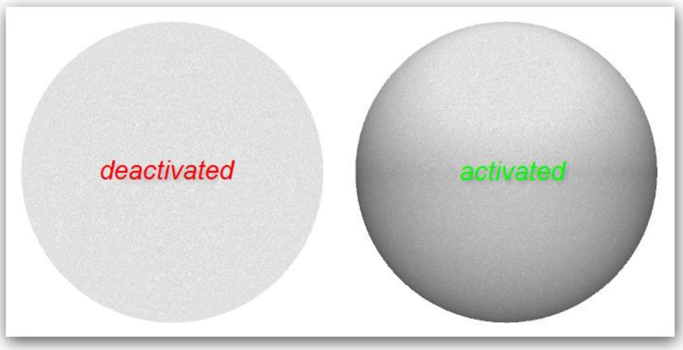
3. Select the tab ***Texture***.

4.  Click the icon ***Open Image*** and select the pixel image for the texture.
Besides the standard textures any pixel image in the supported formats can be selected.

5. For the selected texture the following settings can be made:

Relative Coordinates
If this option is active, the texture covers the complete surface, if the scale is set to one.
If this option is deactivated the texture will get the size auf 100 x 100 in model units and is only displayed once, if the option ***Repeat*** is also deactivated.

Modulate Colour
If this option is deactivated, the texture is displayed in the original colour.
Activate this option, to mix the texture with the base colour of the solid.
For a realistic view, the base colour of the solid should be white. Select grayscales to darken the texture. Using different colours then white, will colour the texture.
Example:



Repeat
If the option ***Relative Coordinates*** is deactivated and the option ***Repeat*** is activated, the texture is multiplied to cover the complete surface.

When deactivating the option, the texture is displayed only once on the surface.

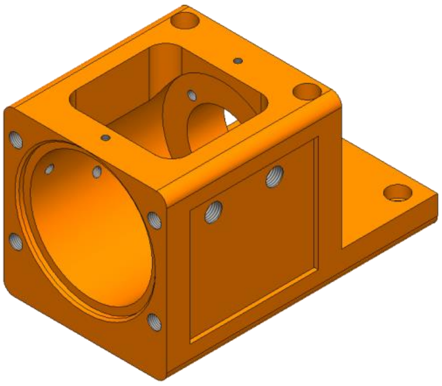
Scaling Homogeneous
If this option is active the texture will keep the ratio between length and height if the texture is scaled in X or Y direction.

Transform Settings
With the transform settings the texture can be ***scaled***, ***mirrored***, ***rotated*** and ***moved*** on the surface.
The result is displayed directly in the 3D window.
The transformations can be changed by entering appropriate values or by moving the sliders.

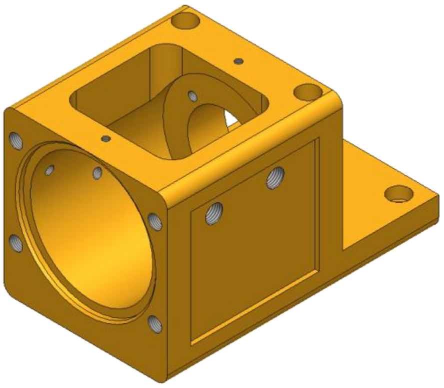
Notes for the sliders
If in the entry field a value is entered which is outside of the scale, the scale will be adjusted automatically to this new value. To resize the value again, enter a value for the maximum and the a value for the minimum and press the ***Ctrl*** key while confirming the entry with ***Enter***. If the upper or lower boundary of the scale is changed will be decided by the current position of the slider. If he is in the left half, the lower boundary will be adjusted.

Erase Texture from Solids

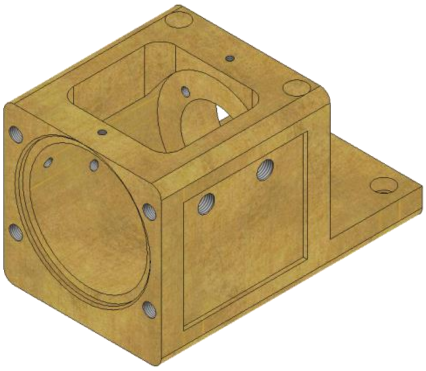
To erase a texture from a solid, in the dialogue window ***Object Display*** select in the list of textures the upper empty line and click the ***Accept*** button.



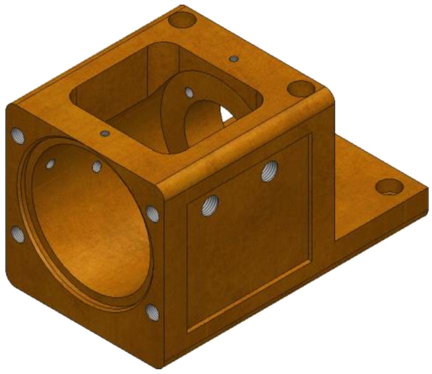
Displayed in
Solid Colour



Displayed with
Material



Displayed with
Texture



Displayed with
Texture and Material

Process display of individual solid faces

After solids have been generated, all their faces are displayed in the same representation type.

If solids are displayed as shaded, the colour, the material display properties, the texture, the degree of transparency and the reflection attributed can later be modified.

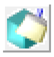
1. Select the menu command ***Process, Solid Face Display***.
2. Use the ***3D Face Selection*** for the selection of surfaces, where you wish to change the display.
3. In the dialog box which then appears, define the required colour, the degree of transparency and, if necessary, the reflection attribute of the face. In the tab Material Manager and Texture also settings can be made for the surface.

Please note:

If the areas are to be displayed with their original display propertied, this menu command can be used for individual areas. If this is to be valid for all areas or a number of solids, use the menu command ***Process, Object Display***. Activate the option **Remove colours from solid faces** on the the ***Solid Display*** option card.

 After clicking the displayed icon the command ***Select all Solids with Face Colour*** is started. All the Solids with face colours will be selected. This command can be used from the user defined Toolbar or from a user defined menu.

Transfer 3D Object Display

 Using the command ***Process***, ***Transfer 3D Object Display*** all the settings defined in the ***Object Display*** dialogue window like solid colour, material display properties, density, transparency and texture can be transferred from one solid to another one.

1. Identify a solid, whose properties should be transferred to other solids.

2. Identify by single clicks or by defining a window with the pressed left mouse button, the solids, to which the properties should be transferred.

Transfer Solid Face Display

 To transfer the display properties like colour, material or texture from one surface to other surfaces use the menu command ***Process***, ***Transfer Solid Face Display***.

1. Identify the surface, whose display properties should be transferred to other surfaces.

2. Use the ***3D Face Selection*** for the selection of surfaces, which should get the display properties of the first identified surface.

5.11 Solid information

 The commands concerning information are displayed after clicking on this icon.

These commands can be used to display information concerning solid data.

Collision check of solids

  The command ***Information, Interference check between 2 Solids*** for collision checks can be loaded using the displayed icon.

This command checks whether two solids collide and therefore have a common volume.

1. Identify the first solid.

2. Identify the second solid.

It is then displayed whether a collision exists for the two solids. If there is no collision the information is displayed if the solids touch or which minimum distance they have to each other.

Interference Check for Assemblies

, After clicking the displayed icons, the command ***Information***, ***Interference check for assemblies*** is started.

It will be checked if there is an interference of solids in their current position and in case there is at least one interference, all other solids will be hided.

1. Select the solids to be checked by dragging a window or by selection with the pressed ***Ctrl*** key.
2. The result of the check is prompted shortly in the Prompt line.
3. In case their is at least one interference, all other solids are hided.
4. By clicking the **A** key while the command is running or afterwards by clicking the icon ***Start Hide Action***, the other solids will be visible again.

Solid calculations

In order to be able to display solid data, use this command in the toolbar.

1. Start the command ***Information, Solid Calculations***.
2. After starting the command identify a single solid
   or
   identify one after another solid
   or
   select the solids by dragging a window.

A dialog window will display the calculated values for the selected solids.

While the dialog is open, further solids can be integrated or excluded to the calculation by identifying them.

Solid calculations dialog box

This dialog box displays the values determined for the selected solid.

The following options are also available:

Option card - General

Display solid centre of gravity

 Click on this icon, in order to identify the centre of gravity of the solid with a marker.
Once the drawing is redrawn, the identification marker is removed.

In addition to the temporarily display of the centre of gravity with a Marker, a Sphere can be generated at this point.

 Define the diameter of the Sphere in the tab ***General*** and click the displayed icon ***Create Sphere in the Centre of Gravity***. A sphere with the current display properties will be created in a group with the name ***Centre of Gravity*** in the root directory ***3D Solids***.

Regard Density

If a material resp. a density is assigned to the solids using the menu command ***Process***, ***Object Display*** these are taking in account when calculating the mass or the moments. For all solids without a material a density of 1 is assumed.

Calculate weight

If a material resp. a density is assigned to the solids using the menu command ***Process***, ***Object Display*** these are taking in account when calculating the mass or the moments. For all solids without a material a density of 1 is assumed.
If no material was assigned before, a material can be selected from the list or a density can be entered. The properties, assigned in this dialog window are not stored with the model and are lost when leaving the command.

Save values in a file

 Click on this icon.
In the dialog box that then opens, enter the directory and name of the file. It has the extension GEO – as long as no other is defined.

Option card - Moments

Enter vector

 Click on this command button in order to generate the vectors as solid axes.

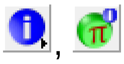
Option card - Inertia

The inertia tensor is a matrix display of the mass distribution of a rigid solid referenced to a certain coordinate system. Its diagonal elements $\mathbf{I_{ii}}$ are called **moment of inertia**, the non-diagonal element $\mathbf{I_{ij}}$ **deviation moments**.

By a main axis transformation, the inertia tensor can always be brought onto the diagonal shape. The axes (**1,2,3**) of this system are called **principle inertia axes** and run through the centre of gravity of the solid. The diagonal elements correspond to the **principle moments of inertia I_1, I_2, I_3**.
With a rotation about the principle inertia axes, the rotation impulse and the angle speed are parallel.

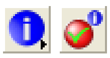
The inertia tensor is calculated based on the global coordinate system. To calculate the inertia tensor based on a different coordinate system the solid has to be transformed in a way that the relative coordinate system and the global coordinate system are identical.

Calculate a Surface Area

 After clicking the displayed icons the command ***Information***, ***Calculate Surface Area*** is started.

The command calculates the area of a surface and displays the calculated value in the status block window.

Check solid data

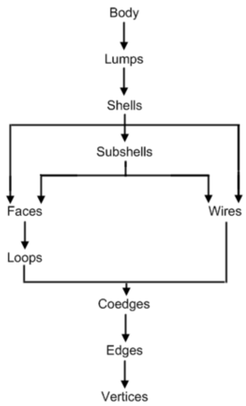
 In order to check solid data, use this icon in the information toolbar.

This command checks whether solids are managed correctly in memory according to the ACIS conventions.

1. Starts the command ***Information, Check Solid***.
2. If solids exist in the selection list, these are checked.
3. If the selection list is empty, the following options can be used for the check. Identify afterwards the individual solids or a number of solids which are to be checked.

It is recommended to check solids with the default settings in the dialogue window !

Make the desired settings and identify the solids to be checked.



ACIS-Topologie

Check level

Defines the number of topological levels of ACIS solids. The range can be defined between 10 and 70 in increments of 10. The standard value is 20. The more levels are checked the longer it takes. The number of levels is independent from the significance of the check. Only the performance is reduced to check the solids.

Check limit

This option allows to stop the check, if a defined number of errors is reached. With the default value 0 all errors of a solid are reported. This option is useful, if the solid contains a lot of errors. By a limitation of errors reported the search algorithm in complex models is accelerated.

Select Solid …

Choose here, if the solid should be selected ***on error***, ***on warning***, ***on note*** or ***on info***.
A selected object can easily be found even in complex models.

Abort on first problem

If this option is active, the check is stopped if a first error is found. This option allows the quick check of several solids, if it is only used to find defect solids, without qualifying the problem. Default is ***not active***.

Check face–face intersection

If this option is active, independently from the current check level the intersection of faces is checked. This check is done otherwise only in check level 70, if this option is deactivated. Default is ***not active***.

Check discontinuity information

Activates the check for curves and surfaces of discontinuities.
Default is ***not active***.

Check edge surface coincidence

This option allows a detailed check, if the edges of a faces are coincident with the surface of the solid.
Default is ***not active***.

Increase level of reported details

Activate this option to get a detailed report of all recognized notes,

warnings and errors.
Default is ***not active***.

Perform self intersection tests

This option checks solids in regards of discontinuities as well as of self intersections. In many cases the check for self intersections is not requested due to performance reasons, if irregular faces were already found. In this case this option should be deactivated.
Default is ***active***.

The ACIS specific data (number of partial solids, edges, faces, etc) and, if necessary, existing errors of the selected solid are determined and saved in a file which is then displayed in the list editor.

If the solid contains errors, it should be erased in order to prevent inconsistencies in the model.

Please note:

, You must also delete those solids that are without any geometry. Use ***Information, Check and Clean Model*** to do this.

Information about Distance and Angle of Faces

, By clicking the displayed icons in the 3D window the command ***Information***, ***Distance / Angle between Faces*** is started.

The minimum distance between faces is determined and the angle between Surface normals. and is displayed in the status block dialogue window.

As an additional information the dX- dY- and dZ-coordinates are displayed. This coordinates refer to the 3D global coordinate system.

1. Start the command by clicking the displayed icons.

2. Identify the first surface.

3. Identify the second surface.

The calculated values are displayed in the status block dialogue window.

Please note:

The angle can only be determined between planar surfaces.

Simplify Solid Geometry

Imported solids as well as solids getting complex surfaces from solid operations can be simplified with this command to enhance the performance. Spline surfaces are replaced by analytical surfaces if possible.

1. Start the command using the menu command ***Extras***, ***Simplify Geometry***.
2. Identify a solid or multiple solids with the ***Ctrl*** key pressed. Unlock the ***Ctrl*** key before identifying the last solid.
   Multiple solids can also be selected by a selection window touching the relevant solids.

Repair Solid Geometry

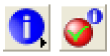
Solids displaying errors often can be repaired with this command.

Before using this command the solids should be simplified using the command ***Extras, Simplify Solid Geometry.***

Solids displaying errors using the command ***Check Solid*** can be repaired using the command ***Repair Solid Geometry***.

1. Start the command using the menu command ***Extras***, ***Repair Solid Geometry***.
2. Identify a solid or multiple solids with the ***Ctrl*** key pressed. Unlock the ***Ctrl*** key before identifying the last solid.
   Multiple solids can also be selected by a selection window touching the relevant solids.

Please note:

 To check a solid the menu command ***Information***, ***Check Solid*** can be used. After identifying one or multiple solids a list is displayed where errors are marked with the headline *****ERRORS***** .

5.12 Insert and export solids

from ***BeckerCAD 3D***

In *BeckerCAD*, you can use the following menu commands to insert solids that are in a SAT file into your current model: ***File, Import, SAT File, Position Freely*** or ***With Absolute Position***.

For export, the menu command ***File, Export*** can be used. This can be used to save solid date in files of formats SAT, VRML and STL (from *BeckerCAD 3D pro*).

Insert fixed solid

from ***BeckerCAD 3D***

Using the menu commands ***File, Import, SAT File, Position Freely*** and ***With Absolute Position***, fixed solids which have been saved in files of SAT format can be loaded and inserted in the 3D view windows. You have the choice of positioning these anywhere in the model or at their original position in the global coordinate system.

In the SAT import it can be defined if groups should be created for each solid as follows:

1. Activate the 3D window
2. Open the ***File***, ***Import***, ***Settings*** dialog box.
3. Choose in the dialog between the following possibilities:

   Group solids during Import:

   Without group – no groups will be created during the import of SAT files. All solids are placed in the active group.

   One group – all solids inside the SAT file are imported into one group. The name of the group is the name of the SAT file.

   Subgroups – one group is created with the name of the SAT file. For each solid in the SAT file a subgroup is created with the name of the SAT file followed by a number.

Insert fixed solids with free positioning

from ***BeckerCAD 3D***

In order to insert fixed solids contained in SAT files at any position in the current model, proceed as follows:

1. Select the menu command ***File, Import, SAT File*, *Position Freely***.
2. Select the required file. Specify another drive and folder, if necessary.
   Confirm this using ***OK***.

   The solids contained in the file are loaded and represented by a surrounding box.

The position of the solid corresponds to the position in reference to the GCS when saving the file.

3. Define the position of a corner of the box.
If the 2D GCS or the 2D LCS is active, this point is projected in the current WP.
If the 3D GCS or the 3D LCS is active, this point can also be defined as a spatial point.

Please note:

A SAT file with free positioning can also be inserted using Drag & Drop.

If solids in other file formats should be inserted, the menu command ***File***, ***Import*** is used. When the optional converter for this format is installed, it can directly been readen into the current session; otherwise it should be converted first into the SAT file format.

In the SAT Import it can be defined if groups should be created for each solid as follows:

1. Activate the 3D window

2. Open the ***File***, ***Import***, ***Settings*** dialog box.

3. Choose in the dialog between the following possibilities:

Group solids during Import:

Without group – no groups will be created during the import of SAT files. All solids are placed in the active group.

One group – all solids inside the SAT file are imported into one group. The name of the group is the name of the SAT file.

Subgroups – one group is created with the name of the SAT file. For each solid in the SAT file a subgroup is created with the name of the SAT file followed by a number.

Insert fixed solids with absolute position

from ***BeckerCAD 3D***

In order to insert fixed solids contained in SAT files with their original position within the global coordinate system into the current model, proceed as follows:

1. Select the menu command ***File, Import, SAT file, With Absolute Position***.

2. Select the required file. If necessary, determine another drive and folder.
Confirm using the command button *OK*.

Please note:
A SAT file can also be inserted by Drag&Drop.

If solids in other file formats should be inserted, the menu command ***File***, ***Import*** is used. When the optional converter for this format is installed, it can directly been readen into the current session; otherwise it should be converted first into the SAT file format.

In the SAT Import it can be defined if groups should be created for each solid as follows:

1. Activate the 3D window

2. Open the ***File***, ***Import***, ***Settings*** dialog box.

3. Choose in the dialog between the following possibilities:

Group solids during Import:

Without group – no groups will be created during the import of SAT files. All solids are placed in the active group.

One group – all solids inside the SAT file are imported into one group. The name of the group is the name of the SAT file.

Subgroups – one group is created with the name of the SAT file. For each solid in the SAT file a subgroup is created with the name of the SAT file followed by a number.

Export fixed solid

from ***BeckerCAD 3D***

You can use any solid generated in a model in another model provided that you have saved it as a SAT file.
You can use the following commands to insert whatever you want in the new model: ***File, Import, SAT File, Position Freely*** or ***With Absolute Position***.

The solids that have been generated in *BeckerCAD* can be used for further tasks in another CAD system or application, provided that files have been saved in one of the formats SAT, VRML or STL (from *BeckerCAD 3D pro*). They are saved in their position and orientation within the GCS.

Specify settings for importing und exporting fixed solids

The following settings can be specified for the import and export of SAT files and STL files:

SAT output format

Determines in which format SAT files should be written when executing the menu command ***File, Export, SAT File***.
This is necessary if solid data is to be read into the program which can only interpret earlier ACIS versions.

Independent of this setting, when saving models, solid data is entered in the respective MOD file according to the current ACIS version.

Group solids during Import:

Without group – no groups will be created during the import of SAT files. All solids are placed in the active group.

One group – all solids inside the SAT file are imported into one group. The name of the group is the name of the SAT file.

Subgroups – one group is created with the name of the SAT file. For each solid in the SAT file a subgroup is created with the name of the SAT file followed by a number.

Stereolithographic settings (from ***BeckerCAD 3D pro*****)**

In these text boxes, settings for the output of STL files can be specified which can be saved using the menu command ***File, Export, STL File***. In order to display the results of the selected settings, select the menu command ***Process, Object Display***. Define ***stereo lithography*** for ***face display***.

Normal deviation
This value must be entered. It affects the exactness with which curved surfaces can become even approximated faces.
If the angle difference between the perpendiculars to an approximated face is larger than the entered angle value, the number of approximated faces and therefore the degree of exactness is increased.
The predetermined standard value is 15°

surface deviation
This value can be entered. If it is to be taken into account, it should be activated using the respective check box.

This value also affects the exactness with which curved surfaces can become even approximated faces:
If the distance between an even approximated face and the curved

surface is larger than the entered distance value, the number of approximated faces and the degree of exactness is increased.

If normal deviations and face deviations are entered at the same time, the precise approximation is calculated.

grid ratio
With this value, the form of the grid can be influenced whose lines borer the even approximated faces for curved surfaces. If it is to be taken into account, the respective check box should be activated.

This value is a relational value over which - depending on the exactness - the form of even rectangles or triangles can be influenced. The smaller the deviation is from the standard value 1, the more even rectangles will be calculated for the approximation. With larger deviations, the number of triangles increases.
Only positive values can be entered. With this, values larger than 1 give the same result as values smaller than 1, if one is the reverse value of the other, e.g. 0.1 and 10.

Export SAT file

from ***BeckerCAD 3D***

The menu command ***File, Export, SAT File*** can be used to save an individual or number of solids from the current 3D view window as a file in SAT format. The file has the extension SAT.

This is created in the **ACIS Version 6.0**. If the file is to be created in an older or newer ACIS format, firstly select the required SAT output format using the menu command ***File, Export, Export Settings***.

The following options are available for selecting the solids to be exported:

Select solid using selection list

1. **Before** starting this command, accept the solid which is to be saved as an SAT file in the selection list.
2. Start the command and determine the folder and file name in the dialog box which then appears.
3. The file is saved after selecting ***OK***.

Select solid using action list

1. If the action list is empty, accept the solids which are to be saved as SAT files into the action list:
2. In the dialog box which then appears, determine the folder and file name.

3. The file is saved after selecting ***OK***.

Please note:

A SAT file saved with this command can be loaded using the menu command ***File, Import, SAT File*** and positioned as a fixed solid in the 3D view window.

Export VRML file

from ***BeckerCAD 3D***

Using the menu command ***File, Export, VRML File*** solids from the 3D view windows can be saved as a file in VRML format. The file has the extension WRL.

The following options are available for selecting solids which are to be exported:

Select solid using selection list:

1. Accept the solid which is to be saved as a VRML file into the selection list **before** loading the command.

2. Start the command and determine the folder and file name in the dialog box which then appears.

3. The file is saved after selecting ***OK***.

Select solid using action list:

1. If the selection list is empty, accept the solid into the action list which is to be saved as a VRML file:

2. In the dialog box which then appears, determine the folder and file name.

3. The file is saved after selecting ***OK***.

A VRLM file can, for example, then be opened in Netscape Navigator 3.0 (with installed VRML-Plug-In) and animated by the user.

Export stereo-lithography file

from ***BeckerCAD 3D pro***

The menu command ***File, Export, STL File*** can be used to save the solids of the current 3D view window as a file of STL format. The file has the extension STL.

The output is carried out according to the settings which are determined using the menu command ***File, Export, Export Settings***. In order to obtain a preview, start ***Process, Display Object*** and then specify the ***Stereo-Lithography*** option for the ***Face Display***.

The following possibilities are available for selecting solids which are to be converted:

Select solid using selection list:

1. Accept the solid which is to be saved as a STL file into the selection list **before** loading the command.
2. Start the command and determine the folder and file name in the dialog box which then appears.
3. The file is saved after selecting ***OK***.

Select solid using action list:

1. If the selection list is empty, accept the solid into the action list which is to be saved as a STL file.
2. In the dialog box which then appears, determine the folder and file name.
3. The file is saved after selecting ***OK***.

A STL file can, for example, then be implemented at a stereo lithography language workstation for 3D modelling of synthetic prototypes.

Render 3D graphics

from ***BeckerCAD 3D***

Another way, besides the ones in *BeckerCAD* to assign Material display properties and Textures, to specify a display of your 3D model in such a way that it will be almost indistinguishable from a photograph is the export of previously selected solids or the complete 3D model as a TGF file.

TGF files, exported from *BeckerCAD*, can be loaded and processed in render programs, for example GAMMA-RAY.

If you have used different colours for the solids/the faces of the solids by using ***Process, Object Display*** or ***Solid Face Display***, they will be given different codings.

Without this step being completed first, you will be unable to use *GAMMA-RAY* to show all the faces with different codings in various textures.
First select ***Settings, Allocation to Code*** in the ***Material Selection*** dialog box in *GAMMA-RAY*.

3D Graphic Files

from ***BeckerCAD 3D***

Using the menu command ***File***, ***Import*** and ***Export*** now ***3D Graphic Files*** can be loaded into a model or saved from within a model. 3D Graphics files do not contain CAD data and cannot be edited; also model views cannot be derived from this data. But they do have the advantage, that because of the missing CAD data they need less memory for example to add complex 3D graphics for the presentation of your 3D models. Another advantage is, that you can get millions of 3D graphics via the Internet, part of them even for free. 3D models can be imported into a model directly as 3D graphic. Also different standard 3D graphic formats are available.
3D Graphic files can also be imported using the menu command ***Extras, 3D Graphic Files***. This menu offers additional commands to select or replace 3D Graphic Files in a model.

3D Graphic Files can be positioned, moved, rotated, scaled and mirrored using the standard commands from the Transform menu, with or without a copy. Points, like end points or the midpoint of edges cannot be snapped. In this snap options used on a 3D Graphic File always the origin, projected to the current work plane, is used.

3D graphic objects in BeckerCAD are based on data from OpenSG. The most powerful and therefore preferred file format for 3D graphic objects is the OSB format (*.osb).

Notes for the OBJ file format

The **OBJ Format** (or **.obj**) is an open file format to store geometric 3D data. The file format developed by *Wavefront Technologies*© is supported from multiple 3D graphic programs and therefore often used for the exchange of data.
In the OBJ Format the geometric characteristics of a 3D scene is stored, the material definition is stored in a separate file with the extension **.mtl** (*material*).
Please note the hints for the Import and Export of OBJ files in the following chapters.

Import 3D Graphic Files

from ***BeckerCAD 3D***

The following file types can be imported as 3D Graphic File:

Import Formats

BeckerCAD 3D model	*.MOD	
Virtual Reality Modeling Language	*.WRL	
Open SG binary	*.OSB	
Open SG ascii format	*.OSG	
Wavefront [1]	*.OBJ	(*.mtx material file of *BeckerCAD*) (*.mtl material file generaly)
Objekt	*.OFF	
Collada scene	*.DAE	
Stereolithographie CAD	*.STL	
Open Flight	*.FLT	
3D Studio 3.x/4.x	*.3DS	
Standard PLY	*.PLY	

When loading a 3D Graphic File into a model, the respective graphic data will be stored with the model. This enhances the performance of the calculation of identical 3D Graphic Files tremendously. Using the option ***share existing scene item*** in the import dialogue window assures that identical files are referenced. In this case select ***model*** in the ***scene object with same name*** area.
If ***file system*** is marked here, a completely new data set is created for the imported 3D Graphic File. This option should only be used to import 3D Graphic Files with the same name but with different contents.

To insert a 3D Graphic File into a model follow these steps:

1. Select the menu command ***File***, ***Import***, ***3D Graphic File*** or ***Extras***, ***3D Graphic***, ***Import***.
2. Select the desired file.
3. Activate the option ***simplify on file load***, if this file should be placed in the model very often. The data structure tree of the 3D graphic file will be reduced then, to be able, to load a big number of the same files into the model.
4. A 3D preview of the 3D Graphic File can be activated using the option ***show preview***. Please be aware, that when selecting big models it takes some time and memory to generate the preview.
5. If the model already contains 3D Graphic Files with the same name you can decide, after activating the option ***share existing scene item***, if the data of the file should be taken from the model (better performance) or if a new data set should be generated. Click on the appropriate list entry on the left side of the preview window.
6. Click the ***Open*** button.

7. By pressing the **X** key you can select between the different insert points: the absolute insert point (= origin of the local coordinate system of the file), the centre of the surrounding box or one of the box corner points.
   Please note: When switching the insert point, this does not affect the local coordinate system of the 3D Graphic File.
8. Press the up / down arrow keys to scale the 3D Graphic according to the value ***Scale*** in the status dialogue window. Alternatively a value for the ***Factor XY*** can be entered in the status dialogue window.
9. Place the 3D Graphic by cursor or by entering the X / Y coordinates in the status dialogue window.

Please note:

(1) During the import, primarily the .mtx will be read and only in the case there is no .mtx file the .mtl file is used.
In the case, that OBJ files, created by other programs, appear overexposed after the import, the value for 'ImportAttenuationFactors' in the file *OpenGL.ini* (program-directory) should be changed from 1.0 to 0.25. A restart of the program is necessary afterwards.
When exporting OBJ files, an additional mty material file is created. This corresponds to the mtl format, but contains *BeckerCAD* specific additions. This file is required for the WEB-Export.

Export 3D Graphic Files

from ***BeckerCAD 3D***

Using the menu command ***File***, ***Export***, ***3D Graphic File*** single or several solids can be exported as 3D Graphic Files in the following formats. The export command can also be executed from the menu command ***Extras***, ***3D Graphic***, ***Export***.

Export formats		
Open SG binary	*.OSB	(Proprietary, prefered format)
Open SG ascii	*.OSG	
Virtual Reality Modeling Language	*.WRL	
Wavefront (1)	*.OBJ	(*.mtx material file of *BeckerCAD*) (*.mtl material file generaly)

Please note:

3D Graphic Files are exported with their current position in relation to the global origin. Maybe it is useful, to move the solid with a defined point to the global origin before the export. This point is used as insert point (local coordinate system) for the 3D Graphic when importing again.

To define the 3D objects to be exported the following possibilities are offered:

Define using the selection list

1. **Before** starting the command insert the desired 3D objects into the selection list.
2. Then start the command and enter the file name, the file type and the folder.

Follow the further steps with point 3.

Define using the action list

1. If the selection list is empty, identify the desired 3D objects. Several objects can be selected with the pressed Ctrl key or by dragging a window.
2. Enter the file name, the file type and the folder in the offered dialogue window.
3. Activate the option ***save scene including lights*** if the current light settings Should be saved with the file.
4. If the option ***simplify on file save*** is active, the data will be stored reduced in the 3D Graphic File. This option should only be used, if this 3D Graphic file will be used very often in different models.

   Please note:

   It is generally not recommended to activate this option. The option to ***simplify on file load*** should be used instead to avoid graphic problems with reduced 3D Graphic Files.
5. After clicking the ***OK*** button the file is stored.

Please note:

When exporting in the ***Virtual Reality Modelling Language (*.wrl)*** file format, the option ***save scene in compressed format*** is not supported. Files in this format cannot be compressed.
When exporting in the ***Open SG ascii (*.osg)*** file format, the option ***save scene inclusive lights*** is not supported. Scene lights cannot be stored in this file format.

(1) OBJ files, exported from *BeckerCAD* contain grouped geometries, which allows other programs to divide the single objects in a scene. The material file (*.mtl) is now also created in the same directory. As not all of the graphic attributes, e.g. textures, can be stored in

the mtl files, an additional file .mtx is also created in the same directory. This .mtx file can only be used by *BeckerCAD* .

Select 3D Graphic

from ***BeckerCAD 3D***

In addition to the standard commands in the ***Selection >*** icon bar 3D Graphics can be selected with the following options using the menu command ***Extras***, ***3D Graphic***, ***Select***. After starting the command select the desired option and identify a 3D Graphic in the model.

Global unique identifier

When identifying a 3D Graphic all 3D Graphics are selected, reference to the same data set. For example if the 3D Graphic was copied in the model or if during the import it was defined to use the data from inside the model.

File name

All the 3D Graphics are selected with the same file name (including folder).

Replace 3D Graphic

from ***BeckerCAD 3D***

To replace 3D Graphics with other ones also contained in the model follow these steps:

1. Start the command ***Extras***, ***3D Graphic***, ***Replace***.

2. Identify the 3D Graphic to be replaced. Several 3D Graphic objects can be collected with the pressed ***Ctrl*** key or by dragging a window.

3. Identify a 3D Graphic object, which should replace the other ones.

Replace 3D Graphic with

from ***BeckerCAD 3D***

To replace a 3D Graphic object by a new 3D Graphic File follow these steps:

1. Start the command ***Extras, 3D Graphic, Replace with ...***

2. Identify the 3D Graphic to be replaced. Several 3D Graphic objects can be collected with the pressed ***Ctrl*** key or by dragging a window.

3. In the dialogue window select the file, which should replace the other ones and click the ***Open*** button.
4. Press the up / down arrow keys to scale the 3D Graphic according to the value Scale in the status dialogue window. Alternatively a value for the ***Factor XY*** can be entered in the status dialogue window.
5. Click on any position to replace the 3D Graphic objects.

The new 3D Graphic object will be placed with its own origin on the origin of the 3D Graphic objects to be replaced. If the positions of the 3D Graphic objects where different in relation to their local coordinate systems (absolute insert point), maybe the newly imported 3D Graphic object will appear on a different position.

5.13 Printing a 3D model

You can print a 3D model using one of the Windows printer drivers that you have installed or save as pixel image. Only the visible 3D objects will be in the output.
The dimensions of the section to be printed can be defined in the respective dialogue window.

Please note:

If visible 2D lines in the 3D window should also be in the output, please use the command described here.

Print 3D objects

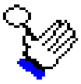
After clicking the menu commands ***File***, ***Print...*** with a 3D window active a dialogue window appears to print from the 3D window, start the print process and save the printer settings into the model.

Use this command to print the 3D objects as displayed in the preview window. If visible edges and silhouettes should also be in the output the following preconditions must be fulfilled.

In the preview window you can define the desired view respective to the current page size. To do so, you have the following options:

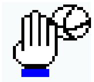
 **Move View**

Click in the preview window with the middle mouse button pressed. Move the mouse to move the view.



Zoom In / Zoom Out

Zooming can be done with the scroll button or by pressing the right mouse button and moving the mouse up and down.

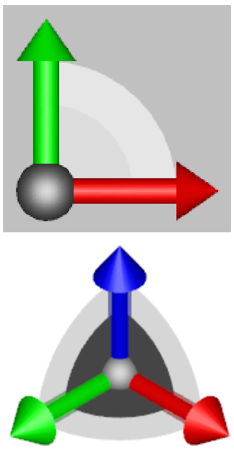
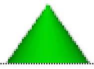
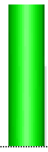
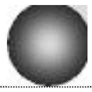
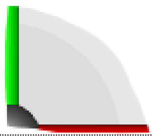
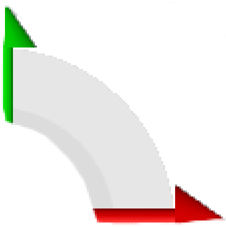


Rotate View

Click in the preview window with the left mouse button pressed. Move the mouse to rotate the view.

Display the interactive coordinate system

If this option is active, the view will rotate by a click on the following position....

		... in view direction of the arrow
		... by 90° (arrow shaft)
		.. displays all objects
		... in the perspective view (inner circle)
		... in a view from the front or from behind (outer circle)

Settings for print 3D objects

After clicking the menu commands ***File***, ***Print...*** with a 3D window active a dialogue window appears to print 3D objects from the 3D window, start the print process and save the printer settings into the model.

Current Printer

In this field the current printer is displayed.

In case you wish to print the 3D view to a different printer, you can make select a printer from this list or via the button ***Printer Setup***.

Number of Copies

Defines the number of copies.

Use Driver Optimization
If supported from the driver, the program will try to create one print job, which will be printed as often as given in *number of copies*. If the driver doesn´t support a copy optimization, the print job will be send as often as defined in *number of copies*.

DPI (dots per inch)
Input field to define the resolution of the points for the output. The higher the value, the better the quality of the output.

Portrait / Landscape
Select the desired option to print in portrait or landscape format.

Pixel Size
Displays the resulting size of the output in pixels according to the resolution DPI and the current paper size.

Maximum Print Area
Displays the size (in mm) of the maximum print area of the current printer corresponding to the selected paper size.
This value is taken from the Windows Print Control and cannot be edited.

Offset
Entry field to define the offset of the print area starting from the left and upper paper frame in mm. Positive and negative values are allowed here. Positive values will move the print area to the right or downwards. When choosing other units than mm, the unit must be entered after every value and confirmed by pressing the ***Enter*** key: 1cm x 0,02m

Paper Size
Selection list of the supported paper sizes of the current printer.

File Size
The file size is calculated from the resolution DPI and the selected paper size. The file will temporarily be stored in the TEMP directory, before transmitting to the printer.

 By clicking the displayed icon the actual view from the 3D window is taken over into the preview window.
Using an orthogonal view in the preview the view angle and the zoom is adapted, in a perspective view due to the different view modes only the view angle can be adapted.

Perspective View
When activated, the current view is printed in a perspective view instead of an orthogonal view.

Display Coordinate System
The interactive coordinate system allows to modify the view in the preview window. It will also appear in the output, when activated.

Antialiasing
This option is dependent on the computer hardware, especially from the graphic board and the installed drivers. With the antialiasing option the lines can be smoother. The higher the value the more the lines are smoothed.

3D Graphic
Allows the print of several 3D file formats, without loading the file before. After clicking this button, you can select a 3D object you wish to print.

3D Model
The view of a 3D graphic is closed and the current 3D model is displayed again.

Activate Color Gradient
Allows to place a gradient in the background for printing. The color will change from the first selected color to the second selected color. When this option is deactivated the background is printed in the given color.

Stamps
Specify further information for the printout, such as the name of the model or the current date/time. By activating the ***headline*** or ***footline***, the information selected in the adjacent list is also printed out.
You can extend the list of information to be output with your own attributes, for example, an attribute from the title block of a drawing in the relevant model. To do this, open the `3DPlot.ini` file located in the user directory with a text editor. The information to be output is defined using Python expressions.

Technical Information about 3D Stamp Printing

All user-specific settings for printing additional information in the 3D view are defined in the `3DPlot.ini` file in the user directory. For example, because update installations do not lose their own customizations.

The basic specifications for the additional information are loaded from the 3DPlot.ini file of the same name, which is located in the *BeckerCAD* installation directory **\Program\Bin**. If necessary, copy the passages from this file to the file in the user directory whose settings you want to change. When you start the ***File***, ***Print ...*** menu command in the 3D window, the entries of the ini file in the user directory are loaded last and are therefore decisive.

General settings

The section `[General]` contains the following general settings:

`[General]`

`BeforePlot=false`	`true` = Text is printed at the beginning `false` = Text is printed at the end When printing from 3D only `false` possible
`FontName="Arial"`	General font
`Rows = 50`	Number of rows on the paper; the result is the text height.
`Indentation = 0.02`	Factor for feed related to the paper width. 0 = Text starts at paper edge
`PlotHeadline=false`	`true` = Header line is printed `false` = Header line is not printed
`Headline=Datum tt.mm.jj`	Current selection to be printed for the header line
`PlotFootline=false`	`true` = Footer line is printed `false` = Footer line is not printed
`Footline=Modellname mit Pfad`	Current selection to be printed for the footer line

Creating Custom Information

Additional attributes for the expression can be created by Python expressions from _Runtime.py, whereby a transfer parameter 'opModel' is provided, which represents the Op model. This allows you to access any further information in the model via the Python API, e.g. the active drawing, including a title block, including an attribute value.
The transfer parameter 'opModel' does not necessarily have to be used, e.g. for date output you do not need it.

Even fixed texts must satisfy the rules of a Python expression, i.e. surrounded with "". If umlauts occur, a "u" must be placed in front of them.
Line up several Python texts by + to combine several 'Python formulas' and / or fixed texts in one line. An additional empty line can be reached by an empty text string "".

Examples:

```
[Date and time]
DateTime('%d.%m.%Y %H:%M')

[Date dd.mm.yy]
DateTime('%d.%m.%y')

[Date dd.mm.yyyy]
DateTime('%d.%m.%Y')

[Time hh:mm]
DateTime('%H:%M')

[Time hh:mm:ss]
DateTime('%H:%M:%S')

[Model name]
ModelName(opModel,0)

[Model name with path]
ModelName(opModel,1)

[Signature]
u"mit freundlichen Grüßen"
unicode(os.getenv('USERNAME'),'cp1252')

[CopyRight]
```

```
"CopyRight by DataSolid GmbH 2020"

[Username]
unicode(os.getenv('USERNAME'),'cp1252')

[Computername]
unicode(os.getenv('COMPUTERNAME'),'cp1252')
```

Create Pixel File from 3D View

The command ***File, Create Pixel File..*** creates a pixel file from the 3D view. The main topic of this command is to create high resolution pixel images in the file formats BMP, JPG and TIF.

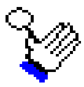
If visible edges and silhouettes should also be in the output the following preconditions must be fulfilled.

The dialogue window to save the pixel file is offered after clicking the ***Create*** or ***Create & Close*** button.

In addition to the extended settings for a perfect print described later on, also an interactive preview is implemented to show the 3D view in relation to the page size. Below please find the commands for the preview:

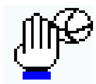
Move View

Click in the preview window with the middle mouse button pressed. Move the mouse to move the view.



Zoom In / Zoom Out

Zooming can be done with the scroll button or by pressing the right mouse button and moving the mouse up and down.

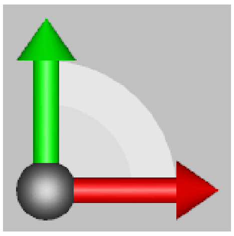
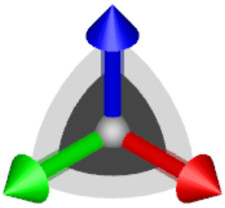
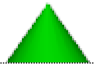
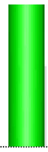
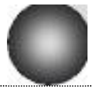
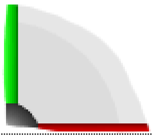
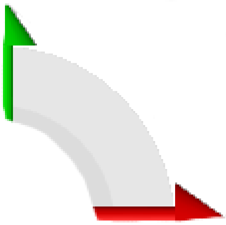


Rotate View

Click in the preview window with the left mouse button pressed. Move the mouse to rotate the view.

Display the interactive coordinate system

If this option is active, the view will rotate by a click on the following position....

 		... in view direction of the arrow
		... by 90° (arrow shaft)
		.. displays all objects
		... in the perspective view (inner circle)
		... in a view from the front or from behind (outer circle)

Settings for Create a Pixel File

After clicking the menu commands ***File***, ***Create Pixel File...*** with a 3D window active a dialogue window appears to create a pixel file from the 3D window.

DPI (dots per inch)
Input field to define the resolution of the points for the output. The higher the value, the better the quality of the output.

Portrait / Landscape
Select the desired option to print in portrait or landscape format.

Pixel Size
Displays the resulting size of the output in pixels according to the resolution DPI and the current paper size.

Size
Input and Output field for the paper size in mm. the size can be entered here or be selected from the underlying list of standard paper sizes. When choosing other units than mm, the unit must be entered after every value and confirmed by pressing the ***Enter*** key: 1cm x 0,02m.

Border
Entry field to define the border width to the selected paper size in mm. When choosing other units than mm, the unit must be entered after every value and confirmed by pressing the ***Enter*** key: 1cm x 0,02m.

Paper Size
Selection list with standard page formats.

 By clicking the displayed icon the actual view from the 3D window is taken over into the preview window.
Using an orthogonal view in the preview the view angle and the zoom is adapted, in a perspective view due to the different view modes only the view angle can be adapted.

Perspective View
When activated, the current view is printed in a perspective view instead of an orthogonal view.

Display Coordinate System
The interactive coordinate system allows to modify the view in the preview window. It will also appear in the pixel file, when activated.

Antialiasing
This option is dependent on the computer hardware, especially from the graphic board and the installed drivers. With the antialiasing option the lines can be smoother. The higher the value the more the lines are smoothed.

3D Graphic
Allows the print of several 3D file formats, without loading the file before. After clicking this button, you can select a 3D object you wish to print.

3D Model
The view of a 3D graphic is closed and the current 3D model is displayed again.

Activate Color Gradient
Allows to place a gradient in the background for printing. The color will change from the first selected color to the second selected color. When this option is deactivated the background is printed in the given color.

Print 3D view

If also the visible 2D lines in the 3D window should be printed you must start the print command in a user defined symbol bar.
The command ***print 3D view …*** can be found in the structure tree under ***General CAD Commands, Input/output***.

After you have started this command, you will see a dialog box for specifying the printer settings, to start printing and to save the printer settings in the model.

current printer
This box shows the description of the Windows standard printer.

Should you want to specify a different printer, click the ***Setup*** button and then select it.

maximum print area
This shows the dimensions (in mm) of the largest possible area that your printer can use with reference to the size of the paper that you have indicated in the settings.
This entry will be taken over from the Windows printer controls and cannot be edited.

no. of copies
This entry will determine how many copies are printed out.

output to file
If this setting is active, you can write the print-out to a file.

scale line width
If this setting is not active, lines will be drawn in the widths specified for layers or for objects.
If the setting is active, however, the line widths will be in the same scale as the drawing that has been adapted to the ***defined size*** for the print area.

print black/white
If this setting is active, all the visible objects will be printed in black and white.
If the setting is not active, however, the visible objects will be printed in the colours specified or in shades of grey, depending on the printer settings.

Selection

Current viewpoint
In the ***Current viewpoint*** option, originating from the current view, the maximum print area of the selected paper format is printed on.

Current window
In the ***Current window*** option only the window currently displayed on the screen will be printed.

Entire drawing
In the ***Entire drawing*** option the view is printed that results after selecting the  ***Show All Objects*** command.

defined size
The entry you make here will determine the size of the area to fit in the

output of your 3D model. The values must not be greater than those entered in the box indicating ***maximum print area***.

displacement
The values entered here will result in a move of the origin of the drawing on the paper.
The total of the values for the ***defined size*** and the ***displacement*** horizontally and vertically must not be greater than the maximum print area size.

hardware offset
Ad a rule a printer cannot print on the entire sheet.
Type a distance from the lower left corner of the sheet as the starting point or the origin for the printer.
The entry to be made is listed in your printer manual.
The standard values will be taken over from the Windows printer control program.

Print quality
After you have opened the dialog box, ***pixels per inch*** will show you the current maximum resolution of your printer in a horizontal and vertical direction.
In order to modify the resolution, use the settings for your printer driver.

For sketch printouts, you can determine the quality of the printout in the options **Current viewpoint** and **Entire drawing** and therefore enable a faster print. The scale goes from **1** for **very fine** up to **10** for **very rough** printout.

Print
Start printing by clicking the ***Print*** or ***Print & Close*** command button.

5.14 Export of 3D Objects for Publishing in Web Browsers

from ***BeckerCAD 3D***

With the web export in *BeckerCAD 3D* you can quickly and easily create HTML files from the 3D window for your own website or even just for viewing in a web browser. In just a few steps the 3D model is converted into a WebGL graphic. The advantage is obvious, because you can make your 3D data available for everyone to view without having to install any third-party software or additional program on a computer.

In addition, a viewer is integrated in the converter, which you can configure as you like for your application. This viewer enables, for

example, for event planning or plant planning, a walk through mode in rooms or plants. Different locations with the designation of the respective position enable fast jumps in large models.
Alternatively, you can also start the Viewer in rotate-view mode, so that the camera rotates around the 3D model and you can view it from all sides. In the Viewer, you can locate objects in the structure and show or hide them using the exported Model Explorer.

Prepared scenarios for the 3D viewer are available for event planning / architecture and mechanical engineering. With the appropriate knowledge of HTML, you can adapt the appearance of the Viewer to your own requirements.

Please note: Some WEB-Browsers prevent by the Same-Origin-Policy opening local HTML files.

However, if you move the viewer files to a Web server, a different protocol is used and is no longer blocked.
In most browsers, however, access to local files can be allowed. To do this, you must change the settings for the browser's security concept.

For this reason, after exporting, the separate Viewer *Electron* will be opened instead of the default browser so you can view the 3D scene.

General procedure

1. Switch the view to the 3D window.
2. If some 3D objects should not be exported into the HTML file, hide them before.
3. Select the menu function ***File***, ***Export***, ***Web-Export***.
4. In the ***target directory*** entry field, enter the drive, folder and the name for web export. You can do this by making an entry in the input field or by clicking the button ... and then select a folder.
5. **Application**
   Prepared scenarios for the 3D viewer are available for event planning / architecture and mechanical engineering. With the appropriate knowledge of HTML, you can adapt the appearance of the Viewer to your own requirements. After the standard installation, the scenarios contain the following settings.

Specifications of the applications

No selection

- If you do not select an application, only the file index.html for the 'pure' 3D viewer will be created.
- The 3D viewer starts in perspective view.

Eventplanning

- The 3D Viewer is created in the files example.html, index.html and scene.html
- The 3D viewer starts in the walk through mode.
- When switching to the *perspective view* in the viewer, the group 'Dach' including the subgroups is automatically hidden to allow the view into the building.

Mechanical Engineering

- The 3D Viewer is created in the files index.html and scene.html.
- The 3D viewer starts in perspective view.
- If you have activated the ***Export Model Explorer*** option in the ***settings***, 3D objects are highlighted when you double-click on an entry in the explorer, and vice versa when you double-click on a 3D object.
- If you have activated the ***Export Model Explorer*** option in the ***settings***, you can open the context menu in the Viewer on an entry in the Explorer and have the additional options:
  Jump to: The object in the graphic is highlighted and the view is centered on the object.

Please note:

Select an application before making further settings in the lower area. For example, the Event Planning application creates the camera position with a predefined distance to the ground for additional locations.

6. **Camera**
   If you have activated ***Place camera automatically***, the pass mode starts in the middle of the 3D scene. In *Perspective View*, the camera is outside the 3D objects and rotates around the center of the scene.

   You can set any position of the camera and a viewing direction if you deactivate ***Automatically place camera***. Enter the values for

the position of the camera and the viewing direction in the input fields or click on ***Select*** and then define the starting position of the camera in the 3D model first and the viewing direction with the second mouse click.
In ***Event planning***, the Z coordinate of 160cm is added, so that the camera is at eye level in the walk through mode.

7. **Locations**
   For fast jumps in the 3D viewer, you can set multiple camera positions in the 3D model. Click ***Add*** and define the location of the camera with the first point and the viewing direction with the second point.

   The name of the position is displayed in the viewer and you can switch between the locations by clicking on a name.
   Double-click on a column to change the name of the camera, its position and viewing direction.
   To delete a line in this area, highlight the line and press ***Delete*** on the keyboard.

8. In the ***Settings*** area, you can activate the following options:

   Consider component identity
   Activate this option to reduce the file size of the viewer and increase performance, especially in large models.

   Unite solids per group
   All solids within a group (without subgroups) are combined to one graphic file to increase the speed when opening and viewing 3D objects in the viewer. In this case, however, the objects within a group can no longer be localized individually in the viewer.

   Activate shadow
   Activates the shadow calculation for solids. Please note that depending on the model size, the calculation has a considerable influence on the performance in the viewer. For this reason, if you activate the shadow here, it can be switched on and off in the Viewer later using the keyboard shortcut ***K***.

   Export Model Explorer
   The group structure of the model is displayed in the viewer in an *Explorer*, which can be opened by clicking the symbol on the left side. The Explorer can be used to show and hide the contents of a group, to localize them in the graphic or in the Explorer by double-clicking.

Export invisible groups

Only if the ***Export Model Explorer*** option is active can you export hidden groups for the viewer in *BeckerCAD*. Using the Explorer the hidden groups can be made visible in the Viewer.

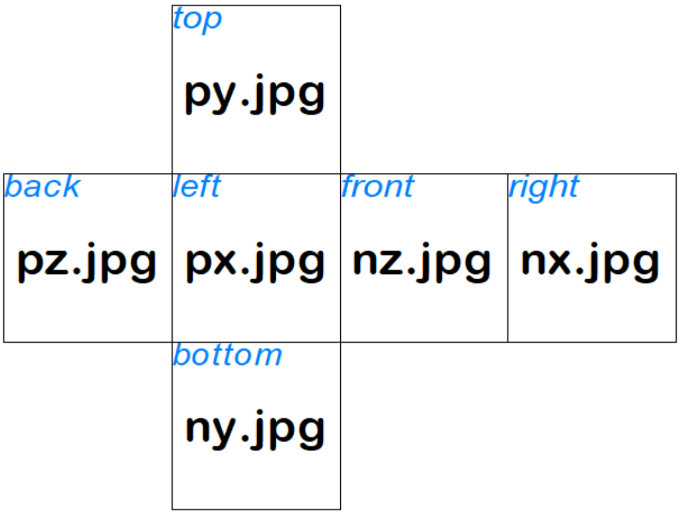
Export locations as a list

The additionally defined camera positions are displayed in a list at the right edge of the viewer and can be accessed via the name of the position in the list.

9. With a ***Skybox*** you create a virtual background for the 3D model. It lies practically in the middle of a huge cube, which is created from single image files. Click on Add to select the directory for the skybox.

   In order to have the wrapping cube displayed completely, you need six image files in jpg format, which should be located in the selected directory. The individual files must have the following file names.

   Example for the opened cube of a Skybox:



10. **With separate Viewer**

    Activate this option <u>only</u> if you want to view the result of the export with the separate Viewer *Electron* on other computers. In this case, all necessary files for the Viewer will be copied into the target directory, so that you can pack and distribute it if necessary.

    Otherwise only a link to the Viewer *Electron* is created in the target directory, via which you can view the WEB export on your own computer.

11. Click on ***Export*** to create the necessary files for viewing the 3D model in the separate Viewer *Electron* or WEB browser.

    The file *index.html* is then opened in the Viewer *Electron* due to the Same-Origin-Policy described above.

The HTML-files for the integration of the WEB Viewer into your homepage including all other necessary files are located in the specified target directory \app\dist resp. \resources\app\dist, if you have exported including the separate Viewer *Electron*.

5.15 Buildings

from ***BeckerCAD 3D***

BeckerCAD contains special commands for creating walls and openings in the 3D area based on a 2D drawing. The building commands are activated via a user-defined toolbar or menu. Call the menu command ***View***, ***Toolbar ...*** to activate a toolbar. If the default toolbar file is loaded, activate the ***Building*** toolbar. Otherwise, you will find the functions described below in the ***Building*** folder on the right side of the command tree.

Create / Edit Walls

from ***BeckerCAD 3D***

The command for creating and editing walls in the 3D area from 2D faces can be activated via a user-defined toolbar. It is located in the folder ***Building***.

From a 2D face to be identified in a 2D drawing, the command generates a swept solid in the 3D area along Z when the absolute coordinates are transferred. The wall is created on the basis of the standard WP; this serves as the floor of the building, so to speak.

1. The prerequisite for executing the Create Wall command is an existing 2D face in a drawing. Before executing this command, create a geometry (e.g. with the command ***Draw 2D***, ***Multi-Line***) and a 2D face within this closed contour.
2. The wall in the 3D area is provided with the display properties during creation, which are preset via the menu command ***Settings***, ***3D Display*** on the ***Solid Display*** tab.

Click on the icon shown in a user-defined toolbar to start the ***Create Wall*** command.

3. In a 2D drawing, identify an object of type 2D face, which, for example, represents the filling face of a wall.
4. In the ***Create wall*** dialog box that opens, specify the ***Height*** for the swept solid.
5. In the ***FF*** input field, enter the value for a distance between the wall and the standard WP if the wall in the 3D area starts with a certain height coordinate other than zero.
6. After you have clicked on Apply, the wall is created in the 3D area.

The 2D face receives an attribute with the assigned information for the solid in the 3D area. By re-identifying the face using the *Create Wall* command, the previously defined values are displayed and can be edited as required. Delete the value in the ***Height*** input box to remove the wall in the 3D area.
Confirm each change in the *Create Wall* dialog with the ***Apply*** button.

If changes in the 2D drawing have not yet been updated in the 3D area, you can recalculate all walls and openings by clicking on the ***Update building*** icon.
Please note that afterwards all walls available in the 3D area will be provided with the display properties that are preset via the menu command ***Settings***, ***3D Display*** on the ***Solid Display*** tab. Any assigned display properties such as colors and textures are lost at this moment.

Create / Edit Openings

from ***BeckerCAD 3D***

The command for creating and editing openings in a 3D wall from 2D symbols can be activated via a user-defined toolbar. It is located in the folder ***Building***.

The 2D face within a symbol to be identified in a 2D drawing uses this command to create a subtraction solid in the 3D area - an opening. As a symbol for an opening, you can save any geometry that must contain a maximum of one continuous 2D face. The opening is displayed as a swept solid along Z and subtracted from a previously created wall solid when the absolute coordinates are transferred. The values for the opening are the height coordinates in relation to the standard WP in the 3D area.

If a 3D graphic file matching the symbol is stored in the **/User/SAT** directory, it is automatically placed at parapet height when the opening is created. The link between the 2D symbol and the 3D object is established by the **name** (symbol name = file name of the 3D graphics file). In addition, a directory structure within a symbol library of the same name must be adopted as the directory structure for the 3D objects below the SAT directory.

1. Before you can create any opening in a wall, you need a symbol with a unique 2D face.

When *BeckerCAD* is installed, the library file *Openings.syl* is copied to the default symbol directory. This file contains symbols for openings in standard wall thicknesses in different lengths.

If you want to create your own symbols for openings, use the functions in the ***Draw 2D >*** toolbox to draw a unique and closed contour. Then create a 2D face within this contour and save everything together as a symbol.
Tip: If you select the ***Save and replace*** mode when saving the symbol, you do not need to reload and place the symbol afterwards.

2. If there is no symbol for the opening in the drawing yet, you can load it via the menu command ***Insert***, ***Symbol*** and place it in the desired position.

3.  Use the symbol shown to start the ***Create opening*** command**.**

4. Identify a line of the symbol.

5. Enter the following values for the opening:
Parapet = Distance between top edge of floor (standard WP) and bottom edge of opening
Height = Height of the opening

6. Click ***Apply*** to create the opening in the 3D area.

The symbol stores the information for the opening. By identifying the symbol again with the ***Create opening*** command, the previously defined values are displayed and can be edited as required. Delete the value in this dialog or the symbol in the drawing to remove the opening in the 3D area.
Confirm each change in the *Create Opening* dialog with the ***Apply*** button.

If changes in the 2D drawing have not yet been updated in the 3D area, you can recalculate all walls and openings by clicking on the ***Update building*** icon.
Please note that afterwards all walls available in the 3D area will be provided with the display properties that are preset via the menu command ***Settings***, ***3D Display*** on the ***Solid Display*** tab. Any assigned display properties such as colors and textures are lost at this moment.

Insert / Edit Opening Element

from ***BeckerCAD 3D***

Use the ***Insert opening element*** command to place symbols in floor plan drawings, e.g. for doors and windows. The symbols, if contained in the library, adapt to the position and width of the wall when you move over a wall geometry. The selection of the opening elements and the associated objects are managed in the Openings.ini file. This file is located in the working directory */User* and can be edited with a text editor if required. Alternatively, you can automatically insert the corresponding 3D object at the corresponding position in the 3D area. You start the command via a user-defined toolbar. It is located in the folder ***Building***.

1. Start the command ***Insert opening element***.
2. Select an opening element from the list.
3. The values displayed in the dialog are loaded from the Openings.ini file. These are relevant if you want to insert the opening parallel in the 3D model. The following applies to the entries delivered during installation:
   Parapet / FF = Distance between top edge of floor (standard WP) and bottom edge of opening
   Height = Height of the opening

   Please note: *The 3D object for the opening does not adjust automatically if you change the height in the dialog.*
4. Activate the ***Insert opening in 3D model*** option if the opening and the opening element are also to be placed in the 3D area.
5. Confirm the settings with ***OK***.
6. The selected symbol is placed on the cursor. Move the mouse pointer over the wall line on which you want to insert the opening

element. The symbol automatically adopts the angle of the drawn section as well as the wall thickness for the opening. The Snap point snap option is active so that you can click on the wall line at the desired position. Select alternative snap options using the middle mouse button or the keyboard commands.

If the dialog hides the view on the drawing, you can hide it with ***Close***. By pressing the space bar it is opened again.

Please note:

The symbol only takes over the alignment of the wall lines for objects in the form of *lines*. The depth of the symbol is only adjusted if there is a 2D face within the wall lines and the parallel wall lines have a distance that is stored as wall thickness in the Openings.ini file.

7. ***Cancel*** or right mouse button terminates the command.

Identify the symbol with the ***Create opening*** command if you want to change the parapet height or the height of the opening (ATTENTION: The height of the 3D object does not adapt automatically when changing the height!). If you want to change the type or width of the symbol, delete the placed symbol and insert a new symbol as described above. If necessary, then click on the ***Update building*** function to recalculate the 3D model.

If you change the values for *parapet* or *height* when inserting the opening element, the changed values are written to the Buildings.ini file in the *working directory /User* and are therefore available again when you select them again. Delete this file to restore the default.

If changes in the 2D drawing have not yet been updated in the 3D area, you can recalculate all walls and openings by clicking on the ***Update building*** icon.
Please note that afterwards all walls available in the 3D area will be provided with the display properties that are preset via the menu command ***Settings***, ***3D Display*** on the ***Solid Display*** tab. Any assigned display properties such as colors and textures are lost at this moment.

Structure and content of the file Opening.ini

The file Opening.ini is located in the working directory/User. This can be opened and edited with a text editor. If you want to make changes to this file, follow the instructions below. Create a backup copy of this file after the change, as this file will be overwritten during an update installation.

[Openings]

This section contains the entries for the list selection in the dialog Insert opening element.
Insert additional lines in the following syntax:
<unique ID>=<Descriptive text for list selection>

In the lower area you enter the following information according to the pattern of the existing entries for each opening element. Repeat the last line for each wall thickness for which you have created a symbol.

[unique ID]

floor=< Default value for parapet height >|< Text before input field >
height=<Default value for height of opening>|< Text before input field >
Library=<file name of symbol library>
<Wall thickness in mm>=<Relative path for symbol in library and 3D object in SAT folder>

6 Problem Resolution

Choose from the following chapters:

6.1 Program crashes and crash reports

If one of our ***CADdy***$^{++}$ programs crashes frequently, you can activate the recording of a crash report on your computer and make it available to the technical support of ***DataSolid GmbH*** to localize the problem.
In the crash report, you can set the following:
• Automatically send the error report
• Automatic restart of the program
• Visual recording of the last work steps before the crash
• Display the files to be sent
• Directory for the crash report
• Scope of data on the hardware used and the user interface in ***CADdy***$^{++}$
When a crash occurs, all available information is collected and provided in a packed file for transfer.
In addition, you should include information about your actions before the crash and the model to the mail.

There are different categories for recording the crash reports. Depending on the enabled category, less or more data is collected. In the event that the error report has become too large to send by e-mail, you can provide the technical support files via an FTP server. In this case, you must announce the provision of the files by e-mail or by telephone.

The recording of the crash report is disabled by default. Contact the Support of ***DataSolid GmbH*** to activate the crash report and make the desired settings.

6.2 Error message "The file is locked by another user"

Problem description

When you try to open a ***CADdy⁺⁺*** Model or Template, you receive the following error message:

„The file is locked by another user“

Reason

This message appears if an owner file for the model / template already exists. This may occur if one or more of the following causes apply:

- *CADdy^{++}* was not terminated properly before and therefore the owner file was not deleted.

– or –

- The file is shared over the network and another user has opened the Model/Template.

Solution

To resolve this problem, first make sure that no other user on the network has opened the file. If the file is not being used, remove the owner file as follows:

6. Start Windows Explorer, and then navigate to the folder that contains the Model/Template you tried to open.
7. Delete the owner file.
   The owner file is located in the same folder as the model / template you want to open. The name of the owner file is the same as the file name followed by the file name extension **`.lock`**. For example, the owner file for `Example.mod` is called `Example.lock`.
8. Start *CADdy^{++}*.
9. Open the Model/Template.

Further information

CADdy^{++} creates an owner file when you open a previously saved Model or Template. An owner file is temporary and contains the computer name on which the file is open.

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