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File Menu

New Drawing (File Menu)

This command starts a new drawing. Any previously loaded drawing is removed (after confirmation) from memory.

The new drawing is based on the elements in the drawing `RELEASE4.T4G` (or the relevant user-dependent file in a multiple user environment). The file has no name until saved; the drawing name `(UNTITLED)` is shown in the title bar.

The drawing details (see Drawing Details) are blank except for the details of the date and author.

If the elements upon which new drawings are based are to be altered, use the command Save Elements as Default. The next new drawing will then be automatically based on these elements. To replace the current elements with those from another drawing or add the elements from another drawing to the current one, use the command Import>TVG 4.0 (*.T4G).

Related command:

No related commands

Open Drawing (File Menu)

This command opens an existing drawing (loads it into memory). The File Selection Dialog appears, where the drawing to be loaded can be chosen.

TommySoftware® CAD/DRAW files usually have the extension `.T4G`.

Before the chosen drawing is loaded, the current drawing, if any, is removed (after confirmation) from memory. After this, the chosen drawing is loaded and displayed.

If the drawing contains blocks which are not currently in memory, an attempt will be made to load them from the relevant library. If a block can not be loaded, the instance will be shown as a text label. As soon as the block becomes available the instance will be displayed correctly. The same goes for TommySoftware® CAD/DRAW's own fonts if no available substitute has been specified with the Fonts command.

If the drawing can not be loaded successfully, a new blank drawing is started (see New Drawing).

Related commands:

- Drawing List
- Import▷TVG 4.0 (*.T4G)

Save Drawing (File Menu)

This command saves the current drawing, if it already has a name. If the drawing has not yet been saved, i.e. if the title bar shows `(UNTITLED)` then this calls the command Save Drawing as instead.

Before saving, a dialog appears asking whether or not to overwrite the existing file. This helps to avoid the accidental overwriting of important data.

Drawing files do not contain Undo and Redo information, which are only available as long as the drawing remains in memory.

Related commands:

- Save Drawing as
- Export>TVG 4.0 (*.T4G)

Save Drawing as (File Menu)

This command saves the current drawing in a given file. The File Selection Dialog appears in which the file to be used can be specified.

TommySoftware® CAD/DRAW drawing files usually have the extension `.T4G`, although other extensions can be used, which can cause confusion.

If you choose to save a drawing which already has a name, a dialog will appear asking whether the new file name should become the current drawing name. In this case, every time the Save Drawing command is used the drawing is saved in the new file. The new drawing name is shown in the title bar.

Drawing files do not contain Undo and Redo information, which are only available as long as the drawing remains in memory.

Related commands:

- Save Drawing
- Export>TVG 4.0 (*.T4G)

Drawing List (File Menu)

This command calls a list of drawings. The list contains up to 14 of the last-used drawing files. If an entry is chosen from the list, the corresponding drawing is loaded.

Before loading the chosen drawing, the current drawing (after a suitable warning) is removed from memory. After that, the chosen drawing is loaded and displayed. To start with, a page view is shown.

If the drawing contains instances of blocks which are not currently in memory, then an attempt will be made to load them from the relevant libraries. If a block cannot be loaded, it is shown in text form. As soon as the block becomes available the instance will be shown correctly. The same goes for TommySoftware® CAD/DRAW's own fonts as long as no substitute font has been specified with the Fonts command.

If the drawing cannot be loaded successfully, a new drawing is started in a similar way to choosing the command New Drawing.

The drawing list is saved automatically and is available as soon as the program is restarted. If the first entry from the list is chosen, the last-open drawing is loaded.

Related command:

- Open Drawing

Import>TVG 4.0 (*.T4G) (File Menu)

Using this command, objects, blocks, instances and elements can be merged into the current drawing from another file. The other file is chosen from the File Selection Dialog. To choose which other elements to load, apply the "Options" button then the Choose T4G Elements dialog appears, where the other elements to be loaded can be chosen.

TommySoftware® CAD/DRAW drawings usually have the extension `.T4G`.

When merging blocks, conflicts can arise. Because blocks are identified by their names, the name must only be used once within a drawing. If the drawing to be merged contains blocks with names which are already in use, they cannot be loaded.

Because the appearance of an object depends on the pen settings, line pattern and layer, merged objects can look very different from their original appearance. The current pen, line pattern and layer settings may be quite different from those used in the original drawing,

Related command:

- Open Drawing
- Export>TVG 4.0 (*.T4G)

Import>Enhanced Metafile (*.EMF) (File Menu)

This command pastes objects from an Enhanced Metafile Format (EMF) file in to the current drawing. A File Selection Dialog, dialog appears, from which the relevant EMF file can be chosen.

After this, the data from this file are converted into objects and pasted into the current drawing. Information about text and bitmap graphics will, however, be ignored.

The pasted objects will be marked internally as if they had been selected. If, for example, these objects are to be moved or scaled they can be selected quickly and easily using the F11 Key.

Related command:

- Export>Enhanced Metafile (*.EMF)

Import>Windows Metafile (*.WMF) (File Menu)

This command pastes objects from a Windows Metafile Format (WMF) file in to the current drawing. A File Selection Dialog, dialog appears, from which the relevant WMF file can be chosen.

After this, the data from this file are converted into objects and pasted into the current drawing. Information about text and bitmap graphics will, however, be ignored.

The pasted objects will be marked internally as if they had been selected. If, for example, these objects are to be moved or scaled they can be selected quickly and easily using the F11 Key.

Related command:

- Export>Windows Metafile (*.WMF)

Import>Windows Bitmap (*.BMP) (File Menu)

This command creates a bitmap object which is used display a bitmap within the drawing.

1. *Enter position*
   The position of the lower left corner of the bitmap can be entered by clicking the mouse anywhere in the drawing.

   Optionally, specific coordinates can be entered. Press the F8 key or press ENTER. Absolute or polar coordinates can then be entered. For further information on coordinate entry, see Coordinate Entry (F8).

After the position has been entered, the File Selection Dialog appears, in which the desired BMP file can be chosen.

The bitmap is then loaded into memory, and if needed, adjusted to the color depth of the monitor. The type and quality of the adjustment depends largely on the screen driver in use.

Up to 50 bitmaps can be in memory concurrently. If a bitmap is display at different locations within the same drawing, it will be loaded to memory only once.

If you are using Windows 95, please read this Important Note for Windows 95 users!

Optionally, bitmap objects may be used or ignored during output (i.e. to the printer, Clipboard, a metafile or another bitmap). Use the Settings>Output command to set this option. By default, bitmaps objects are used.

Related commands:

- Import>Update Bitmap List
- Export>Windows Bitmap (*.BMP)
- Export>Windows Bitmap, Section (*.BMP)

Import>Update Bitmap List (File Menu)

This command can be used to update the program-internal bitmap list. It results in all bitmaps being removed from memory and then reloaded from disk if required.

Related commands:

- Import>Windows Bitmap (*.BMP)

Export>TVG 4.0 (*.T4G) (File Menu)

This command is used to save objects and other elements from the current drawing in a new drawing file.

TommySoftware® CAD/DRAW drawing files normally have the extension `.T4G`. However other extensions can be used, which easily leads to confusion.

1. *Choose objects*
   The objects to be copied must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing the objects, the File Selection Dialog appears in which the file to be used can be chosen. When pressing the "Options" button in the file selection dialog, the Choose T4G Elements dialog appears, where the elements to be saved can be chosen.

If the current elements are to be saved without the additional objects, blocks and geometry as standard elements, the Save Elements as Default can be used.

Related commands:

- Save Drawing
- Save Drawing As
- Import>TVG 4.0 (*.T4G)

Export>Enhanced Metafile (*.EMF) (File Menu)

Using this command, the chosen objects can be saved to a file in Enhanced Metafile (EMF) format.

1. *Choose objects*
   The objects to be copied must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing the objects, the File Selection Dialog appears in which the file to be used can be chosen. When pressing the "Options" button in the file selection dialog, the dialog window Settings Output appears in which the output parameters can be edited.

If you are using Windows 95, please read this Important Note for Windows 95 users!

The data are created as if for printing, i.e. the *output* pen properties are used.

Related command:

- Import>Enhanced Metafile (*.EMF)

Export>Windows Metafile (*.WMF) (File Menu)

Using this command, the chosen objects can be saved to a file in Windows Metafile (WMF) format.

1. *Choose objects*
   The objects to be copied must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing the objects, the File Selection Dialog appears in which the file to be used can be chosen. When pressing the "Options" button in the file selection dialog, the dialog window Settings Output appears in which the output parameters can be edited.

If you are using Windows 95, please read this Important Note for Windows 95 users!

The data are created as if for printing, i.e. the *output* pen properties are used.

Related command:

- Import>Windows Metafile (*.WMF)

Export>Windows Bitmap (*.BMP) (File Menu) Level 2

Using this command, the chosen objects can be saved to a file in Windows Bitmap (BMP) format.

1. *Choose objects*
   The objects to be copied must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing the objects, the File Selection Dialog appears in which the file to be used can be chosen. When pressing the "Options" button in the file selection dialog, the dialog window Export Bitmap appears in which the bitmap creation parameters can be edited.

If you are using Windows 95, please read this Important Note for Windows 95 users!

The data are created as if for printing, i.e. the *output* pen properties are used.

Related commands:

- Export>Windows Bitmap, Section (*.BMP)
- Import>Windows Bitmap (*.BMP)

Export>Windows Bitmap, Section (*.BMP) (File Menu) Level 2

Using this command, a rectangular section of the drawing can be saved to a file in Windows Bitmap (BMP) format.

1. *Section: Enter corner point 1*
   The first corner point of the section can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Section: Enter corner point 2*
   The second corner point of the section can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After entering the section, the File Selection Dialog appears in which the file to be used can be chosen. When pressing the "Options" button in the file selection dialog, the dialog window Export Bitmap appears in which the bitmap creation parameters can be edited.

If you are using Windows 95, please read this Important Note for Windows 95 users!

The data are created as if for printing, i.e. the *output* pen properties are used.

Related commands:

- Export>Windows Bitmap (*.BMP)
- Import>Windows Bitmap (*.BMP)

Miscellaneous>Erase Drawing (File Menu) Level 2

This command removes the current drawing from memory and erases the drawing file from the hard disk. It is particularly useful during the maintenance of drawing collections.

A confirmation dialog appears first. If confirmed, then the current drawing is removed from memory and the associated drawing file is deleted from the hard disk.

Afterwards, a new drawing is started (New Drawing).

This command is final and can *not* be undone using the Undo command. The drawing file is deleted and permanently lost.

Related command:

No related commands

Miscellaneous>Create Duplicate (File Menu) Level 2

This command creates a duplicate of the current drawing. A file with the same name as the current drawing's file name but with the extension `.DUP` is created. This command can only be used if the current drawing already has a name.

If a duplicate with the same name already exists, then a warning appears to avoid accidental overwriting of files.

Apart from the extension, a duplicate file is a " normal" drawing. They can be loaded as a duplicate using the command Miscellaneous>Load Duplicate or like any other drawing using the command Open Drawing.

A duplicate of a drawing should be created, for example, if some variant of the current drawing is to be tried out or operations are to be carried out which will change the drawing drastically and which can not be undone using the Undo command.

Related commands:

- Miscellaneous>Load Duplicate
- Miscellaneous>Archive Drawing
- Miscellaneous>Autosave
- Save Drawing

Miscellaneous>Load Duplicate (File Menu) Level 2

This command loads a duplicate of the current drawing. Duplicate files can be created with the commands Miscellaneous>Create Duplicate and Miscellaneous>Autosave.

The duplicate must have the same name as the file in which the current drawing is saved except that the extension is `.DUP` . This command can only be used if the current drawing already has a name.

Before loading the duplicate, the current drawing is removed from memory. After a warning message has been displayed, the duplicate is loaded and displayed. To begin with, a page view is always shown.

If the drawing contains instances of blocks which are not currently in memory, then an attempt will be made to load them from the relevant libraries. If a block cannot be loaded, it is shown in text form. As soon as the block becomes available the instance will be shown correctly. The same goes for TommySoftware® CAD/DRAW's own fonts as long as no substitute font has been specified with the Fonts command.

If the duplicate can not be loaded successfully, then a new drawing is started as if the command New Drawing had been carried out.

Related commands:

- Miscellaneous>Create Duplicate
- Miscellaneous>Archive Drawing
- Miscellaneous>Autosave
- Open Drawing

Miscellaneous>Archive Drawing (File Menu) Level 2

The command allows to archive a drawing. This means that a new version of the current drawing is created where all external references have been replaced by references to internal blocks. Such a drawing is then independent of possible libraries and may be passed on without the need to include any library.

The File Selection Dialog appears in which the file to be used can be specified.

TommySoftware® CAD/DRAW drawing files usually have the extension `.T4G`, although other extensions can be used, which can cause confusion.

Drawing files do not contain Undo and Redo information, which are only available as long as the drawing remains in memory.

The conversion of external blocks to internal blocks makes necessary an adaption of the blocks' names. For each library, a new folder of the form `~x~` will be created, where x is a running number. Within this folder, the blocks' names and the hierarchical structure remains basically intact. To ensure uniqueness, each block name is followed by a number.

This name conversion is secure as long as it is only executed *once*. We strongly recommend not to modify an archived drawing by inserting new external references and then archiving it again. This may cause hidden and unrecoverable name conflicts that make the drawing file unusable. When editing a drawing, be sure to always use the original non-archived drawing!

Related commands:

- Miscellaneous>Create Duplicate
- Miscellaneous>Load Duplicate
- Miscellaneous>Autosave
- Open Drawing

Miscellaneous>Autosave (File Menu) Level 2

This command customizes the autosave function. This is done with the Autosave dialog.

After a time interval has elapsed, the program works out whether the drawing has altered since the last save. If this is the case the autosave is activated. When saving under the current drawing's name, a dialog appears which can be used to cancel the autosave if it is not required on this occasion. When saving as a duplicate, the autosave is done *without* a dialog appearing.

Autosave only works when TommySoftware® CAD/DRAW is working in the foreground and no dialog is open. The elapsed time is however measured and an autosave is carried out at the latest when the dialog is closed or TommySoftware® CAD/DRAW is activated again.

If the current drawing has not been assigned to a file name when the autosave is executed, and if the "Save as duplicate" option is active, the drawing will be saved to a file titled `~AUTO~.DUP`, located in the standard drawing directory.

Related commands:

- Miscellaneous>Create Duplicate
- Miscellaneous>Load Duplicate
- Miscellaneous>Archive Drawing
- Save Drawing

Print Drawing (File Menu)

This command is used to print a drawing or part of a drawing. Any output device for which a driver is installed can be used.

Choosing the command calls the Print Drawing Dialog, in which the print settings can be specified.

If only certain objects are to be printed, they must be selected before choosing this command, using for example the command Permanent Selection>Set. The "Selection Only" check box can then be enabled.

If you are using Windows 95, please read this Important Note for Windows 95 users!

If you use an HPGL-Plotter with the HPPLOT.DRV driver supplied by Microsoft (or Hewlett-Packard and Micrografx), you must read the text Notes for Plotter Output! It contains notes on the correct use of such a plotter with TommySoftware® CAD/DRAW.

Related command:

- Print Section

Print Section (File Menu)

This command is used to print a rectangular section of a drawing. Any output device for which a driver is installed can be used.

1. *Section: Enter corner point 1*
   The first corner point of the section can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Section: Enter corner point 2*
   The second corner point of the section can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After entering the section, the Print Drawing Dialog is called, in which the print settings can be specified.

If only certain objects of the section are to be printed, they must be selected before choosing this command, using for example the command Permanent Selection>Set. The "Selection Only" check box can then be enabled.

If you are using Windows 95, please read this Important Note for Windows 95 users!

If you use an HPGL-Plotter with the HPPLOT.DRV driver supplied by Microsoft (or Hewlett-Packard and Micrografx), you must read the text Notes for Plotter Output! It contains notes on the correct use of such a plotter with TommySoftware® CAD/DRAW.

Related command:

- Print Drawing

Exit (File Menu)

This command closes TommySoftware® CAD/DRAW. If unsaved work is still in memory, you are given the option of saving the work or not.

If Save Settings on Exit has been enabled, the current settings are saved.

All windows for which the "Save Position" entry in their system menu is enabled, have their position (and possibly their size) saved. In addition, information on which windows are displayed is also saved. The same goes for the menu bar.

Related command:

No related commands

Exit

Contents

Edit Menu

Undo (Edit Menu)

This command can be used to undo the last operation carried out. It can be used with all commands which create, alter or delete objects, instances or geometry objects. For example, the following commands, among others, can be undone:

Shape>Edit Properties
Shape>Move Objects>Standard
Draw>Line>Standard
Draw>Hatching>Objects
Trimming>Trim Object>Cut Out
Library>Insert Block

Some operations cannot be undone. These are mainly operations which alter settings or definitions, as well as alterations to blocks. These are examples of operations which cannot be undone:

File>New Drawing
Configure>Set Origin
Configure>Settings>Output
Library>Generate Block
Library>Clean Up Internal Blocks

Successive operations can also be undone. The maximum number of undoable operations depends on the "Undo Steps" setting in the command Options>General.

The undo information needed to undo operations is not saved along with the drawing, i.e. it is lost when the drawing is saved.

Related commands:

- Redo
- Erase Undo Information

Redo (Edit Menu)

This command redoes an operation canceled with the Undo command. It can only be carried out immediately after an undo. If another operation has been carried out after the undo, you cannot redo the undone operation.

Several undone commands can be redone. Successive operations can also be undone. The maximum number of redoable operations depends on the "Undo Steps" setting in the command Options>General.

The undo information needed to undo and redo operations is not saved along with the drawing, i.e. it is lost when the drawing is saved.

Related commands:

- Undo
- Erase Undo Information

Erase Undo Information (Edit Menu)

This command removes the current undo information from memory. Undo information is generated every time you carry out an operation which can be undone using the Undo command.

The size of the undo information depends on the number of objects which have been altered at the same time. If a large number of objects have been altered, then a memory bottleneck can arise. In this case, you should consider erasing the undo information.

The larger the maximum number of undo levels, the more undo information is generated. If you often get memory shortages when working with large numbers of objects, then you should either increase the available object memory or reduce the number of undo levels (see Options>General).

The undo information needed to undo operations is not saved along with the drawing, i.e. it is lost when the drawing is saved.

Related commands:

- Undo
- Redo

Cut (Edit Menu)

This command copies objects to the Windows Clipboard and then deletes them from the drawing.

1. *Choose objects*
   The objects to be cut must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing the objects, they are copied to the clipboard and remain there until new data is copied to the clipboard, which can also be done from other applications.

The original objects in the drawing are deleted without a further warning. If needed, they can be restored using Undo. This does not affect the contents of the clipboard.

The objects are placed on the clipboard in two different formats. The first format is the usual TommySoftware® CAD/DRAW (TVG 4.0) drawing format, which contains information about objects and instances. In addition, there is no loss of quality when using TommySoftware® CAD/DRAW. At present, this format can only be used by TommySoftware® CAD/DRAW and is ignored by other applications.
The second format is Enhanced Metafile Format (EMF). This format can be used by most current 32bit applications. The data are created as if for printing, i.e. the *output* pen properties are used. In addition, the current settings set with Settings>Output are used so that you can influence what is copied to the clipboard.

Related commands:


Copy


Paste

Copy (Edit Menu)

This command copies selected objects to the Windows Clipboard.

1. *Choose objects*
   The object*s* to be copied must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing the objects, they are copied to the clipboard, where they remain until new data is written to the clipboard, which can also be done by other Windows applications.

The objects are placed on the clipboard in two different formats. The first format is the usual TommySoftware® CAD/DRAW (TVG 4.0) drawing format, which contains information about objects and instances. In addition, there is no loss of quality when using TommySoftware® CAD/DRAW. At present, this format can only be used by TommySoftware® CAD/DRAW and is ignored by other applications.
The second format is Enhanced Metafile Format (EMF). This format can be used by most current 32bit applications. The data are created as if for printing, i.e. the *output* pen properties are used. In addition, the current settings set with Settings>Output are used so that you can influence what is copied to the clipboard.

Related commands:

- Cut
- Paste

Paste (Edit Menu)

This command pastes the contents of the Windows Clipboard into the current drawing. This process can use two different formats.

If the Clipboard contains data in Enhanced Metafile Format (EMF), these are converted to objects and inserted into the drawing. Text and bitmap data within the metafile data will however be ignored.

If the Clipboard contains data in TommySoftware® CAD/DRAW (TVG 4.0) format, these are used instead of the WMF data. All objects, instances and blocks are inserted into the current drawing.

Conflicts can arise when inserting blocks. As blocks are identified by their names, each name must be unique within a drawing. If blocks in the clipboard have names already in use in the current drawing, then they cannot be pasted.

The pasted objects and instances are marked internally as if they had been selected. Because of this, they can quickly be selected by using the F11 Key if, for example, they are to be moved or scaled.

As the appearance of an object depends on the current pen, line pattern and layer settings, the appearance of pasted objects may be different to their original appearance. The current pen, line pattern and layer settings may be very different to those in the original drawing.

Related commands:


Cut


Copy

Comment>Draw (Edit Menu) Level 2

This command is used to insert a comment.

1. *Insert reference point*
   The comment's reference point can be inserted by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter destination point*
   The destination point, the center of the comment field can be inserted by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After entering the target point, a dialog appears into which a comment of up to 250 characters can be typed.

The appearance of all the comments can be set using the command Comment>ParametersIDM_EDIT_COMMENT_PARAM. Comments can only be viewed on screen. They cannot be printed out or exported in any way.

Related command

No related commands

Comment>Parameters (Edit Menu) Level 2

This command can be used to set the display parameters for comments. This is done with the Comment Parameters dialog.

Altering these parameters changes the display properties of both existing and new comments.

Affected command:

Comment>Draw

Drawing Details (Edit Menu)

This command can be used to view and edit the current drawing's details. This is done via the Edit Details dialog.

Related commands:


Save Drawing


Save Drawing as

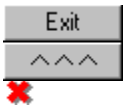
Command Queue (Edit Menu)

This command calls a list of the last 20 commands used, enabling quick and easy access to them.

The command queue works in the same way as the normal text menu. Individual entries can appear with a tick against them (if the corresponding function is active) or grayed out (if the command is not currently available).

Related command:

No related commands



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Page Formats (Configure Menu)

This command can be used to view and alter the current page format. This is done from the Page Formats dialog.

Altering the page format does not affect the drawing objects. The screen is however redrawn to give an overview of the new page.

Usually, the size of the page is shown on screen by a violet-colored dashed line. This can be turned off using the command Options>Windows.

Related command:

No related commands

Layers>List (Configure Menu)

This command can be used to display a list of all currently defined layers. In this list, a new current layer can be chosen.

If the command Shape>Modify Objects is active at the time a layer is selected from this list, the chosen layer will be assigned to the currently selected objects. In this case, the active layer is not changed.

Related command:

- Layers>Edit

Layers>Edit (Configure Menu)

This command can be used to manage the current drawing's layers. You can define, alter, delete or make a different layer the current layer.

This is done from the Edit Layers dialog.

Related command:

Layers>List

Layers>Defaults (Configure Menu)

This command can be used to set the default layer for certain types of objects in the current drawing. These default layers make it easier to allocate various types of objects to different layers automatically.

The default layers are specified from the Layer Defaults dialog.

Affected Commands:

- Marking
- Division>Distance
- Division>Object
- Division>Array
- Trim Surface>Generate Surface
- Dimension Line>Straight
- Dimension Line>Curved
- Dimension>Length, Points
- Dimension>Length, Object
- Dimension>Length, Object - Point
- Dimension>Length, Object - Object
- Dimension>Radius, Points
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- Dimension>Diameter, Points
- Dimension>Diameter, Object
- Dimension>Angle, Points
- Dimension>Angle, Line Line
- Dimension>Angle, Circle Part
- Dimension>Arc Length, Points
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- Calculate>Perimeter / Length
- Text>Standard
- Text>Frame
- Text>Reference
- Hatching>Objects
- Hatching>Generated Surface
- Block>Insert

Layers>Reset Status (Configure Menu)

This command can be used to reset all layers' properties to their default status. This means, all layer will be displayed and output, neither frozen nor shaded down, and they will be used during snap point detection.

This command cannot be reversed by means of the Layer Management>Undo command!

Related command:

Layers>Edit

Layer Management>Select (Configure Menu) Level 2

This command can be used to select a new active layer by identifying an object that is assigned to the desired layer.

1. *Identify any object*
   Any object can be identified, including objects in frozen layers.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

After identifiying an object, its layer assignment is determined and that layer is made the active layer.

Related commands:

- Layer Management>Edit
- Layer Management>Assign
- Layer Management>Disable
- Layer Management>Enable
- Layer Management>Freeze
- Layer Management>Melt
- Layer Management>Ignore
- Layer Management>Use
- Layer Management>Shade Down
- Layer Management>Light Up
- Layer Management>Hide
- Layer Management>Concentrate On
- Layer Management>Undo

Layer Management>Edit (Configure Menu) Level 2

This command can be used to edit the properties of a layer by identifying an object that is assigned to the desired layer.

1. *Identify any object*
   Any object can be identified, including objects in frozen layers.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

After identifiying an object, its layer assignment is determined and that layer can be edited. This is done from the Edit Layers dialog.

Related commands:

Layer Management>Select

Layer Management>Assign

Layer Management>Disable

Layer Management>Enable

Layer Management>Freeze

Layer Management>Melt

Layer Management>Ignore

Layer Management>Use

Layer Management>Shade Down

Layer Management>Light Up

Layer Management>Hide

Layer Management>Concentrate On

Layer Management>Assign (Configure Menu) Level 2

This command can be used to assign existing objects and instances to the currently active layer.

1. *Choose any objects*
   Choose the objects whose layer assignments are to be altered. Click on objects to choose them, including objects in frozen layers. For further information on object selection, see the chapter Object Selection.

After choosing the objects, they will be assigned to the currently active layer.

If the duplicate function is active, a copy of the chosen objects is created and the copy is altered. This means that a copy with different layer assignment can be created.

Related commands:

Layer Management>Select

Layer Management>Edit

Layer Management>Disable

Layer Management>Enable

Layer Management>Freeze

Layer Management>Melt

Layer Management>Ignore

Layer Management>Use

Layer Management>Shade Down

Layer Management>Light Up

Layer Management>Hide

Layer Management>Concentrate On

Layer Management>Disable (Configure Menu) Level 2

This command can be used to disable layers by identifying objects that are assigned to the desired layers. To disable a layer means to set the attributes "Freeze", "Ignore" and "Shade Down", i.e. the layer will be displayed in a gray color, objects assigned to it can usually not be chosen and will not be considered during snap point detection. In addition, the attribute "Output" is cleared, so this layer will neither be printed nor copied to the clipboard.

1. *Choose any objects*
   Choose the objects whose assigned layers are to be altered. Click on objects to choose them, including objects in frozen layers. For further information on object selection, see the chapter Object Selection.

After identifiying the objects, their layer assignment is determined and all affected layers are disabled.

Related commands:

Layer Management>Select

Layer Management>Edit

Layer Management>Assign

Layer Management>Enable

Layer Management>Freeze

Layer Management>Melt

Layer Management>Ignore

Layer Management>Use

Layer Management>Shade Down

Layer Management>Light Up

Layer Management>Hide

Layer Management>Concentrate On

Layer Management>Undo

Layer Management>Enable (Configure Menu) Level 2

This command can be used to anable layers by identifying objects that are assigned to the desired layers. To enable a layer means to clear the attributes "Freeze", "Ignore" and "Shade Down", i.e. the layer will be displayed in its original color(s) again, objects assigned to it can be chosen and will be considered during snap point detection. In addition, the attribute "Output" is set, so this layer will be printed and copied to the clipboard again.

1. *Choose any objects*
   Choose the objects whose assigned layers are to be altered. Click on objects to choose them, including objects in frozen layers. For further information on object selection, see the chapter Object Selection.

After identifiying the objects, their layer assignment is determined and all affected layers are enabled.

Related commands:

Layer Management>Select

Layer Management>Edit

Layer Management>Assign

Layer Management>Disable

Layer Management>Freeze

Layer Management>Melt

Layer Management>Ignore

Layer Management>Use

Layer Management>Shade Down

Layer Management>Light Up

Layer Management>Hide

Layer Management>Concentrate On

Layer Management>Undo

Layer Management>Freeze (Configure Menu) Level 2

This command can be used to freeze layers by identifying objects that are assigned to the desired layers. To freeze a layer means to set the attribute "Freeze", i.e. objects assigned to it can usually not be chosen and thus not be modified.

1. *Choose any objects*
   Choose the objects whose assigned layers are to be altered. Click on objects to choose them, including objects in frozen layers. For further information on object selection, see the chapter Object Selection.

After identifiying the objects, their layer assignment is determined and all affected layers are frozen.

Related commands:

Layer Management>Select

Layer Management>Edit

Layer Management>Assign

Layer Management>Disable

Layer Management>Enable

Layer Management>Melt

Layer Management>Ignore

Layer Management>Use

Layer Management>Shade Down

Layer Management>Light Up

Layer Management>Hide

Layer Management>Concentrate On

Layer Management>Melt (Configure Menu) Level 2

This command can be used to melt layers by identifying objects that are assigned to the desired layers. To melt a layer means to clear the attribute "Freeze", i.e. objects assigned to it can be chosen and thus be modified.

1. *Choose any objects*
   Choose the objects whose assigned layers are to be altered. Click on objects to choose them, including objects in frozen layers. For further information on object selection, see the chapter Object Selection.

After identifiying the objects, their layer assignment is determined and all affected layers are melted.

Related commands:

Layer Management>Select

Layer Management>Edit

Layer Management>Assign

Layer Management>Disable

Layer Management>Enable

Layer Management>Freeze

Layer Management>Ignore

Layer Management>Use

Layer Management>Shade Down

Layer Management>Light Up

Layer Management>Hide

Layer Management>Concentrate On

Layer Management>Ignore (Configure Menu) Level 2

This command can be used to ignore layers by identifying objects that are assigned to the desired layers. To ignore a layer means to set the attribute "Ignore", i.e. objects assigned to it will not be considered during snap point detection.

1. *Choose any objects*
   Choose the objects whose assigned layers are to be altered. Click on objects to choose them, including objects in frozen layers. For further information on object selection, see the chapter Object Selection.

After identifiying the objects, their layer assignment is determined and all affected layers are ignored.

Related commands:

Layer Management>Select

Layer Management>Edit

Layer Management>Assign

Layer Management>Disable

Layer Management>Enable

Layer Management>Freeze

Layer Management>Melt

Layer Management>Use

Layer Management>Shade Down

Layer Management>Light Up

Layer Management>Hide

Layer Management>Concentrate On

Layer Management>Use (Configure Menu) Level 2

This command can be used to use layers by identifying objects that are assigned to the desired layers. To use a layer means to clear the attribute "Ignore", i.e. objects assigned to it will be considered during snap point detection.

1. *Choose any objects*
   Choose the objects whose assigned layers are to be altered. Click on objects to choose them, including objects in frozen layers. For further information on object selection, see the chapter Object Selection.

After identifiying the objects, their layer assignment is determined and all affected layers are used.

Related commands:

- Layer Management>Select
- Layer Management>Edit
- Layer Management>Assign
- Layer Management>Disable
- Layer Management>Enable
- Layer Management>Freeze
- Layer Management>Melt
- Layer Management>Ignore
- Layer Management>Shade Down
- Layer Management>Light Up
- Layer Management>Hide
- Layer Management>Concentrate On
- Layer Management>Undo

Layer Management>Shade Down (Configure Menu) Level 2

This command can be used to shade down layers by identifying objects that are assigned to the desired layers. To shade down a layer means to set the attribute "Shade Down", i.e. the layer will be displayed in a gray color.

1. *Choose any objects*
   Choose the objects whose assigned layers are to be altered. Click on objects to choose them, including objects in frozen layers. For further information on object selection, see the chapter Object Selection.

After identifiying the objects, their layer assignment is determined and all affected layers are shaded down.

Related commands:

- Layer Management>Select
- Layer Management>Edit
- Layer Management>Assign
- Layer Management>Disable
- Layer Management>Enable
- Layer Management>Freeze
- Layer Management>Melt
- Layer Management>Ignore
- Layer Management>Use
- Layer Management>Light Up
- Layer Management>Hide
- Layer Management>Concentrate On
- Layer Management>Undo

Layer Management>Light Up (Configure Menu) Level 2

This command can be used to light up layers by identifying objects that are assigned to the desired layers. To light up a layer means to clear the attribute "Shade Down", i.e. the layer will be displayed in its original color(s) again.

1. *Choose any objects*
   Choose the objects whose assigned layers are to be altered. Click on objects to choose them, including objects in frozen layers. For further information on object selection, see the chapter Object Selection.

After identifiying the objects, their layer assignment is determined and all affected layers are lit up.

Related commands:

 Layer Management>Select

 Layer Management>Edit

 Layer Management>Assign

 Layer Management>Disable

 Layer Management>Enable

 Layer Management>Freeze

 Layer Management>Melt

 Layer Management>Ignore

 Layer Management>Use

 Layer Management>Shade Down

 Layer Management>Hide

 Layer Management>Concentrate On

 Layer Management>Undo

Layer Management>Hide (Configure Menu) Level 2

This command can be used to hide layers by identifying objects that are assigned to the desired layers. To hide a layer means to clear the attributes "Display" and "Output", i.e. the layer will neither be displayed on screen nor output to printer or clipboard.

1. *Choose any objects*
   Choose the objects whose assigned layers are to be altered. Click on objects to choose them, including objects in frozen layers. For further information on object selection, see the chapter Object Selection.

After identifiying the objects, their layer assignment is determined and all affected layers are hidden.

Related commands:

 Layer Management>Select

 Layer Management>Edit

 Layer Management>Assign

 Layer Management>Disable

 Layer Management>Enable

 Layer Management>Freeze

 Layer Management>Melt

 Layer Management>Ignore

 Layer Management>Use

 Layer Management>Shade Down

 Layer Management>Light Up

 Layer Management>Concentrate On

 Layer Management>Undo

Layer Management>Concentrate On (Configure Menu) Level 2

This command can be used to concentrate on layers by identifying objects that are assigned to the desired layers. To concentrate on layers means to disable all other layers by clearing their attributes "Freeze", "Ignore" and "Shade Down", i.e. those layers will be displayed in a gray color, objects assigned to them can usually not be chosen and will not be considered during snap point detection. In addition, the attribute "Output" is cleared, so those layer will neither be printed nor copied to the clipboard.

1. *Choose any objects*
   Choose the objects to whose layers the concentration shall be done. Click on objects to choose them, including objects in frozen layers. For further information on object selection, see the chapter Object Selection.

After identifiying the objects, their layer assignment is determined and all but the affected layers are disabled.

Related commands:

 Layer Management>Select

 Layer Management>Edit

 Layer Management>Assign

 Layer Management>Disable

 Layer Management>Enable

 Layer Management>Freeze

 Layer Management>Melt

 Layer Management>Ignore

 Layer Management>Use

 Layer Management>Shade Down

 Layer Management>Light Up

 Layer Management>Hide

 Layer Management>Undo

Layer Management>Undo (Configure Menu) Level 2

This command reverses the last command that changed the layer attributes. It will work for changes made by means of one of the following commands residing in the "Layer Management" submenu.

Only changes made to the layer attributes "Display", "Output", "Freeze", "Ignore" and "Shade Down" will be reversed. Changes to the transmission of layer properties, as well as creation, deletion and renaming of layers will not be influenced!

Affected commands:

 Layers>Edit

 Layers>Reset Status

 Layer Management>Disable

 Layer Management>Enable

 Layer Management>Freeze

 Layer Management>Melt

 Layer Management>Ignore

 Layer Management>Use

 Layer Management>Shade Down

 Layer Management>Light Up

 Layer Management>Hide

 Layer Management>Concentrate On

Coordinate Systems>List (Configure Menu)

This command can be used to display a list of all currently defined coordinate systems. In this list, a new current coordinate system can be chosen.

Related commands:


Coordinate Systems>Edit


Coordinate Systems>Set Origin


Coordinate Systems>Display Grid Active


Coordinate Systems>Edit Display Grid


Coordinate Systems>Position Grid Active


Coordinate Systems>Edit Position Grid

Coordinate Systems>Edit (Configure Menu)

This command can be used to manage the coordinate systems in the current drawing. They can be defined, altered, deleted or made active.

This is done from the Define Coordinate Systems dialog.

Related commands:


Coordinate Systems>List


Coordinate Systems>Set Origin


Coordinate Systems>Display Grid Active


Coordinate Systems>Edit Display Grid


Coordinate Systems>Position Grid Active


Coordinate Systems>Edit Position Grid

Coordinate Systems>Set Origin (Configure Menu)

This command positions the origin of the active window's coordinate system.

1. *Enter position*
   The origin's position can be entered by clicking the mouse pointer anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After entering the position, the coordinate system's origin moves to this point. The origin position mode of this coordinate system is set to "Specified Position".

Setting the origin position cannot be undone using the Undo command.

Related commands:


Coordinate Systems>List


Coordinate Systems>Edit


Coordinate Systems>Display Grid Active


Coordinate Systems>Edit Display Grid


Coordinate Systems>Position Grid Active


Coordinate Systems>Edit Position Grid

Coordinate Systems>Display Grid Active (Configure Menu)

This command can be used to show and hide the display grid. The display grid of the window in which the crosshair is located is changed. If the mouse is outside the drawing window the current window's setting is changed.

 Optionally, the display grid can be turned on or off by clicking with the left mouse button on the button in the panel shown to the left. In this case always the active window's setting is changed. The button shows the current display grid status.

Related commands:

- Coordinate Systems>List
- Coordinate Systems>Edit
- Coordinate Systems>Set Origin
- Coordinate Systems>Edit Display Grid
- Coordinate Systems>Position Grid Active
- Coordinate Systems>Edit Position Grid

Coordinate Systems>Edit Display Grid (Configure Menu)

This command can be used to edit the display grid of the current coordinate system by means of the dialog window Coordinate System Display Grid.

 Optionally, the display grid can be edited by clicking with the right mouse button on the button in the panel shown to the left. In this case always the active window's setting is changed. The button shows the current display grid status.

Related commands:

- Coordinate Systems>List
- Coordinate Systems>Edit
- Coordinate Systems>Set Origin
- Coordinate Systems>Display Grid Active
- Coordinate Systems>Position Grid Active
- Coordinate Systems>Edit Position Grid

Coordinate Systems>Position Grid Active (Configure Menu)

This command can be used to turn the position grid on and off. The position grid of the window in which the crosshair is located is changed. If the mouse is outside the drawing window the current window's setting is changed.

 Optionally, the position grid can be turned on or off by clicking with the left mouse button on the button in the panel shown to the left. In this case always the active window's setting is changed. The button shows the current display grid status.

Related commands:

 Coordinate Systems>List

 Coordinate Systems>Edit

 Coordinate Systems>Set Origin

 Coordinate Systems>Display Grid Active

 Coordinate Systems>Edit Display Grid

 Coordinate Systems>Edit Position Grid

Coordinate Systems>Edit Position Grid (Configure Menu)

This command can be used to edit the position grid of the current coordinate system by means of the dialog window Coordinate System Position Grid.

 Optionally, the position grid can be edited by clicking with the right mouse button on the button in the panel shown to the left. In this case always the active window's setting is changed. The button shows the current display grid status.

Related commands:

- Coordinate Systems>List
- Coordinate Systems>Edit
- Coordinate Systems>Set Origin
- Coordinate Systems>Display Grid Active
- Coordinate Systems>Edit Display Grid
- Coordinate Systems>Position Grid Active

Pens>List (Configure Menu)

This command can be used to display a list of all currently defined pens. In this list, a new current pen can be chosen.

If the command Shape>Modify Objects is active at the time a pen is selected from this list, the chosen pen will be assigned to the currently selected objects. In this case, the active pen is not changed.

Related commands:

Pen>Edit

Pen>Line Pattern

Pens>Edit (Configure Menu)

This command can be used to manage the pens in the current drawing. They can be defined, altered, deleted or made active.

This is done from the Define Pens dialog.

Related commands:

Pen>List

Pen>Line Pattern

Pens>Defaults (Configure Menu)

This command can be used to alter the default pen used for drawing certain types of objects in the current drawing. This makes it easier to automatically draw certain types of object with the same pen.

This is done from the Pen Defaults dialog in which the default pens for certain object types can be specified.

Affected Commands:

 Dimension Line>Straight

 Dimension Line>Curved

 Dimension>Length, Points

 Dimension>Length, Object

 Dimension>Length, Object - Point

 Dimension>Length, Object - Object

 Dimension>Radius, Points

 Dimension>Radius, Object

 Dimension>Diameter, Points

 Dimension>Diameter, Object

 Dimension>Angle, Points

 Dimension>Angle, Line Line

 Dimension>Angle, Circle Part

 Dimension>Arc Length, Points

 Dimension>Arc Length, Circle Parts

 Dimension>Coordinates

 Calculate>Area

 Calculate>Perimeter / Length

 Text>Standard

 Text>Frame

 Text>Reference

Pens>Line Pattern (Configure Menu)

This command can be used to manage the line patterns in the current drawing. They can be defined, altered and deleted

This is done from the Define Line Pattern dialog.

Related commands:

Pen>List

Pen>Edit

Zoom>Section (Configure Menu)

This command changes the area displayed in the active drawing window so that a given rectangular area is displayed at the maximum possible size.

1. *Section: Enter corner point 1*
   The first corner point of the section can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Section: Enter corner point 2*
   The second corner point of the section can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After entering the two corner points, the display is recalculated and redrawn.

These alterations to the visible section and the zoom factor can be canceled with the command Zoom>Previous View

The current zoom factor used for each drawing window is normally shown in the lower left corner of the drawing window, under the page symbol. It is expressed as a percentage of the original page size.

Related commands:

 Zoom>Overview

 Zoom>Page

 Zoom>Original Size

 Zoom>Factor

 Zoom>Set Center

 Zoom>Double Zoom

 Zoom>Halve Zoom

Zoom>Overview (Configure Menu)

This command alters the area displayed in the current drawing window so that you can see all of the drawing's objects and instances.

If there is currently no object on the screen, then a page overview is displayed.

These alterations to the visible section and the zoom factor can be canceled with the command Zoom>Previous View

The current zoom factor used for each drawing window is normally shown in the lower left corner of the drawing window, under the page symbol. It is expressed as a percentage of the original page size.

Related commands:

 Zoom>Section

 Zoom>Page

 Zoom>Original Size

 Zoom>Factor

 Zoom>Set Center

 Zoom>Double Zoom

 Zoom>Halve Zoom

Zoom>Page (Configure Menu)

You can use this command so that you can see the whole page as large as possible in the current drawing window.

These alterations to the visible section and the zoom factor can be canceled with the command Zoom>Previous View.

The current zoom factor used for each drawing window is normally shown in the lower left corner of the drawing window, under the page symbol. It is expressed as a percentage of the original page size.

Related commands:


Zoom>Section


Zoom>Overview


Zoom>Original Size


Zoom>Factor


Zoom>Set Center


Zoom>Double Zoom


Zoom>Halve Zoom

Zoom>Original Size (Configure Menu)

This command changes the zoom factor used for the current drawing window so that all objects and instances appear at their original size.

So that the original size can be worked out exactly, the visible screen size must be configured first using the command Options>General.

These alterations to the visible section and the zoom factor can be canceled with the command Zoom>Previous View

The current zoom factor used for each drawing window is normally shown in the lower left corner of the drawing window, under the page symbol. It is expressed as a percentage of the original page size (100% in this case).

Related commands:


Zoom>Section


Zoom>Overview


Zoom>Page


Zoom>Factor


Zoom>Set Center


Zoom>Double Zoom


Zoom>Halve Zoom

Zoom>Factor (Configure Menu)

This command can be used to specify a zoom factor for the current drawing window. It calls a small dialog box, in which the factor to be used (between 0.00000000001 and 10000000000) is entered.

These alterations to the visible section and the zoom factor can be canceled with the command Zoom>Previous View.

The current zoom factor used for each drawing window is normally shown in the lower left corner of the drawing window, under the page symbol. It is expressed as a percentage of the original page size.

Related commands:

 Zoom>Section

 Zoom>Overview

 Zoom>Page

 Zoom>Original Size

 Zoom>Set Center

 Zoom>Double Zoom

 Zoom>Halve Zoom

Zoom>Previous View (Configure Menu)

This command restores the previous view of the current drawing window. The last 20 changes to the view (changes to the zoom factor, scrolling, etc.) are saved separately for each visible window.

If this command is carried out using NUM / key on the numeric keypad, then the command applies to the window over which the mouse pointer or crosshair is located, which may not be the *active* drawing window.

Related command:

No related commands

Zoom>Set Center (Configure Menu)

This command moves the position over which the mouse pointer or crosshair is located to the center of the drawing window.

If the visible section, and therefore the zoom factor, is changed by using this command, the change can be canceled using the command Zoom>Previous View.

This command is designed to be carried out using the NUM × key on the numeric keypad as the crosshair position only makes sense then. It is included in the menu so that it can be assigned to a different key.

Related commands:


Zoom>Section


Zoom>Overview


Zoom>Page


Zoom>Original Size


Zoom>Factor


Zoom>Double Zoom


Zoom>Halve Zoom

Zoom>Double Zoom (Configure Menu)

This command doubles the size of the view in the active window, i.e. it doubles the zoom factor.

If the visible section, and therefore the zoom factor, is changed by using this command, the change can be canceled using the command Zoom>Previous View.

If this command is carried out using the NUM + key on the numeric keypad, it will apply to the window over which the mouse pointer or crosshair is located, which may not be the active drawing window. In this case, the area around the current crosshair position is altered.

Related commands:


Zoom>Section


Zoom>Overview


Zoom>Page


Zoom>Original Size


Zoom>Factor


Zoom>Set Center


Zoom>Halve Zoom

Zoom>Halve Zoom (Configure Menu)

This command reduces the view in the active window by a factor of two, i.e. the zoom factor is halved.

If the visible section, and therefore the zoom factor, is changed by using this command, the change can be canceled using the command Zoom>Previous View.

If this command is carried out using the NUM - key on the numeric keypad, it will apply to the window over which the mouse pointer or crosshair is located, which may not be the active drawing window. In this case, the section around the current crosshair position is altered.

Related commands:


Zoom>Section


Zoom>Overview


Zoom>Page


Zoom>Original Size


Zoom>Factor


Zoom>Set Center


Zoom>Double Zoom

Save Elements as Default (Configure Menu)

This command saves the current drawing elements as the defaults. Those are for example: page format, layers, coordinate systems, pens and line patterns.

The elements are saved in the file `RELEASE4.T4G` (or in the appropriate user file, when in multi-user mode) They are used as the basic settings whenever a new drawing is started, or they can be merged into an existing drawing using the command Import>TVG 4.0 (*.T4G).

If the elements are to be saved in a named file, to differentiate element groups, the command Export>TVG 4.0 (*.T4G) can be used. See TVG 4.0 File Elements and File Format for further details.

Related commands:

Save Drawing

Save Drawing as

Export>TVG 4.0 (*.T4G)

Windows>Rearrange (Configure Menu)

This command can be used to rearrange all currently visible floating windows (toolbox, panel, block list, etc.).

Related command:

No related commands

Windows>Enlarge Window 2 (Configure Menu)

This command can be used to enlarge the second drawing window. The effect is to swap the first (initially, the largest) and second drawing windows.

As the height/width relationship of all drawing windows is the same, the visible drawing area remains the same, but the zoom factor of both windows may change.

The information for restoring the previous view levels is adjusted for both windows.

Related commands:

Windows>Enlarge Window 3

Windows>Enlarge Window 4

Windows>Enlarge Window 3 (Configure Menu)

This command can be used to enlarge the third drawing window. The effect is to swap the first (initially, the largest) and third drawing windows.

As the height/width relationship of all drawing windows is the same, the visible drawing area remains the same, but the zoom factor of both windows may change.

The information for restoring the previous view levels is adjusted for both windows.

Related commands:

Windows>Enlarge Window 2

Windows>Enlarge Window 4

Windows>Enlarge Window 4 (Configure Menu)

This command can be used to enlarge the fourth drawing window. The effect is to swap the first (initially, the largest) and fourth drawing windows.

As the height/width relationship of all drawing windows is the same, the visible drawing area remains the same, but the zoom factor of both windows may change.

The information for restoring the previous view levels is adjusted for both windows.

Related commands:

Windows>Enlarge Window 2

Windows>Enlarge Window 3

Windows>Display Overview (Configure Menu)

This command turns display of the Overview Window on or off

The view window status is saved when the program is closed.

Related commands:


Windows>Display Panel


Windows>Display Pen List


Windows>Display Layer List


Windows>Display Status


Windows>Display Toolbox


Windows>Display Property Window


Windows>Display Block List


Windows>Display Guide

Windows>Display Panel (Configure Menu)

This command turns display of the Panel on or off

The panel status is saved when the program is closed.

Related commands:

 Windows>Display Overview

 Windows>Display Pen List

 Windows>Display Layer List

 Windows>Display Status

 Windows>Display Toolbox

 Windows>Display Property Window

 Windows>Display Block List

 Windows>Display Guide

Windows>Display Pen List (Configure Menu)

This command turns display of the Pen List on or off

The pen list status is saved when the program is closed.

Related commands:

 Windows>Display Overview

 Windows>Display Panel

 Windows>Display Layer List

 Windows>Display Status

 Windows>Display Toolbox

 Windows>Display Property Window

 Windows>Display Block List

 Windows>Display Guide

Windows>Display Layer List (Configure Menu)

This command turns display of the Layer List on or off

The layer list status is saved when the program is closed.

Related commands:


Windows>Display Overview


Windows>Display Panel


Windows>Display Pen List


Windows>Display Status


Windows>Display Toolbox


Windows>Display Property Window


Windows>Display Block List


Windows>Display Guide

Windows>Display Status (Configure Menu)

This command turns display of the Status Line on or off

The status line is displayed automatically as soon as the Coordinate Entry (F8) command is chosen.

The status line status is saved when the program is closed.

Related commands:

 Windows>Display Overview

 Windows>Display Panel

 Windows>Display Pen List

 Windows>Display Layer List

 Windows>Display Toolbox

 Windows>Display Property Window

 Windows>Display Block List

 Windows>Display Guide

Windows>Display Toolbox (Configure Menu)

This command turns display of the Toolbox on or off.

The Toolbox status is saved when the program is closed.

Related commands:

 Windows>Display Overview

 Windows>Display Panel

 Windows>Display Pen List

 Windows>Display Layer List

 Windows>Display Status

 Windows>Display Property Window

 Windows>Display Block List

 Windows>Display Guide

Windows>Display Property Window (Configure Menu)

This command turns display of the Property Window on or off.

The property window status is saved when the program is closed.

Related commands:


Windows>Display Overview


Windows>Display Panel


Windows>Display Pen List


Windows>Display Layer List


Windows>Display Status


Windows>Display Toolbox


Windows>Display Block List


Windows>Display Guide

Windows>Display Block List (Configure Menu)

This command turns display of the Block List on or off.

The block list status is saved when the program is closed.

Related commands:


Windows>Display Overview


Windows>Display Panel


Windows>Display Pen List


Windows>Display Layer List


Windows>Display Status


Windows>Display Toolbox


Windows>Display Property Window


Windows>Display Guide

Windows>Display Guide (Configure Menu)

This command turns display of the Guide on or off.

The guide status is saved when the program is closed.

Related commands:


Windows>Display Overview


Windows>Display Panel


Windows>Display Pen List


Windows>Display Layer List


Windows>Display Status


Windows>Display Toolbox


Windows>Display Property Window


Windows>Display Block List

Program Status (Configure Menu)

The current program status can be called with this command. The information appears in the Program Status dialog.

Related command:

About TommySoftware® CAD/DRAW

Options>Windows (Configure Menu)

This command can be used to configure all input and output windows. This is done from the Window Options dialog.

The settings specified here are saved permanently, even after the program is closed. In multi-user mode, different settings are saved for each user.

Related commands:

Option>General

Option>Paths

Option>File Options

Options>General (Configure Menu)

This command can be used to configure all parts of the system concerned with memory and the monitor. This is done from the Memory and Monitor dialog.

If values in this dialog are altered which affect the monitor or the size of the buttons in the panel, the whole TommySoftware® CAD/DRAW working area is redrawn.

If values which affect memory management are altered, then the memory is reorganized. The current memory content is temporarily stored on the hard disk.

The settings specified here are saved permanently, even after the program is closed. In multi-user mode, different settings are saved for each user.

Related commands:


Option>Windows


Option>Paths


Option>File Options

Options>Paths (Configure Menu)

This command can be used to set the default paths. The default paths tell the program where to look for certain important files. These directories are used as initial directories in the File Selection Dialog.

The Standard Paths dialog, in which the default paths can be specified, appears.

The settings specified here are saved permanently, even after the program is closed. In multi-user mode, different settings are saved for each user.

Related commands:


Option>Windows


Option>General


Option>File Options

Options>File Options (Configure Menu)

This command can be used to alter the settings relating to the loading and saving of drawing and setting files. This is done from the File Options dialog.

The file options that may be altered here can have harmful effects that will not be noticable immediately! Please be very careful when editing these file options, and leave all settings at their default values unless another setting in inevitably necessary!

The settings specified here are saved permanently, even after the program is closed. In multi-user mode, different settings are saved for each user.

Related commands:


Option>Windows


Option>General


Option>Paths

Settings>Screen (Configure Menu)

This command can be used to alter the appearance of objects on the screen. This is done from the Screen Settings dialog.

The settings changed here only affect the screen display of the elements in the drawing. They have no direct effect on output to a printer or plotter, the clipboard, a metafile or bitmap.

Related commands:

 Settings>Output

 Settings>Plotter

 Settings>Accuracy

 Settings>Colors

 Settings>Time Format

 Settings>Block List

 Settings>Change Key Assignment

 Settings>Display Key Assignment

 Settings>Mouse Button Usage

 Settings>Open

 Settings>Save As

 Settings>Details

 Save Settings as Default

 Save Settings on Exit

Settings>Output (Configure Menu)

This command can be used to alter settings which influence the appearance of the drawing when it is output. This is done from the Output Settings dialog.

These settings only affect the *output* of the drawing on a printer or plotter, to the clipboard, a bitmap or a metafile. They do not affect any drawing elements or the way in which the elements are displayed in the drawing window.

Erasers are created by overpainting the foreground with the background color. The fill color specified in the output properties for the output section is used whenever output is not to the screen. This color is also effective if the output section itself is *not* output!

Affected Commands:


Save Drawing


Save Drawing as


Export>TVG 4.0 (*.T4G)


Export>Enhanced Metafile (*.EMF)


Export>Windows Metafile (*.WMF)


Export>Windows Bitmap (*.BMP)


Export>Windows Bitmap, Section (*.BMP)


Print Drawing


Print Section


Cut


Copy

Related commands:


Settings>Screen


Settings>Plotter


Settings>Accuracy


Settings>Colors


Settings>Time Format

Settings>Plotter (Configure Menu)

This command can be used to alter the settings which can affect the drawing when it is output to a printer or a plotter. This is done via the Plotter Settings dialog.

The "Pen Adaptation active" check box should only be checked if output is to a plotter whose plotter driver does not select pens correctly.

These settings only affect output to a printer or plotter, not to the clipboard, a bitmap or metafile. They do not alter any of the elements in the drawing and do not affect their display on screen.

If you are using an HPGL plotter and the HPPLOT.DRV driver supplied by Microsoft or by Hewlett-Packard or Micrografx, you should read the Notes for Plotter Output, which contains important tips on using such a plotter with TommySoftware® CAD/DRAW.

Affected commands:


Print Drawing


Print Section

Related commands:


Settings>Screen


Settings>Output


Settings>Accuracy


Settings>Colors


Settings>Time Format


Settings>Block List


Settings>Change Key Assignment


Settings>Display Key Assignment


Settings>Mouse Button Usage


Settings>Open


Settings>Save As


Settings>Details


Save Settings as Default

Save Settings on Exit

Settings>Accuracy (Configure Menu)

This command can be used to alter the settings which influence the accuracy of output or conversion. This is done from the Accuracy Settings dialog.

These settings only affect the *display* or *output* of drawing elements. None of the elements are directly altered. The transformation precision does however affect the accuracy of objects generated with certain commands.

Related commands:


Settings>Screen


Settings>Output


Settings>Plotter


Settings>Colors


Settings>Time Format


Settings>Block List


Settings>Change Key Assignment


Settings>Display Key Assignment


Settings>Mouse Button Usage


Settings>Open


Settings>Save As


Settings>Details


Save Settings as Default


Save Settings on Exit

Settings>Colors (Configure Menu)

This command can be used to alter the colors used to display screen elements. This is done from the Color Settings dialog.

These settings affect the display of the window on the screen and the *Display* of some drawing elements within the drawing window. They do not directly affect any of the drawing elements and do not affect output to a printer or plotter, the clipboard, a bitmap or a metafile.

Related commands:

 Settings>Screen

 Settings>Output

 Settings>Plotter

 Settings>Accuracy

 Settings>Time Format

 Settings>Block List

 Settings>Change Key Assignment

 Settings>Display Key Assignment

 Settings>Mouse Button Usage

 Settings>Open

 Settings>Save As

 Settings>Details

 Save Settings as Default

 Save Settings on Exit

Settings>Time Format (Configure Menu)

This command can be used to set the format used for time and date details. A dialog box with a list of possible formats appears. The formats show how the current time and date would appear as examples.

The format is used for all time and date details in drawings and libraries and also for printing time/date details in headers and footers.

Related commands:

 Settings>Screen

 Settings>Output

 Settings>Plotter

 Settings>Accuracy

 Settings>Colors

 Settings>Block List

 Settings>Change Key Assignment

 Settings>Display Key Assignment

 Settings>Mouse Button Usage

 Settings>Open

 Settings>Save As

 Settings>Details

 Save Settings as Default

 Save Settings on Exit

Settings>Block List (Configure Menu)

This command can be used to set the block list display parameters. This is done from the Block List Parameters dialog

The alterations take effect as soon as the dialog is closed by clicking on "OK". If the sort mode was changed, the buttons are sorted. If displayed, the block list is then redrawn.

Related commands:

 Settings>Screen

 Settings>Output

 Settings>Plotter

 Settings>Accuracy

 Settings>Colors

 Settings>Time Format

 Settings>Change Key Assignment

 Settings>Display Key Assignment

 Settings>Mouse Button Usage

 Settings>Open

 Settings>Save As

 Settings>Details

 Save Settings as Default

 Save Settings on Exit

Settings>Change Key Assignment (Configure Menu)

This command can be used to assign commands to keys. To do this, the command which is to be assigned to a key is displayed.

1. *Select Command*
   The command which is to be assigned to a key must be selected. The command can be selected by any valid method, i.e. from the menus, the text menu, the pop-up menu, the toolbox or via an existing shortcut key.
   When selecting with a shortcut key, submenus continue to be displayed in a "normal" way, i.e. a submenu cannot be assigned directly to a key. To do this, the "Complete Submenu" checkbox must be enabled in the dialog which appears.

After selecting the command, the Change Key Assignment dialog appears, where the key to which the command is to be allocated can be chosen.

Related commands:

 Settings>Screen

 Settings>Output

 Settings>Plotter

 Settings>Accuracy

 Settings>Colors

 Settings>Time Format

 Settings>Block List

 Settings>Display Key Assignment

 Settings>Mouse Button Usage

 Settings>Open

 Settings>Save As

 Settings>Details

 Save Settings as Default

 Save Settings on Exit

Settings>Display Key Assignments (Configure Menu)

This command can be used to display the current key assignments. Individual key assignments can be deleted, and in addition the default key assignments can be restored.

To display and edit key assignments the Display Key Assignment dialog appears.

Related commands:

 Settings>Screen

 Settings>Output

 Settings>Plotter

 Settings>Accuracy

 Settings>Colors

 Settings>Time Format

 Settings>Block List

 Settings>Change Key Assignment

 Settings>Mouse Button Usage

 Settings>Open

 Settings>Save As

 Settings>Details

 Save Settings as Default

 Save Settings on Exit

Settings>Mouse Button Usage (Configure Menu)

This command can be used to specify the function of the right hand and (on three-button mice) the center buttons. In particular you can choose between the use of the graphical pop-up menu or the text menu.

The Mouse Button Usage Settings dialog appears to enable the mouse button functions to be specified.

Affected Commands:


Control Keys>Middle Mouse Button


Control Keys>Right Mouse Button

Related commands:


Settings>Screen


Settings>Output


Settings>Plotter


Settings>Accuracy


Settings>Colors


Settings>Time Format


Settings>Block List


Settings>Change Key Assignment


Settings>Display Key Assignment


Settings>Open


Settings>Save As


Settings>Details


Save Settings as Default


Save Settings on Exit

Settings>Open (Configure Menu)

This command can be used to open existing settings files, that is, to load them into memory and make them active. The setting file to be used is chosen from a File Selection Dialog.

Settings files contain all the directly altered settings and the status of the larger dialogs. This includes the contents of the Toolbox and the current settings of the following dialogs:

- Autosave
- Comment Parameters
- Screen Settings
- Output Settings
- Accuracy Settings
- Plotter Settings
- Color Settings
- Block List Settings
- Mouse Button Settings
- Display Key Assignments
- Choose Color

Related commands:

 Settings>Screen

 Settings>Output

 Settings>Plotter

 Settings>Accuracy

 Settings>Colors

 Settings>Time Format

 Settings>Block List

 Settings>Change Key Assignment

 Settings>Display Key Assignment

 Settings>Mouse Button Usage

 Settings>Save As

 Settings>Details

 Save Settings as Default

 Save Settings on Exit

Settings>Save As (Configure Menu)

This command can be used to save the current settings in a settings file so that they can be loaded again later.

A File Selection Dialog appears, in which the name of the file to be saved can be entered. After this, the Edit Details dialog appears in which the details of the settings file can be altered.

Settings files contain all the directly altered settings and the status of the larger dialogs. This includes the contents of the Toolbox and the current settings of the following dialogs:

- Autosave
- Comment Parameters
- Screen Settings
- Output Settings
- Accuracy Settings
- Plotter Settings
- Color Settings
- Block List Settings
- Mouse Button Settings
- Display Key Assignments
- Choose Color

Related commands:

 Settings>Screen

 Settings>Output

 Settings>Plotter

 Settings>Accuracy

 Settings>Colors

 Settings>Time Format

 Settings>Block List

 Settings>Change Key Assignment

 Settings>Display Key Assignment

 Settings>Mouse Button Usage

 Settings>Open

 Settings>Details

 Save Settings as Default

Save Settings on Exit

Settings>Details (Configure Menu)

This command can be used to view and edit the current settings. To do this, the Edit Details dialog appears.

Related commands:

 Settings>Screen

 Settings>Output

 Settings>Plotter

 Settings>Accuracy

 Settings>Colors

 Settings>Time Format

 Settings>Block List

 Settings>Change Key Assignment

 Settings>Display Key Assignment

 Settings>Mouse Button Usage

 Settings>Open

 Settings>Save As

 Save Settings as Default

 Save Settings on Exit

Save Settings as Default (Configure Menu)

This command can be used to save the current settings as the default, used the next time the program is started.

The current settings are saved in the file `RELEASE4.INF` (or in the appropriate user file, when in multi-user mode). If this file already exists, a warning appears so as to avoid the accidental overwriting of the defaults.

Settings files contain all the directly altered settings and the status of the larger dialogs. This includes the contents of the Toolbox and the current settings of the following dialogs:

- Autosave
- Comment Parameters
- Screen Settings
- Output Settings
- Accuracy Settings
- Plotter Settings
- Color Settings
- Block List Settings
- Mouse Button Settings
- Display Key Assignments
- Choose Color

Related commands:

 Settings>Screen

 Settings>Output

 Settings>Plotter

 Settings>Accuracy

 Settings>Colors

 Settings>Time Format

 Settings>Block List

 Settings>Change Key Assignment

 Settings>Display Key Assignment

 Settings>Mouse Button Usage

 Settings>Open

 Settings>Save As


Settings>Details


Save Settings on Exit

Save Settings on Exit (Configure Menu)

This command tells the program to save the current settings as the default on exit. These settings are then used the next time the program is started.

The current settings are saved in the file `RELEASE4.INF` (or in the appropriate user file, when in multi-user mode).

This function has been provided because users asked for it. However, we recommend that it is *not* used, but instead to save settings as required using the command Save Settings as Default. By automatically saving settings, it is often the case that unwanted changes to the settings are saved.

Related commands:


Settings>Screen


Settings>Output


Settings>Plotter


Settings>Accuracy


Settings>Colors


Settings>Time Format


Settings>Block List


Settings>Change Key Assignment


Settings>Display Key Assignment

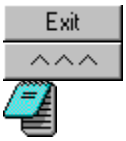

Settings>Mouse Button Usage


Settings>Open


Settings>Save As


Settings>Details


Save Settings as Default



Shape Menu

- Edit Properties
- Edit Text
- Delete Objects
- Modify Objects
- Move / Copy Objects
  - Standard
  - Perpendicular
  - Parallel
  - Relative
- Scale Objects
  - Factors
  - Proportional
  - Non-Proportional
  - Reference Distance
- Rotate Objects
  - Center
  - Arbitrary
  - Reference Points
- Reflect Objects
  - Horizontal
  - Vertical
  - Line
- Distort Objects
  - Horizontal
  - Vertical
- Align Objects
  - Centered, Page Horizontal
  - Centered, Page Vertical
  - Centered, Page Both
  - Centered, Frame Horizontal
  - Centered, Frame Vertical
  - Centered, Frame Both
  - Horizontal, Left
  - Horizontal, Middle
  - Horizontal, Right
  - Vertical, Top
  - Vertical, Middle
  - Vertical, Bottom
- Move Single Point
- Move Points
  - Standard
  - Perpendicular
  - Parallel
  - Relative
- Group

- Create Group
- Ungroup

Change Order

- To Back
- Behind Object
- Before Object
- To Front

Edit Properties (Shape Menu)

This command can be used to edit the properties of existing objects and instances. In addition, all object-specific parameters can be changed. This is done separately for each property, although several objects can be changed at the same time.

1. *Choose objects*
   Choose the objects whose properties are to be altered. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing the objects, the Edit Properties dialog appears in which the object properties can be viewed and altered.

If object specific settings (e.g. text or dimension parameters) are altered, the change is applied to all selected objects of the same type. For example, if the size of a "Standard Text" object is changed, this change is applied to *all* standard text objects but *not* to frame, reference or dimension text.

If the duplicate function is active, a copy of the chosen objects is created and the copy is altered. This means that a copy with entirely different parameters can be created.

Related command:

Edit Text

Edit Text (Shape Menu)

This command can be used to edit existing text objects of all types: standard text, frame text, reference text, dimension text, instance attributes, position numbers, and comments.

1. *Identify text object*
   An object is identified by clicking on its outline. If more than one object outline lies at the spot clicked on, a small dialog appears at the bottom of the screen, from which the correct object can be chosen.

After identifying the object one of the following dialogs appears in which the text can be edited:

Edit Text
Dimension and Tolerances
Coordinates and Tolerances
Define Attributes

Related commands:


Edit Properties


Edit Dimension>Update

Delete Objects (Shape Menu)

This command can be used to delete selected objects and instances from the drawing.

1. *Choose objects*
   The objects to be deleted must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing the objects, they are deleted without further confirmation being needed. If needed, they can be restored using the Undo command.

The duplicate function has no effect on this command, that is, the objects are still deleted if the duplicate function is active.

Related commands:

No related commands

Modify Objects (Shape Menu)

This command can be used to modify objects and instances directly with the mouse. All basic operations like moving, scaling, rotation and distortion are available. In cooperation with the Property Window, the properties of the objects and instances can be modified.

1. *Choose objects*
   The objects to be modified must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The first reference point defines the action to be performed. You can click onto one of the symbols displayed around the chosen objects to activate a special operation. Clicking somewhere else inside the surrounding frame activates the Move action.

3. *Enter reference point*
   Once the desired action has been determined by the first reference point, the second reference point determines the execution of that action. A cross frame displays the projected result. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the duplicate function is active during the second reference point entry (for example if the CTRL key was held down), a copy of the chosen objects is generated and modified. In this way objects can be copied.

If the duplicate was activated by holding the CTRL key down, several second reference points can be entered immediately after each other to create multiple modified copies of the chosen objects. As soon as the CTRL key is released, the action is terminated. Anyway, the command remains active, so a new action can immediately be chosen.

Related commands:

 Move / Copy Objects>Standard

 Scale Objects>Proportional

 Scale Objects>Non-Proportional

 Rotate Objects>Arbitrary

 Distort Object>Horizontal

 Distort Object>Vertical

Move / Copy Objects>Standard (Shape Menu)

This command can be used to move or copy objects and instances in the drawing. The movement can be in any direction and is specified using two reference points.

1. *Choose objects*
   The objects to be moved or copied must be chosen first. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The reference point for the movement can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter target point*
   The target point for the movement can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the duplicate function is active during target point entry (for example if the CTRL key was held down), a copy of the chosen objects is generated and moved. In this way objects can be copied.

If the duplicate was activated by holding the CTRL key down, several target points can be entered immediately after each other to create multiple copies of the chosen objects. As soon as the CTRL key is released, the command is terminated. Anyway, it remains active, so a new set of objects can immediately be chosen for moving or copying.

Related commands:


Move / Copy Objects>Perpendicular


Move / Copy Objects>Parallel


Move / Copy Objects>Relative


Modify Objects


Multiple Copy>Rotation


Multiple Copy>Gradual

Move / Copy Objects>Perpendicular (Shape Menu) Level 2

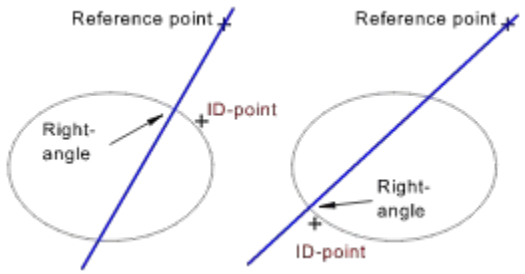
This command can be used to move or copy objects and instances. The movement can only be perpendicular to a reference object.

1. *Choose objects*
   The objects to be moved or copied must be chosen first. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Identify reference object*
   Any part of an existing object can be identified as a reference object. This can be a side of a rectangle, the arc of an ellipse segment or an arc within a surface.
   Identify the reference object by clicking on it with the mouse. If several possible objects lie near the position clicked on, a small dialog appears at the bottom of the screen from which the correct object can be chosen.

   The identified object is referred to as the "Reference Object".

   If an ellipse part is identified then the point at which the object was identified (the "ID-Point") is important. If a perpendicular is dropped onto an ellipse, it can end on either side of the ellipse. The point nearer to the ID-Point is used as the perpendicular's end point (see graphic).



3. *Enter reference point*
   The reference point for the movement can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this point to the reference object.

   The target point can then be placed on the line between the reference point and the foot of the perpendicular. This line, the "Reference line" is displayed. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter target point*
   The target point for the movement can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from here to the reference line to work out the target point for the movement. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, you can enter a length. This determines the size of the move in the direction of the reference line beginning from a previously entered start point.

If the duplicate function is active during target point entry (for example if the CTRL key was held down), a copy of the chosen objects is generated and moved. In this way objects can be copied.

If the duplicate was activated by holding the CTRL key down, several target points can be entered immediately after each other to create multiple copies of the chosen objects. As soon as the CTRL key is

released, the command is terminated. Anyway, it remains active, so a new set of objects can immediately be chosen for moving or copying.

Related commands:

 Move / Copy Objects>Standard

 Move / Copy Objects>Parallel

 Move / Copy Objects>Relative

 Modify Objects

Move / Copy Objects>Parallel (Shape Menu) Level 2

This command can be used to move or copy objects within a drawing. Movement can only take place parallel to a reference object.

1. *Choose objects*
   The objects to be moved or copied must be chosen first. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Identify reference object*
   Any part of an existing object can be used as a reference object. This can be an edge of a rectangle or an arc within a surface. Ellipses and ellipse parts cannot be identified.
   Identify the reference object by clicking on it with the mouse. If several possible objects lie near the position clicked on, a small dialog appears at the bottom of the screen from which the correct object can be chosen.

   The identified object is referred to as the "Reference Object".

3. *Enter reference point*
   The reference point for the movement can be entered by clicking the mouse anywhere in the drawing. If the previously identified reference object is a line then the reference point determines the position of a line which runs parallel to the reference object. This line is displayed and referred to as the "Reference line". If however the reference object is a circle or circle part, then it determines the position of a circle concentric to the original circle (the "Reference circle"). The target point can then be placed on the reference line/circle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter target point*
   The target point for the movement can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this point to the reference line or reference circle to work out the target point for the movement. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If the reference object is a line, a length can be entered. This gives the size of the movement in the direction of the reference line, beginning from a specified starting point. In all other cases an angle, measured from the reference circle's center, can be entered. The angle can be given relative to the reference point angle.

If the duplicate function is active during target point entry (for example if the CTRL key is held down) then a copy of the chosen objects is created and the copy is moved. In this way objects can be copied.

If the duplicate was activated by holding the CTRL key down, several target points can be entered immediately after each other to create multiple copies of the chosen objects. As soon as the CTRL key is released, the command is terminated. Anyway, it remains active, so a new set of objects can immediately be chosen for moving or copying.

Related commands:

Move / Copy Objects>Standard

Move / Copy Objects>Perpendicular

Move / Copy Objects>Relative

Modify Objects

Move / Copy Objects>Relative (Shape Menu)

This command can be used to move or copy objects or instances in the drawing. The movement is specified numerically.

1. *Choose objects*
   The objects to be moved or copied must be chosen first. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter horizontal and vertical movement*
   After choosing the objects, a dialog appears in which the amount of horizontal and vertical movement can be entered. The duplicate function can be briefly turned on by enabling the "Duplicate" check box.

If the duplicate function is on when the amount of movement is entered, a copy of the chosen objects is created and the copy is moved. In this way objects can be copied.

Related commands:

 Move / Copy Objects>Standard

 Move / Copy Objects>Perpendicular

 Move / Copy Objects>Parallel

 Multiple Copy>Rotation

 Multiple Copy>Gradual

 Modify Objects

Scale Objects>Factors (Shape Menu)

This command can be used to scale objects and instances in the drawing (i.e. change their size). The scaling is done with respect to the center of the frame surrounding the objects.

1. *Choose objects*
   The objects to be scaled have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter horizontal and vertical factors*
   After choosing the objects, a dialog appears where the horizontal and vertical scaling factors can be entered. In addition the duplicate function can be briefly turned on by checking the "Duplicate" checkbox.

If the duplicate function is active when the scaling factors are entered, then a copy of the chosen objects is made and scaled. Scaled copies of objects can be made like this.

Some objects cannot in principle be scaled unproportionally. Above all, these are circles and circle parts, and also surfaces which contain circular arcs. If you try to scale such an object unproportionally, a message apppears which warns that you must first convert the circles and circle parts to ellipses and ellipse parts. This allows you to cancel the operation, without actually making the changes.

Related commands:


Scale Objects>Proportional


Scale Objects>Non-Proportional


Scale Objects>Reference Distance


Modify Objects

Scale Objects>Proportional (Shape Menu)

This command can be used to scale (i.e. change the size of) objects and instances in the drawing. The objects are so scaled (and moved) that they fit into a specified rectangular area. The scaling is proportional, i.e. the horizontal and vertical scaling factors are the same.

1. *Choose objects*
   The objects to be scaled have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Area: Enter corner point 1*
   The first corner point of the area can be specified by clicking the mouse button anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Area: Enter corner point 2*
   The second corner point of the area can be specified by clicking the mouse button anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the duplicate function is active during entry of the area's second corner point (for example if the CTRL key is held down), a copy of the chosen objects is created and then moved and scaled. In this way, objects can be copied and scaled at the same time.

Related commands:


Scale Objects>Factors


Scale Objects>Non-Proportional


Scale Objects>Reference Distance


Modify Objects

Scale Objects>Non-Proportional (Shape Menu)

This command can be used to scale objects and instances (i.e. change their sizes) in the drawing. The objects are scaled and moved so that they fit in a given rectangular area. The scaling is not proportional, that is the horizontal and vertical scaling factors can be different.

1. *Choose objects*
   The objects to be scaled have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Area: Enter corner point 1*
   The first corner point of the area can be specified by clicking the mouse button anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Area: Enter corner point 2*
   The second corner point of the area can be specified by clicking the mouse button anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the duplicate function is active during entry of the area's second corner point (for example if the CTRL key is held down), a copy of the chosen objects is created and then moved and scaled. In this way, objects can be copied and scaled at the same time.

Some objects cannot in principle be scaled unproportionally. Above all, these are circles and circle parts, and also surfaces which contain circular arcs. If you try to scale such an object unproportionally, a message apppears which warns that you must first convert the circles and circle parts to ellipses and ellipse parts. This allows you to cancel the operation, without actually making the changes.

Related commands:


Scale Objects>Factors


Scale Objects>Proportional


Scale Objects>Reference Distance


Modify Objects

Scale Objects>Reference Distance (Shape Menu) Level 2

This command can be used to scale objects and instances in the drawing, that is to change their size. The objects are scaled in such a way that that a specified reference length has a given size. The scaling is proportional, that is, the horizontal and vertical scaling factors are the same. The scaling is performed in relation to the center of a frame surrounding the objects.

This command is particularly useful for matching objects to a particular scale. If the "Original" length of at least one length in the drawing is known, then the whole drawing can be brought to the correct size without further calculation.

1. *Choose objects*
   The objects to be scaled have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter starting point*
   The starting point for the reference distance can be entered by clicking with the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter end point*
   The end point for the reference distance can be entered by clicking with the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter length*
   After entering the reference distance a dialog appears in which the desired length for the previously entered reference distance can be entered.

If the duplicate function is active during length entry, then a copy of the chosen objects is created and the copy is scaled. This method can be used to create a ready scaled copy of objects.

Related commands:


Scale Objects>Factors


Scale Objects>Proportional


Scale Objects>Non-Proportional


Modify Objects

Rotate Objects>Center (Shape Menu)

This command can be used to rotate objects and instances in the drawing. The rotation is carried out in relation to the center of a frame surrounding the objects.

1. *Choose objects*
   The objects to be rotated must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter angle of rotation*
   After choosing the objects, a dialog appears in which the desired angle of rotation can be entered. In addition the duplicate function can be briefly activated by checking the "Duplicate" check box.

If the duplicate function was turned on during angle entry, then a rotated copy of the objects is created. This is a way to create a copy of the objects which is already rotated.

Related commands:


Rotate Objects>Arbitrary


Rotate Objects>Reference Points


Modify Objects

Rotate Objects>Arbitrary (Shape Menu)

This command can be used to rotate objects and instances in the drawing. The center of rotation and the angle are both specified by point entry.

1. *Choose objects*
   The objects to be rotated must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The reference point, i.e. the center of rotation, can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter angle*
   The rotation angle can be entered by clicking the mouse anywhere in the drawing. Its position relative to the center of rotation determines the angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter a value which gives the rotation angle.

If the duplicate function is on during angle entry (for example because the CTRL key was held down), then a copy of the chosen objects is created and rotated. Rotated copies of objects can be created like this.

Related commands:


Rotate Objects>Center


Rotate Objects>Reference Points


Modify Objects

Rotate Objects>Reference Points (Shape Menu) Level 2

This command can be used to rotate objects and instances in the drawing. The center of rotation and the angle are both specified by point entry.

1. *Choose objects*
   The objects to be rotated must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The reference point, i.e. the center of rotation, can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter start angle*
   The rotations's start angle can be entered by clicking the mouse anywhere in the drawing. The position of this point relative to the reference point determines the start angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the arc's start angle.

4. *Enter end angle*
   The rotations's end angle can be entered by clicking the mouse anywhere in the drawing. The position of this point relative to the reference point determines the end angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the end angle of the rotation. The angle can also be given relative to the previously entered start angle.

If the duplicate function is on during end-angle entry (for example because the CTRL key was held down), then a copy of the chosen objects is created and rotated. Rotated copies of objects can be created like this.

Related commands:


Rotate Objects>Center


Rotate Objects>Arbitrary


Modify Objects

Reflect Objects>Horizontal (Shape Menu)

This command can be used to reflect objects and instances in the drawing horizontally. The reflection takes place about the central vertical axis of the frame surrounding the chosen objects.

1. *Choose objects*
   The objects to be reflected must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

If the duplicate function is active when the last object is identified, (for example because the CTRL key is held down), then a copy of the chosen objects is created and reflected horizontally. Reflected copies of objects can be created like this.

Related commands:

Reflect Objects>Vertical

Reflect Objects>Line

Reflect Objects>Vertical (Shape Menu)

This command can be used to reflect objects and instances in the drawing vertically. The reflection takes place about the central horizontal axis of the frame surrounding the chosen objects.

1. *Choose objects*
   The objects to be reflected must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

If the duplicate function is active when the last object is identified, (for example because the CTRL key is held down), then a copy of the chosen objects is created and reflected vertically. Reflected copies of objects can be created like this.

Related commands:

Reflect Objects>Horizontal

Reflect Objects>Line

Reflect Objects>Line (Shape Menu)

This command can be used to reflect objects and instances in the drawing. The reflection takes place about a specified line.

1. *Choose objects*
   The objects to be reflected must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Identify reference line*
   Any existing straight line can be used as the reference line. This can also be a side of a rectangle or a leg of a circle segment.
   Click the mouse on the line to identify it. If more than one line lies at the spot clicked on, a small dialog appears at the bottom of the screen, from which the correct line can be chosen.

If the duplicate function is active while the reference line is identified, (for example because the CTRL key is held down), then a copy of the chosen objects is created and reflected. Reflected copies of objects can be created like this.

Related commands:


Reflect Objects>Horizontal


Reflect Objects>Vertical

Distort Object>Horizontal (Shape Menu)

This command can be used to distort objects and instances in the drawing horizontally. The distortion takes place relative to a specified reference point.

1. *Choose objects*
   The objects to be distorted must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The reference point for the distortion can be specified by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter angle*
   The distortion angle can be specified by clicking the mouse anywhere in the drawing. The angle is determined by this point's position relative to the reference point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter a value which gives the distortion angle.

If the duplicate function is active during angle entry (for example if the CTRL key is kept held down) then a copy of the chosen objects is created and distorted. Distorted copies of objects can be made like this.

Some objects cannot in principle be distorted. Above all, these are circles and circle parts, and also surfaces which contain circular arcs. If you try to distort such an object, a message apppears which warns that you must first convert the circles and circle parts to ellipses and ellipse parts. This allows you to cancel the operation, without actually making the changes.

Related commands:

Distort Object>Vertical

Modify Objects

Distort Object>Vertical (Shape Menu)

This command can be used to distort objects and instances in the drawing vertically. The distortion takes place relative to a specified reference point.

1. *Choose objects*
   The objects to be distorted must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The reference point for the distortion can be specified by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter angle*
   The distortion angle can be specified by clicking the mouse anywhere in the drawing. The angle is determined by this point's position relative to the reference point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter a value which gives the distortion angle.

If the duplicate function is active during angle entry (for example if the CTRL key is kept held down) then a copy of the chosen objects is created and distorted. Distorted copies of objects can be made like this.

Some objects cannot in principle be distorted. Above all, these are circles and circle parts, and also surfaces which contain circular arcs. If you try to distort such an object, a message apppears which warns that you must first convert the circles and circle parts to ellipses and ellipse parts. This allows you to cancel the operation, without actually making the changes.

Related commands:

Distort Object>Horizontal

Modify Objects

Align Objects>Centered, Page Horizontal (Shape Menu)

This command can be used to center objects and instances horizontally on the page. They are moved horizontally so that they are the same distance from the left and right edges of the page. All chosen objects and instances are treated as a single item, i.e. they are all moved by the same amount.

1. *Choose objects*
   All the objects to be centered have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

If the duplicate function is active when the last object is chosen, then a copy of the chosen objects is made and moved. Aligned copies of objects can be made like this.

Related commands:

 Align Objects>Centered, Page Vertical

 Align Objects>Centered, Page Both

 Align Objects>Centered, Frame Horizontal

 Align Objects>Centered, Frame Vertical

 Align Objects>Centered, Frame Both

 Align Objects>Horizontal, Left

 Align Objects>Horizontal, Middle

 Align Objects>Horizontal, Right

 Align Objects>Vertical, Top

 Align Objects>Vertical, Middle

 Align Objects>Vertical, Bottom

Align Objects>Centered, Page Vertical (Shape Menu)

This command can be used to center objects and instances vertically on the page. They are moved vertically so that they are the same distance from the top and bottom of the page. All chosen objects and instances are treated as a single item, i.e. they are all moved by the same amount.

1. *Choose objects*
   All the objects to be centered have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

If the duplicate function is active when the last object is chosen, then a copy of the chosen objects is made and moved. Aligned copies of objects can be made like this.

Related commands:

 Align Objects>Centered, Page Horizontal

 Align Objects>Centered, Page Both

 Align Objects>Centered, Frame Horizontal

 Align Objects>Centered, Frame Vertical

 Align Objects>Centered, Frame Both

 Align Objects>Horizontal, Left

 Align Objects>Horizontal, Middle

 Align Objects>Horizontal, Right

 Align Objects>Vertical, Top

 Align Objects>Vertical, Middle

 Align Objects>Vertical, Bottom

Align Objects>Centered, Page Both (Shape Menu)

This command can be used to center objects and instances horizontally and vertically on the page. They are moved horizontally so that they are the same distance from the left and right edges and the top and bottom of the page. All chosen objects and instances are treated as a single item, i.e. they are all moved by the same amount.

1. *Choose objects*
   All the objects to be centered have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

If the duplicate function is active when the last object is chosen, then a copy of the chosen objects is made and moved. Aligned copies of objects can be made like this.

Related commands:

 Align Objects>Centered, Page Horizontal

 Align Objects>Centered, Page Vertical

 Align Objects>Centered, Frame Horizontal

 Align Objects>Centered, Frame Vertical

 Align Objects>Centered, Frame Both

 Align Objects>Horizontal, Left

 Align Objects>Horizontal, Middle

 Align Objects>Horizontal, Right

 Align Objects>Vertical, Top

 Align Objects>Vertical, Middle

 Align Objects>Vertical, Bottom

Align Objects>Centered, Frame Horizontal (Shape Menu)

This command can be used to center objects and instances horizontally relative to a rectangular frame. They are moved horizontally so that they are the same distance from the left and right edges of the frame. All chosen objects and instances are treated as a single item, i.e. they are all moved by the same amount.

1. *Choose objects*
   All the objects to be centered have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Area: Enter corner point 1*
   The first corner point of the frame can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Area: Enter corner point 2*
   The second corner point of the frame can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the duplicate function is active during entry of the second corner point (for example, because the CTRL key was held down) a copy of the chosen objects is created and the copy is centered. This is how to copy objects centered.

Related commands:

 Align Objects>Centered, Page Horizontal

 Align Objects>Centered, Page Vertical

 Align Objects>Centered, Page Both

 Align Objects>Centered, Frame Vertical

 Align Objects>Centered, Frame Both

 Align Objects>Horizontal, Left

 Align Objects>Horizontal, Middle

 Align Objects>Horizontal, Right

 Align Objects>Vertical, Top

 Align Objects>Vertical, Middle

 Align Objects>Vertical, Bottom

Align Objects>Centered, Frame Vertical (Shape Menu)

This command can be used to center chosen objects vertically with respect to a specified rectangular frame. They are moved horizontally so that they are the same distance from the top and bottom of the frame. All chosen objects and instances are treated as a single unit, that is they are all moved by the same amount.

1. *Choose objects*
   All the objects to be centered have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Area: Enter corner point 1*
   The first corner point of the frame can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Area: Enter corner point 2*
   The second corner point of the frame can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the duplicate function is active during entry of the second corner point (for example, because the CTRL key was held down) a copy of the chosen objects is created and the copy is centered. This is how to copy objects centered.

Related commands:

 Align Objects>Centered, Page Horizontal

 Align Objects>Centered, Page Vertical

 Align Objects>Centered, Page Both

 Align Objects>Centered, Frame Horizontal

 Align Objects>Centered, Frame Both

 Align Objects>Horizontal, Left

 Align Objects>Horizontal, Middle

 Align Objects>Horizontal, Right

 Align Objects>Vertical, Top

 Align Objects>Vertical, Middle

 Align Objects>Vertical, Bottom

Align Objects>Centered, Frame Both (Shape Menu)

This command can be used to center chosen objects both horizontally and vertically with respect to a specified rectangular frame. They are moved horizontally so that they are the same distance from the left and right of the frame and vertically so that they are the same distance from the top and bottom of the frame. All chosen objects and instances are treated as a single unit, that is they are all moved by the same amount.

1. *Choose objects*
   The objects to be centered must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Area: Enter corner point 1*
   The first corner point of the frame can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Area: Enter corner point 2*
   The second corner point of the frame can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the duplicate function is active during entry of the second corner point (for example, because the CTRL key was held down) a copy of the chosen objects is created and the copy is centered. This is how to copy objects centered.

Related commands:

 Align Objects>Centered, Page Horizontal

 Align Objects>Centered, Page Vertical

 Align Objects>Centered, Page Both

 Align Objects>Centered, Frame Horizontal

 Align Objects>Centered, Frame Vertical

 Align Objects>Centered, Frame Both

 Align Objects>Horizontal, Left

 Align Objects>Horizontal, Middle

 Align Objects>Horizontal, Right

 Align Objects>Vertical, Top

 Align Objects>Vertical, Middle

Align Objects>Horizontal, Left (Shape Menu) Level 2

This command can be used to align objects and instances horizontally along a straight line. They are moved horizontally so that they end at the given reference line. Each chosen object and instance is treated separately, i.e. is moved by an individual amount.

1. *Choose objects*
   All the objects to be aligned have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Identify reference line*
   Any existing straight line can be used as the reference line. This can also be a side of a rectangle or a leg of a circle segment.
   Click the mouse on the line to identify it. If more than one line lies at the spot clicked on, a small dialog appears at the bottom of the screen, from which the correct line can be chosen.

If the duplicate function is active while the reference line is identified, (for example because the CTRL key is held down), then a copy of the chosen objects is created and aligned. Algined copies of objects can be created like this.

Related commands:

 Align Objects>Centered, Page Horizontal

 Align Objects>Centered, Page Vertical

 Align Objects>Centered, Page Both

 Align Objects>Centered, Frame Horizontal

 Align Objects>Centered, Frame Vertical

 Align Objects>Horizontal, Middle

 Align Objects>Horizontal, Right

 Align Objects>Vertical, Top

 Align Objects>Vertical, Middle

 Align Objects>Vertical, Bottom

Align Objects>

Align Objects>Horizontal, Middle (Shape Menu) Level 2

This command can be used to align objects and instances horizontally along a straight line. They are moved horizontally so that they are centered to the given reference line. Each chosen object and instance is treated separately, i.e. is moved by an individual amount.

1. *Choose objects*
   All the objects to be aligned have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Identify reference line*
   Any existing straight line can be used as the reference line. This can also be a side of a rectangle or a leg of a circle segment.
   Click the mouse on the line to identify it. If more than one line lies at the spot clicked on, a small dialog appears at the bottom of the screen, from which the correct line can be chosen.

If the duplicate function is active while the reference line is identified, (for example because the CTRL key is held down), then a copy of the chosen objects is created and aligned. Algined copies of objects can be created like this.

Related commands:

 Align Objects>Centered, Page Horizontal

 Align Objects>Centered, Page Vertical

 Align Objects>Centered, Page Both

 Align Objects>Centered, Frame Horizontal

 Align Objects>Centered, Frame Vertical

 Align Objects>Centered, Frame Both

 Align Objects>Horizontal, Left

 Align Objects>Horizontal, Right

 Align Objects>Vertical, Top

 Align Objects>Vertical, Middle

 Align Objects>Vertical, Bottom

Align Objects>Horizontal, Right (Shape Menu) Level 2

This command can be used to align objects and instances horizontally along a straight line. They are moved horizontally so that they end at the given reference line. Each chosen object and instance is treated separately, i.e. is moved by an individual amount.

1. *Choose objects*
   All the objects to be aligned have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Identify reference line*
   Any existing straight line can be used as the reference line. This can also be a side of a rectangle or a leg of a circle segment.
   Click the mouse on the line to identify it. If more than one line lies at the spot clicked on, a small dialog appears at the bottom of the screen, from which the correct line can be chosen.

If the duplicate function is active while the reference line is identified, (for example because the CTRL key is held down), then a copy of the chosen objects is created and aligned. Algined copies of objects can be created like this.

Related commands:

 Align Objects>Centered, Page Horizontal

 Align Objects>Centered, Page Vertical

 Align Objects>Centered, Page Both

 Align Objects>Centered, Frame Horizontal

 Align Objects>Centered, Frame Vertical

 Align Objects>Centered, Frame Both

 Align Objects>Horizontal, Left

 Align Objects>Horizontal, Middle

 Align Objects>Vertical, Top

 Align Objects>Vertical, Middle

 Align Objects>Vertical, Bottom

Align Objects>Vertical, Top (Shape Menu) Level 2

This command can be used to align objects and instances vertically along a straight line. They are moved vertically so that they end at the given reference line. Each chosen object and instance is treated separately, i.e. is moved by an individual amount.

1. *Choose objects*
   All the objects to be aligned have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Identify reference line*
   Any existing straight line can be used as the reference line. This can also be a side of a rectangle or a leg of a circle segment.
   Click the mouse on the line to identify it. If more than one line lies at the spot clicked on, a small dialog appears at the bottom of the screen, from which the correct line can be chosen.

If the duplicate function is active while the reference line is identified, (for example because the CTRL key is held down), then a copy of the chosen objects is created and aligned. Algined copies of objects can be created like this.

Related commands:

 Align Objects>Centered, Page Horizontal

 Align Objects>Centered, Page Vertical

 Align Objects>Centered, Page Both

 Align Objects>Centered, Frame Horizontal

 Align Objects>Centered, Frame Vertical

 Align Objects>Centered, Frame Both

 Align Objects>Horizontal, Left

 Align Objects>Horizontal, Middle

 Align Objects>Horizontal, Right

 Align Objects>Vertical, Middle

 Align Objects>Vertical, Bottom

Align Objects>Vertical, Middle (Shape Menu) Level 2

This command can be used to align objects and instances vertically along a straight line. They are moved vertically so that they are centered to the given reference line. Each chosen object and instance is treated separately, i.e. is moved by an individual amount.

1. *Choose objects*
   All the objects to be aligned have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Identify reference line*
   Any existing straight line can be used as the reference line. This can also be a side of a rectangle or a leg of a circle segment.
   Click the mouse on the line to identify it. If more than one line lies at the spot clicked on, a small dialog appears at the bottom of the screen, from which the correct line can be chosen.

If the duplicate function is active while the reference line is identified, (for example because the CTRL key is held down), then a copy of the chosen objects is created and aligned. Algined copies of objects can be created like this.

Related commands:

 Align Objects>Centered, Page Horizontal

 Align Objects>Centered, Page Vertical

 Align Objects>Centered, Page Both

 Align Objects>Centered, Frame Horizontal

 Align Objects>Centered, Frame Vertical

 Align Objects>Centered, Frame Both

 Align Objects>Horizontal, Left

 Align Objects>Horizontal, Middle

 Align Objects>Horizontal, Right

 Align Objects>Vertical, Top

 Align Objects>Vertical, Bottom

Align Objects>Vertical, Bottom (Shape Menu) Level 2

This command can be used to align objects and instances vertically along a straight line. They are moved vertically so that they end at the given reference line. Each chosen object and instance is treated separately, i.e. is moved by an individual amount.

1. *Choose objects*
   All the objects to be aligned have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Identify reference line*
   Any existing straight line can be used as the reference line. This can also be a side of a rectangle or a leg of a circle segment.
   Click the mouse on the line to identify it. If more than one line lies at the spot clicked on, a small dialog appears at the bottom of the screen, from which the correct line can be chosen.

If the duplicate function is active while the reference line is identified, (for example because the CTRL key is held down), then a copy of the chosen objects is created and aligned. Algined copies of objects can be created like this.

Related commands:

 Align Objects>Centered, Page Horizontal

 Align Objects>Centered, Page Vertical

 Align Objects>Centered, Page Both

 Align Objects>Centered, Frame Horizontal

 Align Objects>Centered, Frame Vertical

 Align Objects>Centered, Frame Both

 Align Objects>Horizontal, Left

 Align Objects>Horizontal, Middle

 Align Objects>Horizontal, Right

 Align Objects>Vertical, Top

 Align Objects>Vertical, Middle

Move Single Point (Shape Menu)

This command can be used to move individual object definition points. Using this function objects which can not be manipulated with the "normal" manipulation commands can be manipulated.

If several definition points are to be moved at once (e.g. to stretch an object) this can be done more easily by means of the command Move Points>Standard.

1. *Select definition point*
   The definition point to be moved can be selected with the mouse, by clicking near it. If the definition points of one or more objects lie at exactly the same position, they are all moved by the same amount. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

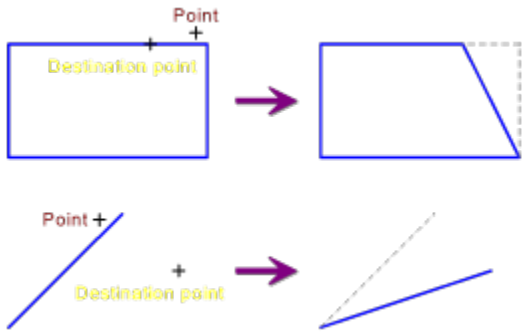
2. *Enter target point*
   The new position for the selected definition point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

To make the definition points more recognizable, you should turn their display on. This can be done with the Settings>Bildschirm.

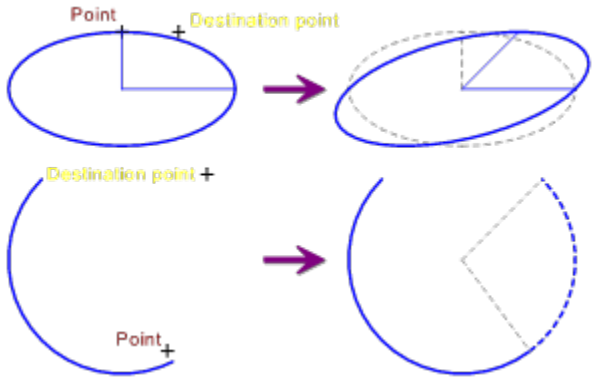
The possibilities which can result from moving individual definition points are varied. The consequences are equally various and can be unexpected. Many definition points have a close relationship to other definition points and can only be moved in certain ways. Under some circumstances, some definition points can not be moved because of object parameter settings. Be sparing in the use of moving individual definition points.

Some frequently required operations can be carried out without risk using this command:

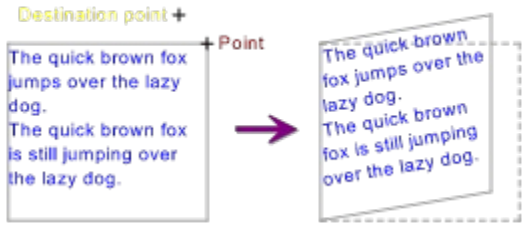
- Moving end and corner points of lines, polylines and polygons (see graphic).



- Altering the half-axes of ellipses or ellipse parts and moving the start or end angle of circle or ellipse parts (see graphic).



- Moving the frame corner points of multi-line texts (see graphic).



Many of these operations can also be carried out with commands from the Shape Menu and the Trimming Menu. In some cases it may however be quicker and easier to move one or more individual definition points.

The duplicate function does not work with this command. The object itself is always changed. A copy is not made automatically, and if needed it must be explicitly created.
If a definition point lies outside an object frame (e.g. the pivot point of a Bézier curve), then the movement will not usually be displayed correctly. In this case, a screen redraw can be forced using the Spacebar. The movement is still effective even if the screen is not redrawn; strictly it is not necessary to redraw the screen.

Related commands:

Move Points>Standard

Move Points>Perpendicular

Move Points>Parallel

Move Points>Relative

Move Points>Standard (Shape Menu)

This command can be used to move several definition points at the same time.

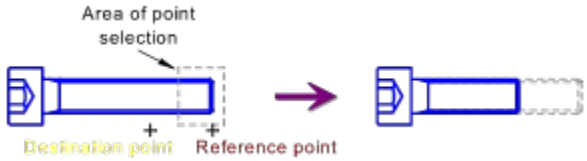
1. *Choose points*
   Choose the points that shall be moved. Click on points to choose them. For further information on point selection, see the chapter Point Selection.

2. *Enter reference point*
   The movement reference point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

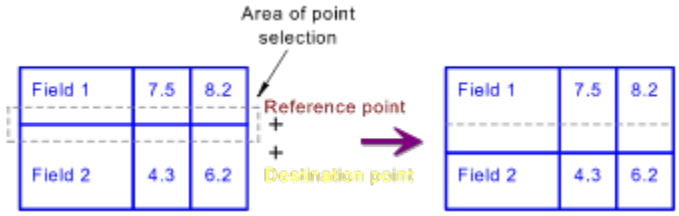
3. *Enter target point*
   The target point for the movement can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The possibilities which can result from moving individual definition points are varied. The consequences are equally various and can be unexpected. Many definition points have a close relationship to other definition points and can only be moved in certain ways. Under some circumstances, some definition points can not be moved because of object parameter settings. Be sparing in the use of moving individual definition points.

Moving several definition points at once to stretch or shorten objects is particularly interesting. To do this, all the definition points of an object are selected and then moved together (see graphic).



Moving definition points is also very useful when working with tables or similar grids built up from rectangles. To move the border between rectangles, it is quicker to move the definition points than to scale all the affected rectangles (see graphic).



The duplicate function does not work with this command, i.e. the objects themselves will be changed. If a copy is required, it must be done before carrying out the command.

Related commands:


Move Points>Perpendicular


Move Points>Parallel


Move Points>Relative


Move Single Point

Move Points>Perpendicular (Shape Menu) Level 2

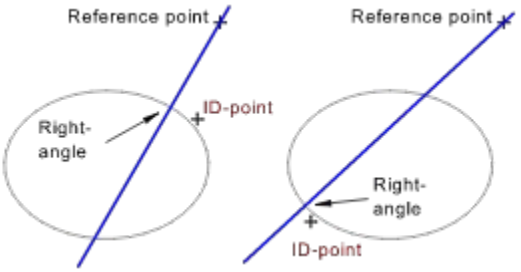
This command can be used to move several definition points at the same time. The movement takes place perpendicular to a specified reference object.

1. *Choose points*
   Choose the points that shall be moved. Click on points to choose them. For further information on point selection, see the chapter Point Selection.

2. *Identify reference object*
   Any part object in an existing object can be identified as a reference object. This may be an edge of a rectangle, the arc of an elliptic segment or a circular arc within a surface.
   To identify it, click on the (part) object with the mouse. If more than one possible (part) object lies close to the point clicked on, then a small dialog appears at the bottom of the screen from which the "right" object can be chosen.

   The identified object is referred to as the "reference object"

   If an ellipse part is identified, then the position at which the (part) object was identified (the ID point) affects the rest of the command. If a perpendicular is dropped on to an ellipse, then it can end at either side of the ellipse. The point closer to the ID point is used as the end of the perpendicular (see graphic).



3. *Enter reference point*
   The movement reference point can be entered using the mouse, by clicking anywhere in the drawing. A perpendicular is dropped from this point to the reference object.

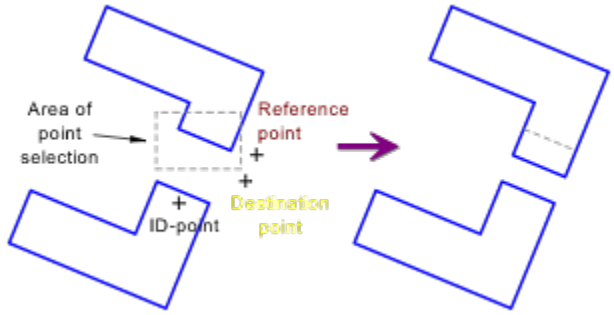
   The target point can be positioned on the straight line between the end of the perpendicular and the reference point. This line, the "reference line", is displayed. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter target point*
   The target point for the movement can be entered using the mouse, by clicking anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to work out the movement target point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, you can enter a length. This determines the size of the movement, in the direction of the reference straight line, beginning at the previously entered start point.

The possibilities which can result from moving individual definition points are varied. The consequences are equally various and can be unexpected. Many definition points have a close relationship to other definition points and can only be moved in certain ways. Under some circumstances, some definition points can not be moved because of object parameter settings. Be sparing in the use of moving individual

definition points.

Moving multiple definition points parallel to a reference object is useful, for example, if a sloping opening drawn in the plan view is to be enlarged or reduced (see graphic).



The duplicate function does not work with this command, i.e. the objects themselves will be changed. If a copy is required, it must be done before carrying out the command.

Related commands:

- Move Points>Standard
- Move Points>Parallel
- Move Points>Relative
- Move Single Point

Move Points>Parallel (Shape Menu) Level 2

This command can be used to move several definition points at the same time. The movement takes place parallel to a specified reference object.

1. *Choose points*
   Choose the points that shall be moved. Click on points to choose them. For further information on point selection, see the chapter Point Selection.

2. *Identify reference object*
   Any part object in an existing object can be identified as a reference object. This can be the edge of a rectangle or a circular arc within a surface. Ellipse (part)s cannot be identified.
   To identify it, click on the (part) object with the mouse. If more than one possible (part) object lies close to the point clicked on, then a small dialog appears at the bottom of the screen from which the "right" object can be chosen.

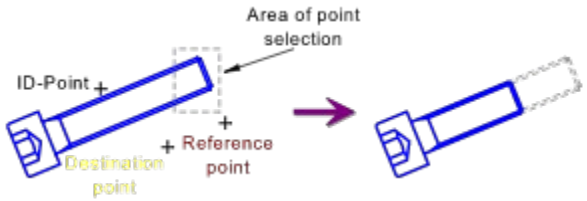
   The identified object is referred to as the "reference object"

3. *Enter reference point*
   The movement reference point can be entered using the mouse, by clicking anywhere in the drawing. If the previously identified reference object is a line, the reference point determines a line which runs parallel to the reference object. This is displayed and referred to as the "reference line". If the reference object is a circle or circle part then the reference point determines a circle concentric with the reference object. This is displayed, and referred to as the "Reference Circle". The target point can then be positioned on the reference circle or reference straight line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter target point*
   The target point for the movement can be entered using the mouse, by clicking anywhere in the drawing. A perpendicular is dropped from this position to the reference circle to work out the target point for the movement. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If the reference object is a line, a length can be entered. This gives the size of the movement in the direction of the reference line, beginning from a specified starting point. In all other cases an angle, measured from the reference circle's center, can be entered. The angle can be given relative to the reference point angle.

The possibilities which can result from moving individual definition points are varied. The consequences are equally various and can be unexpected. Many definition points have a close relationship to other definition points and can only be moved in certain ways. Under some circumstances, some definition points can not be moved because of object parameter settings. Be sparing in the use of moving individual definition points.

Moving definition points parallel to a reference object is particularly useful to stretch or shorten several sloping objects at once. To do this, all the definition points at one end of an object are selected and then moved at the same time (see graphic).



The duplicate function does not work with this command, i.e. the objects themselves will be changed. If a copy is required, it must be done before carrying out the command.

Related commands:

- Move Points>Standard
- Move Points>Perpendicular
- Move Points>Relative
- Move Single Point

Move Points>Relative (Shape Menu)

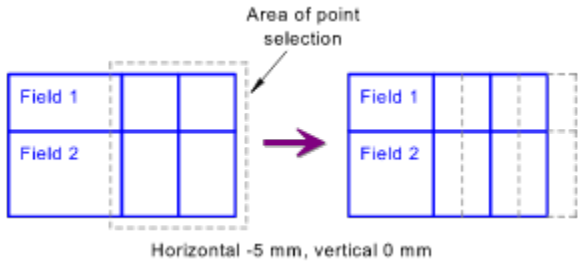
This command can be used to move several definition points at the same time. The movement is determined by entering a numeric value.

1. *Choose points*
   Choose the points that shall be moved. Click on points to choose them. For further information on point selection, see the chapter Point Selection.

2. *Enter horizontal and vertical movement*
   After choosing the command, a dialog appears, in which the size of the movement in horizontal and vertical directions can be entered.

The possibilities which can result from moving individual definition points are varied. The consequences are equally various and can be unexpected. Many definition points have a close relationship to other definition points and can only be moved in certain ways. Under some circumstances, some definition points can not be moved because of object parameter settings. Be sparing in the use of moving individual definition points.

Particularly interesting is the relative movement of multiple definition points, for example to move the end and corner points of several lines and rectangles at the same time by a given amount. To do this, all corner and end points are selected and then moved (see graphic).



The duplicate function does not work with this command, i.e. the objects themselves will be changed. If a copy is required, it must be done before carrying out the command.

Related commands:

Move Points>Standard

Move Points>Perpendicular

Move Points>Parallel

Move Single Point

Group>Create Group (Shape Menu)

This command can be used to combine objects and instances into a group, which will then be treated as a single item.

1. *Choose objects*
   The objects which are to be grouped must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

A group is then made from the chosen objects. To do this, a block containing these objects is generated. The block has a unique name, which is however not normally relevant when working with the group. The chosen objects are then replaced with an instance of the block. In principle, a group *is* a block.

One change occurs when creating a group. The objects which a group consists of are always constructed one after another - they make up a single unit. The order in which the objects in the group are built up compared to the other objects and instanced in the drawing may change.

Groups can be nested, that is a group may itself contains groups. Ungrouping is done step by step so that sub-groups remain intact.

TommySoftware® CAD/DRAW automatically maintains groups, but only when loading and saving drawings. If all pointers to a group are deleted, then that group is deleted at the latest when the drawing is saved. If it has to be used again, then it can be inserted with the command Block>Insert. Groups are stored in the "#G" folder of the library "* Internal Blocks". Their names however are not very helpful and so the selection of groups has to be done by displaying them.

Related command:

Group>Ungroup

Block>Create (Insertion Point)

Block>Create (Frame)

Group>Ungroup (Shape Menu)

This command can be used to break up groups. Their objects are then treated as individual objects again

1. *Choose objects*
   The groups to be broken up must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

All of the groups contained in the chosen groups are broken up; all other objects remain unchanged. Only the topmost layer in each group is broken up, so if a group is nested (i.e. it contains more groups) the sub-groups or instances of blocks are maintained.

TommySoftware® CAD/DRAW automatically maintains groups, but only when loading and saving drawings. If all pointers to a group are deleted, then that group is deleted at the latest when the drawing is saved. If it has to be used again, then it can be inserted with the command Block>Insert. Groups are stored in the "#G" folder of the library "* Internal Blocks". Their names however are not very helpful and so the selection of groups has to be done by displaying them.

Related commands:


Group>Create Group


Trim Object>Resolve Completely


Text>Resolve


Resolve Instances

Change Order>To Back (Shape Menu)

This command can be used to move instances and objects to the start of the object list, so that they are drawn first and therefore lie in the background.

The internal order of the chosen objects remains unchanged.

1. *Choose objects*
   The objects whose position in the list is to be changed must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

If the duplicate function is active when the last object is chosen, then a copy of the chosen objects is made and moved to the start of the list.

Related commands:


Change Order>Behind Object


Change Order>Before Object


Change Order>To Front

Change Order>Behind Object (Shape Menu) Level 2

This command can be used to move instances and objects to a position in front of another object, so that they are drawn before it and therefore lie in its background.

The internal order of the chosen objects remains unchanged.

1. *Choose objects*
   The objects whose position in the list is to be changed must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Identify object*
   The reference object is identified by clicking on its outline. If more than one line lies at the spot clicked on, a small dialog appears at the bottom of the screen, from which the correct line can be chosen.

If the duplicate function is active during identification of the reference object (for example, because the CTRL key was held down), a copy of the chosen objects is moved in the list.

Related commands:


Change Order>To Back


Change Order>In Front of Object


Change Order>To Front

Change Order>Before Object (Shape Menu) Level 2

This command can be used to move instances and objects to a position in the list after another object, so that they are drawn after it and therefore lie in its foreground.

The internal order of the chosen objects remains unchanged.

1. *Choose objects*
   The objects whose position in the list is to be changed must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Identify object*
   The reference object is identified by clicking on its outline. If more than one object outline lies at the spot clicked on, a small dialog appears at the bottom of the screen, from which the correct object can be chosen.

If the duplicate function is active during identification of the reference object (for example, because the CTRL key was held down), a copy of the chosen objects is moved in the list.

Related commands:


Change Order>To Back


Change Order>Behind Object


Change Order>To Front

Change Order>To Front (Shape Menu)

This command can be used to move objects and instances to the end of the object list so that they are drawn last and therefore lie at the front of the drawing.

The internal order of the chosen objects remains unchanged.

1. *Choose objects*
   The objects whose position in the list is to be changed must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

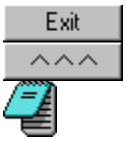
If the duplicate function is active during selection, a copy of the chosen objects is generated and pushed to the end of the list.

Related commands:

 Change Order>To Back

 Change Order>Behind Object

 Change Order>Before Object



Draw Menu

Line
- Standard
- On Straight Line
- Horizontal
- Vertical
- Mid-Perpendicular
- Perpendicular
- Parallel
- Parallel, Numerical
- Angle to Line
- Bisector of Angle
- Center Cross
- Polyline
- Equidistant
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Tangent
- Object - Point
- Object - Point, Horizontal
- Object - Point, Vertical
- Object - Point - End-Points
- Angle - Object - End-Points
- Object - Object

Polygon
- Triangle
- Quadrangle
- Parallelogram
- Rectangle
- Arbitrary

Polyeder
- Standard
- On Circle
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Circle
- Standard
- On circle
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- Tangential
- Line - Line - Line
- Radius - Point - Point

- Radius - Object - Point
- Radius - Object - Object

Circular Arc
- Standard
- On Circle
- Circumcircle
- Diameter
- Concentric to Circle
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- Radius - Point - Point
- Radius - Object - Point
- Radius - Object - Object

Ellipse
- Rectangular
- Rotated
- Distorted
- On Ellipse

Elliptical Arc
- Rectangular
- Distorted
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Arc Mode
- Arc
- Sector
- Segment
- Toggle Arc Mode
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Freehand Line

Spline

Curve

Surface

Hatching
- Objects
- Generated Surface
- Hatching Type List
- Edit Hatching Types
- Edit Line Sequences
- Set Fixed Point

Line>Standard (Draw Menu)

This command is used to draw a line.

1. *Enter start point*
   The start point can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter end point*
   The end point of the line can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Line>On Straight Line


Line>Horizontal


Line>Vertical


Line>Mid-Perpendicular


Line>Perpendicular


Line>Parallel


Line>Parallel, Numerical


Line>Angle to Line


Line>Bisector of Angle

Line>On Straight Line (Draw Menu) Level 2

This command creates a line which runs along an extension of an existing line or on a geometry line.

1. *Identify reference line*
   Any line in an existing object can be identified as a reference line, including a side of a rectangle or a circle segment leg.

   The line is identified by clicking on it with the mouse. If several possible lines lie close to the position clicked on, a small dialog appears near the bottom of the screen, from which the correct line can be chosen.

   After the line has been identified, a straight line which shows the extension of the identified line appears. This is the "Reference Line".

2. *Enter start point*
   The start point can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference line to determine the start point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter end point*
   The end point of the line can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference line to determine the end point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter a length. This gives the length of the line from a previously entered start point.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

 Line>Standard

 Line>Horizontal

 Line>Vertical

 Line>Mid-Perpendicular

 Line>Perpendicular

 Line>Parallel

 Line>Parallel, Numerical

 Line>Angle to Line

 Line>Bisector of Angle

Line>Horizontal (Draw Menu) Level 2

This command draws a horizontal line. "Horizontal" refers to the position of the line in relation to the page, not the view. If the drawing is displayed rotated, then the line will also be displayed rotated.

1. *Enter start point*
   The start point can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

   After entering the start point, a straight line running horizontally through the start point appears. This is referred to as the "Reference straight line".

2. *Enter end point*
   The end point of the line can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to determine the end point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter a length. This gives the length of the line from a previously entered start point.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

 Line>Standard

 Line>On Straight Line

 Line>Vertical

 Line>Mid-Perpendicular

 Line>Perpendicular

 Line>Parallel

 Line>Parallel, Numerical

 Line>Angle to Line

 Line>Bisector of Angle

Line>Vertical (Draw Menu) Level 2

This command draws a vertical line. "Vertical" refers to the position of the line in relation to the page, not the view. If the drawing is displayed rotated, then the line will also be displayed rotated.

1. *Enter start point*
   The start point can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

   After entering the start point, a straight line running vertically through the start point appears. This is referred to as the "Reference straight line".

2. *Enter end point*
   The end point of the line can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to determine the end point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter a length. This gives the length of the line from a previously entered start point.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

 Line>Standard

 Line>On Straight Line

 Line>Horizontal

 Line>Mid-Perpendicular

 Line>Perpendicular

 Line>Parallel

 Line>Parallel, Numerical

 Line>Angle to Line

 Line>Bisector of Angle

Line>Mid-Perpendicular (Draw Menu) Level 2

This command draws a line which runs along the mid-perpendicular of another line. This line is drawn so that it appears as a right angle in "normal" view. If working with a distorted view (e.g. an isometric view) then the visible angle will *not* be 90 degrees.

1. *Identify reference line*
   Any line in an existing object can be identified as a reference line, including a side of a rectangle or a circle segment leg. Geometry lines cannot be identified as they have no "middle".
   The line is identified by clicking on it with the mouse. If several possible lines lie close to the position clicked on, a small dialog appears near the bottom of the screen, from which the correct line can be chosen.

   After identification a straight line appears which runs along the mid-perpendicular of the identified line. This is referred to as the "Reference Straight Line".

2. *Enter start point*
   The start point can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this point to the reference straight line to determine the new line's start point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter end point*
   The end point of the line can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to determine the end point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter a length. This gives the length of the line from a previously entered start point.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

 Line>Standard

 Line>On Straight Line

 Line>Horizontal

 Line>Vertical

 Line>Perpendicular

 Line>Parallel

 Line>Parallel, Numerical

 Line>Angle to Line

Line>Bisector of Angle

Line>Perpendicular (Draw Menu)

This command draws a line which is perpendicular to another line, geometry line, circle, geometry circle or ellipse. This line is drawn so that it appears as a right angle in "normal" view. If working with a distorted view (e.g. an isometric view) then the visible angle will *not* be 90 degrees.

Only one end point of the line is specified, the other is already given as perpendicular base point. If a perpendicular is to be drawn where both end points are to be specified, use the command Line>Angle to Line instead.

1. *Identify reference object*
   Any part of an existing object can be used as a reference object. This can be a rectangle side, the arc of an ellipse segment or a circular arc within a surface.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The identified object is referred to as the "Reference Object". After identification a straight line showing the line of the perpendicular appears. This is referred to as the "Reference Straight Line"

2. *Enter reference point*
   The position of the perpendicular is determined by a reference point through which the reference straight line should run. This reference point can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If a circle is identified as the reference object, then you can enter an angle. This indicates the direction of the reference straight line.

3. *Enter start point*
   The start point of the line can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to determine the start point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter end point*
   The end point of the line can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to determine the end point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter a length. This determines the length of the line beginning from the previously entered start point.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Line>Standard


Line>On Straight Line


Line>Horizontal


Line>Vertical


Line>Mid-Perpendicular


Line>Parallel


Line>Parallel, Numerical


Line>Angle to Line


Line>Bisector of Angle

Line>Parallel (Draw Menu)

This command draws a line which runs parallel to another line or geometry line.

The distance between the lines is not determined numerically, but by entering the start point. To use a fixed distance, use the command Line>Parallel, Numerical.

1. *Identify reference line*
   Any line in an existing object can be identified as a reference line, including a side of a rectangle or a circle segment leg.
   The line is identified by clicking on it with the mouse. If several possible lines lie close to the position clicked on, a small dialog appears near the bottom of the screen, from which the correct line can be chosen.

   After identification a straight line appears which runs parallel to the identified line and moves with the mouse. This line is referred to as the "Reference Straight Line".

2. *Enter reference point*
   The reference point can be entered by clicking the mouse anywhere in the drawing. It determines the position of a straight line on which the final line will lie. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter start point*
   The start point can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter end point*
   The end point of the line can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to determine the end point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter a length. This gives the length of the line from a previously entered start point.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Line>Standard


Line>On Straight Line


Line>Horizontal


Line>Vertical


Line>Mid-Perpendicular


Line>Perpendicular


Line>Parallel, Fixed


Line>Angle to Line


Line>Bisector of Angle

Line>Parallel, Numerical (Draw Menu)

This command draws a line which runs parallel to another line or a geometry line.

The suffix "Numerical" shows that the distance between the two lines is a specific, fixed distance entered via a dialog. If the numeric value of the distance is not known, and the distance is to be specified implicitly by using the mouse to enter the start point, then the command Line>Parallel should be used instead.

1. *Enter distance*
   After choosing the command, a dialog appears into which the distance can be entered. This distance remains valid until either another command is chosen, or the command is restarted by choosing Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify reference line*
   Any line in an existing object can be identified as a reference line, including a side of a rectangle or a circle segment leg.
   The line is identified by clicking on it with the mouse. If several possible lines lie close to the position clicked on, a small dialog appears near the bottom of the screen, from which the correct line can be chosen.

   After identification a straight line which runs parallel to the identified line and moves when the mouse is moved close to the identified line. This line is referred to as the "Reference Straight Line".

3. *Enter reference point*
   The reference point can be entered by clicking the mouse anywhere in the drawing. It determines the position of a straight line on which the final line with lie. The reference straight line always snaps to the same side of the identified line as the crosshair. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter start point*
   The start point can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

5. *Enter end point*
   The end point of the line can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to determine the end point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter a length. This gives the length of the line from a previously entered start point.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

Line>Standard


Line>On Straight Line


Line>Horizontal


Line>Vertical


Line>Mid-Perpendicular


Line>Perpendicular


Line>Parallel


Line>Angle to Line


Line>Bisector of Angle

Line>Angle to Line (Draw Menu)

This command draws a line which runs at a specific angle to a line or geometry line.

1. *Enter angle*
   After choosing this command, a dialog appears into which the angle to be used can be entered. This angle remains valid until either a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify reference line*
   Any line in an existing object can be identified as a reference line, including a side of a rectangle or a circle segment leg.
   The line is identified by clicking on it with the mouse. If several possible lines lie close to the position clicked on, a small dialog appears near the bottom of the screen, from which the correct line can be chosen.

   After identification a straight line appears which runs at the previously entered angle to the identified line and moves with the mouse. This line is referred to as the "Reference Straight Line".

3. *Enter reference point*
   The reference point can be entered by clicking the mouse anywhere in the drawing. It determines the position of a straight line on which the final line will lie. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter start point*
   The start point can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

5. *Enter end point*
   The end point of the line can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to determine the end point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter a length. This gives the length of the line from a previously entered start point.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Line>Standard


Line>On Straight Line


Line>Horizontal


Line>Vertical


Line>Mid-Perpendicular


Line>Perpendicular


Line>Parallel


Line>Parallel, Numerical


Line>Bisector of Angle

Line>Bisector of Angle (Draw Menu)

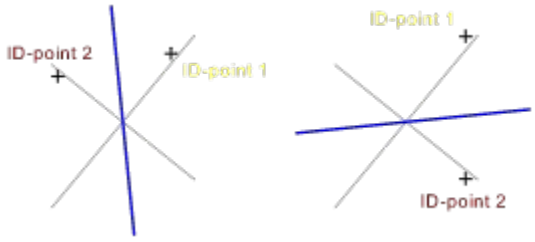
This command draws a line which runs along the bisector of the angle between two lines or geometry lines.

1. *Identify reference line 1*
   Any line in an existing object can be identified as a reference line. This can also be a side of a rectangle or a leg of a circular segment.
   The line is identified by clicking on it with the mouse. If several possible lines lie close to the position clicked on, a small dialog appears near the bottom of the screen, from which the correct line can be chosen.

   The position at which the line was identified (the "ID Point") is important for the rest of the procedure. Together with the position at which the second line is identified, it determines which of the two possible angle bisectors should be used (see below).

2. *Identify reference line 2*
   Any line in an existing object can be identified as the second reference line. This can also be a side of a rectangle or a leg of a circular segment.
   The line is identified by clicking on it with the mouse. If several possible lines lie close to the position clicked on, a small dialog appears near the bottom of the screen, from which the correct line can be chosen.

   The position at which the line was identified (the "ID Point") is important for the rest of the procedure. Together with the position at which the first line is identified, it determines which of the two possible angle bisectors should be used (see graphic).



   After identification of the second line a straight line which runs through the ID point determining the angle bisectors appears. This line is referred to as the "Reference Straight Line".

3. *Enter start point*
   The start point can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this point to the reference straight line to determine the new line's start point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter end point*
   The end point of the line can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to determine the end point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter a length. This gives the length of the line from a previously entered start point.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Line>Standard


Line>On Straight Line


Line>Horizontal


Line>Vertical


Line>Mid-Perpendicular


Line>Perpendicular


Line>Parallel


Line>Parallel, Numerical


Line>Angle to Line

Line>Center Cross (Draw Menu)

This command draws two lines, one horizontal and one vertical. Their mid-points cross at the center of an identified circle or ellipse.

This command is particularly useful to draw center crosses for drilled holes and other construction elements.

1. *Identify reference circle / ellipse*
   Any circle or ellipse which exists independently or as part of another object can be identified as a reference object. This can also be the arc of an ellipse segment or a circular arc within a surface. Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

2. *Enter point*
   The length of both lines is determined by a point which lies on the outline of a circle tangent to both lines. This point can be entered by clicking with the mouse anywhere within the drawing. A perpendicular is dropped from this position to the reference straight line to determine the end point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter a specific radius. This gives the radius of the circle tangent to both lines. This is always a circle, even if an ellipse was identified beforehand.

The two lines are assigned to the current layer. You also see a reference to the currently active pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

Line>Horizontal

Line>Vertical

Line>Polyline (Draw Menu)

This command draws a polyline. A polyline is an open series of connected lines.

1. *Enter start point*
   The start point of a polyline can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point*
   After entering the start point, several points can be entered, one after another. The polyline begins at the start point and runs through all the subsequently-entered points. These points can be entered by clicking the mouse at any point in the drawing for each point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter the length and/or angle of the next section relative to the previous section. This is an easy way, for example, to enter bends with a given angle.

   Polyline input is ended by clicking the right mouse button. Unlike most other commands, this does not lead to the previous entry being ignored; instead the polyline is inserted into the drawing. If required, this can be undone using the Undo command.

A polyline can consist of up to 1000 sections. If this limit is reached during point entry, no further points can be entered. A polyline is stored internally as a curve. It can be manipulated using the same commands as can be used on curves.

The curve created is assigned to the current layer. It also contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel polylines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

All point entries can be undone step by step using Previous Point (ESC) or by pressing the ESC key. This enables incorrect entries to easily be corrected without having to enter the entire polyline again.

Related commands:


Curve


Surface


Spline


Polygon>Arbitrary


Freehand Line

Line>Equidistant (Draw Menu)

This command generates a line equidistant to a specified object. An equidistant line is a line which is the same distance from all points of the reference object. In the simplest case this is a line parallel to another line or a circle concentric with another circle.

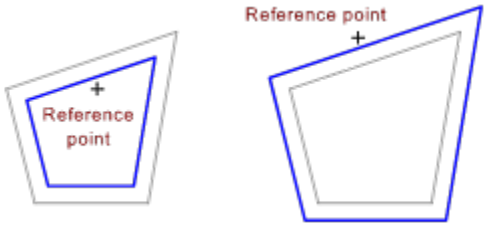
1. *Enter distance*
   After choosing the command, a dialog appears into which the distance can be entered. This distance remains valid until either another command is chosen, or the command is restarted by choosing Change Parameters (+ESC) or the key combination SHIFT+ESC.

   The distance must be practical. For example, you cannot draw an equidistant larger than 10 mm within a circle with a radius of 10 mm.

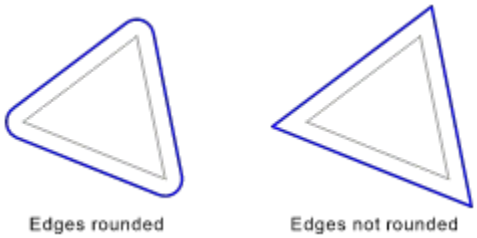
2. *Identify object*
   Any object consisting of "normal" elements can be identified as a reference object. These are lines, circles and circle parts, ellipses and elliptical arcs as well as all objects consisting of these elements. The objects are always treated as a whole. Some elements, for example text, dimensions and instances, cannot be used.
   An object is identified by clicking on it. If more than one object lies at the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The identified object is referred to as the "Reference Object".

3. *Enter reference point*
   After identifying the reference object a point is entered which determines on which side of the reference object the equidistant should be drawn (see graphic).



4. *Round corners?*
   When drawing an equidistant for an object made up of several sections, after entering the reference point a window appears to ask whether or not the corners should be rounded or not (see graphic).



   Because of the definition of equidistants (the same distance from all points), all corners must

theoretically be rounded. In many cases this is not required, and therefore this procedure is provided. There are also situations in which only one of the two options ("rounded" or "not rounded") functions. This often the case when there are pointed transitions between lines and circular arcs.

Equidistants are stored internally in different forms, according to the reference object's type. Lines create lines, circles create circles. All other object types create curves (open outlines) or surfaces (closed outlines).

The object is allocated to the current layer. It also contains a reference to the current pen.

If a line equidistant to a surface containing Bézier curves is created, there may be (extreme) inaccuracies. During the calculation of the equidistant line, the Bézier curve is treated as a polyline with three sections (Start point -Angel point 1 - Angel Point 2 - End point). If the Bézier curve is relatively short and flat, this simplification is not usually very effective.

This command does not work with text in either TommySoftware® CAD/DRAW's own format or TrueType-format. To draw a line equidistant to text, it must first be converted to curves or surfaces using the command Text>Resolve. An equidistant line can then be created (see Graphic)

The problem with Bézier curves described above doe not arise with TrueType fonts as the Bézier curves in TrueType fonts are relatively short and flat. However, rounded corners should always be used in order to make the result look smoother.

Related commands:

- Line>Parallel
- Line>Parallel, Numerical
- Circle>Concentric
- Circular Arc>Concentric to Circle
- Circular Arc>Concentric to Circle Part

Line>Zigzag Line (Draw Menu)

This command draws a zigzag line. The height and width of the zags varies according to the line width which the zigzag line is displayed.

1. *Enter start point*
   The start point of the zigzag line can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter end point*
   The end point of the zigzag line can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

Zigzag lines are usually treated by the program like simple lines. This is especially the case for commands from the Trimming Menu.

The zigzag line is assigned to the current layer. It also contains a reference to the current pen.

When choosing the Change Parameters (+ESC) command, the Zigzag dialog appears where the new distance between zags can be entered. This distance between zags is used for all subsequently drawn zigzag lines.

The distance between zags in existing zigzag lines is not changed when the current zag distance is changed. To alter the distance between zags in an existing zigzag line, use the command Edit Properties .

Related command:

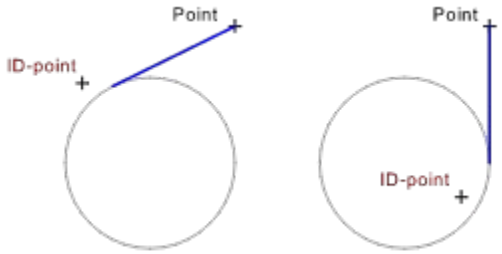
Line>Standard

Tangent>Object - Point (Draw Menu)

This command draws a line which begins at a given point and ends tangential to a circle or ellipse.

1. *Identify reference circle / ellipse*
   Any circle or ellipse which exists independently or as part of another object can be identified as a reference object. This can also be the arc of an ellipse segment or a circular arc within a surface. Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The position at which the object is identified (the "ID point") is important for the rest of the process. If a tangent is placed on a curve or ellipse it can go on either side. The tangent point used as the end of the line is the one nearest to the ID point (see graphic).



2. *Enter point*
   The start point of the tangent can be entered by clicking the mouse anywhere in the drawing. A tangent is drawn from this point to the reference object. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

   This command does not work if the point is inside the object or on its outline; in this case, input is not allowed.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

Tangent>Object - Point, Horizontal

Tangent>Object - Point, Vertical

Tangent>Object - Point - End-Point

Tangent>Angle - Object - End-Point

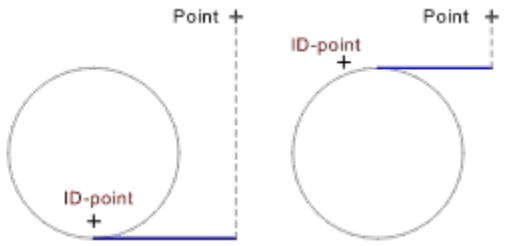
Tangent>Object - Object

Tangent>Object - Point, Horizontal (Draw Menu)

This command draws a horizontal line which ends tangential to a circle or ellipse. "Horizontal" refers to the position relative to the page, not the view. If the drawing is shown rotated on screen, then the line will also be shown rotated.

1. *Identify reference circle / ellipse*
   Any circle or ellipse which exists independently or as part of another object can be identified as a reference object. This can also be the arc of an ellipse segment or a circular arc within a surface. Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The position at which the object was identified (the "ID point") affects the rest of the command. If a horizontal tangent is placed on a circle or an ellipse, this can occur at the top or the bottom. The line will end at the tangent point which is closer to the ID point (see graphic).



   After identifying the object, a straight line appears which runs horizontally and is tangent to the identified object. This line is referred to as the "Reference Straight Line".

2. *Enter point*
   The end point of the tangent can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to determine the tangent's end point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter a length. This gives the length of the line beginning from the tangent point.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

Tangent>Object - Point

Tangent>Object - Point, Vertical

Tangent>Object - Point - End-Point

Tangent>Angle - Object - End-Point

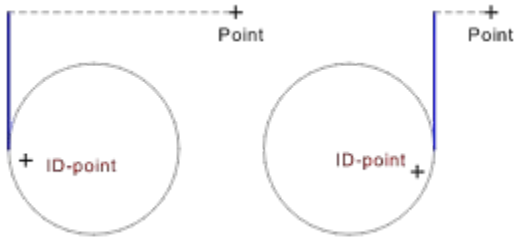
Tangent>Object - Object

Tangent>Object - Point, Vertical (Draw Menu)

This command draws a vertical line which ends tangential to a circle or ellipse. "Vertical" refers to the position relative to the page, not the view. If the drawing is shown rotated on screen, then the line will also be shown rotated.

1. *Identify reference circle / ellipse*
   Any circle or ellipse which exists independently or as part of another object can be identified as a reference object. This can also be the arc of an ellipse segment or a circular arc within a surface. Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The position at which the object was identified (the "ID point") affects the rest of the command. If a vertical tangent is placed on a circle or an ellipse, this can occur at the left or the right. The line will end at the tangent point which is closer to the ID point (see graphic).



   After identifying the object, a straight line appears which runs vertically and is tangent to the identified object.

2. *Enter point*
   The end point of the tangent can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to determine the tangent's end point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter a length. This gives the length of the line beginning from the tangent point.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

Tangent>Object - Point

Tangent>Object - Point, Horizontal

Tangent>Object - Point - End-Point

Tangent>Angle - Object - End-Point

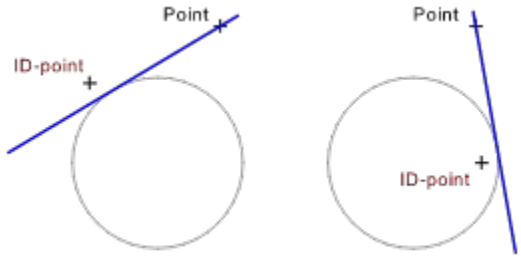
Tangent>Object - Object

Tangent>Object - Point - End-Point (Draw Menu)

This command draws a line on a straight line running through a given point and being tangent to a circle or ellipse.

1. *Identify reference circle / ellipse*
   Any circle or ellipse which exists independently or as part of another object can be identified as a reference object. This can also be the arc of an ellipse segment or a circular arc within a surface. Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The position at which the object is identified (the "ID point") affects the rest of the command. If a vertical tangent is placed on a circle or an ellipse, this can occur at the left or the right. The line will end at the tangent point which is closer to the ID point (see graphic).



2. *Enter point*
   The point through which the tangent should run, can be entered by clicking the mouse anywhere in the drawing. A tangent is drawn from this point to the identified object. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

   After entering the point, a straight line appears which runs through the point and is tangent to the identified object. This line is referred to as the "Reference Straight Line".

3. *Enter start point*
   The start point can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this point to the reference straight line to determine the new line's start point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter end point*
   The end point of the line can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to determine the end point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter a length. This gives the length of the line from a previously entered start point.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

- Tangent>Object - Point
- Tangent>Object - Point, Horizontal
- Tangent>Object - Point, Vertical
- Tangent>Angle - Object - End-Point
- Tangent>Object - Object

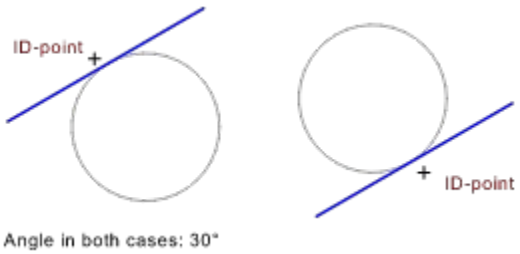
Tangent>Angle - Object - End-Point (Draw Menu)

This command draws a line along a straight line which is tangent to a given circle or ellipse at a particular angle.

1. *Enter angle*
   After choosing this command, a dialog appears into which the angle to be used can be entered. This angle remains valid until either a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify reference circle / ellipse*
   Any circle or ellipse which exists independently or as part of another object can be identified as a reference object. This can also be the arc of an ellipse segment or a circular arc within a surface. Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The position at which the object is identified (the "ID point") affects the rest of the command. If a tangent is placed at a particular angle to a circle or an ellipse, it can be placed on either side of the ellipse/circle. The line will end at the tangent point which is closer to the ID point (see graphic).



   After identification of the object, a straight line appears which runs at a specified angle and is tangent to the identified object. This line is referred to as the "Reference Straight Line".

3. *Enter start point*
   The start point can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this point to the reference straight line to determine the new line's start point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter end point*
   The end point of the line can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this position to the reference straight line to determine the end point of the new line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter a length. This gives the length of the line from a previously entered start point.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

- Tangent>Object - Point
- Tangent>Object - Point, Horizontal
- Tangent>Object - Point, Vertical
- Tangent>Object - Point - End-Point
- Tangent>Object - Object

Tangent>Object - Object (Draw Menu)

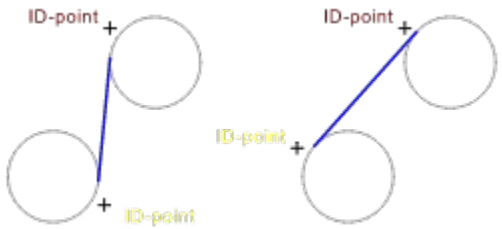
This command draws a line which ends tangential to two circles or ellipses.

1. *Identify reference circle / ellipse 1*
   Any circle or ellipse which exists independently or as part of another object can be identified as the first reference object. This can also be the arc of an ellipse segment or a circular arc within a surface. Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The position at which the object is identified (the "ID point") affects the rest of the command. There are up to four possibilities when drawing a tangent between two circles. Together with a second ID point, this ID point determines which of the four possible tangents are drawn (see below).

2. *Identify reference circle / ellipse 2*
   Any circle or ellipse which exists independently or as part of another object can be identified as the second reference object. This can also be the arc of an ellipse segment or a circular arc within a surface.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The position at which the object is identified (the "ID point") affects the rest of the command. There are up to four possibilities when drawing a tangent between two circles. Together with the first ID point, this ID point determines which of the four possible tangents are drawn (see graphic).



If one circle or ellipse lies wholly inside another, it is not possible to work out a line tangent to both circles, and a corresponding error message appears.

The line is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

Tangent>Object - Point

Tangent>Object - Point, Horizontal


Tangent>Object - Point, Vertical


Tangent>Object - Point - End-Point


Tangent>Angle - Object - End-Point

Polygon>Triangle (Draw Menu)

This command draws an arbitrary surface with three corners (a triangle).

1. *Enter corner point 1*
   The first corner point of the triangle can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter corner point 2*
   The second corner point of the triangle can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter corner point 3*
   The third corner point of the triangle can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The surface is assigned to the current layer. It also contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel triangles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Polygon>Quadrangle


Polygon>Parallelogram


Polygon>Rectangle


Polygon>Arbitrary


Surface

Polygon>Quadrangle (Draw Menu)

This command draws an arbitrary surface with four corners.

1. *Enter corner point 1*
   The first corner point of the quadrangle can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter corner point 2*
   The second corner point of the quadrangle can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter corner point 3*
   The third corner point of the quadrangle can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter corner point 4*
   The fourth corner point of the quadrangle can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The corner points can be positioned so that the sides of the quadrangle overlap. The surface is still handled correctly by the program.

The surface is assigned to the current layer. It also contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel quadrangles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Polygon>Triangle


Polygon>Parallelogram


Polygon>Rectangle


Polygon>Arbitrary


Surface

Polygon>Parallelogram (Draw Menu)

This command draws a parallelogram. A parallelogram is a quadrangle in which the opposite sides are parallel. It can also be seen as a distorted rectangle.

The parallelogram is particularly useful when working in isometric and dimetric views. If a rectangle is to be drawn in either of these view, then it *must* be done with this command and not with Polygon>Rectangle because the result on screen is not right-angled.

1. *Enter corner point 1*
   The first corner point of the parallelogram can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter corner point 2*
   The second corner point of the parallelogram can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter corner point 3*
   The third corner point of the parallelogram can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The fourth corner point is calculated so that the opposite sides are parallel.

The surface is assigned to the current layer. It also contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel parallelograms will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Polygon>Triangle


Polygon>Quadrangle


Polygon>Rectangle


Polygon>Arbitrary


Surface

Polygon>Rectangle (Draw Menu)

This command draws a rectangle. A rectangle is a quadrangle where each side is at right angles to the adjacent side, and is therefore a special type of parallelogram. This command always draws a rectangle whose sides are horizontal and vertical. Horizontal and vertical refers to the position relative to the page, not the view. If the drawing is rotated, the sides are displayed rotated as well.

If a rectangle is to be drawn when working in isometric or dimetric view, this *must* be done using the command Polygon>Parallelogram and not this command, as the result on the page is *not* right-angled.

1. *Enter corner point 1*
   The first corner point of the rectangle can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter corner point 2*
   The first corner point of the rectangle can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). By using relative coordinates, a rectangle of a specific size can easily be drawn.

The two other corners are positioned automatically so that the adjacent sides form right angles.

The surface is assigned to the current layer. It also contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel rectangles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Polygon>Triangle


Polygon>Quadrangle


Polygon>Parallelogram


Polygon>Arbitrary


Surface

Polygon>Arbitrary (Draw Menu)

This command draws an arbitrary polygon. A polygon is a closed sequence of connected lines.

1. *Enter start point*
   The start point of the polygon can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point*
   After entering the start point, more points can be entered. The polygon begins at the start point, runs through all the subsequently-entered points and ends at the start point. These points can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, you can specify the length and/or angle of the next section relative to the previous one. This makes it easy, for example, to enter sections at a specific angle.

   Polygon entry is ended by clicking the right mouse button. In contrast to most other commands this does not lead to the last entry being ignored but instead inserts the polygon. This can be undone using the Undo command.

A polygon can have up to 1000 segments. If this limit is reached during entry, then further point entry is not allowed. A polygon is stored internally as a surface. It can be manipulated using the same commands as surfaces.

The surface is assigned to the current layer. It also contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel polygons will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

All point entries can be undone step by step using Previous Point (ESC) or by pressing the ESC key. This makes it easy to correct mistakes without having to re-enter the whole polygon.

Related commands:


Polygon>Triangle


Polygon>Quadrangle


Polygon>Parallelogram


Polygon>Rectangle


Surface


Curve


Line>Polyline

Polyeder>Standard (Draw Menu)

This command draws a polyeder. A polyeder is a regular polygon whose corner points all lie on an imaginary circle. A polyeder is, as a result, a closed sequence of joined lines.

A polyeder is specified by entering the center and radius of the imaginary circle on which its corners lie.

1. *Enter number of sides*
   After choosing the command, a dialog appears in which the desired number of sides can be entered. Values between 3 and 99 inclusive are allowed. This number of sides remains valid until either a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Enter center point*
   The center point of the imaginary circle can be entered by clicking the mouse button anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter radius*
   The radius of the imaginary circle can be entered by clicking the mouse button at any point in the drawing through which the circle should run. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter a specific radius. This determines the radius of the circle.

4. *Enter angle*
   After specifying the imaginary circle, the polyeder can be rotated. To do this, the angle (measured from the center of the circle) at which the first corner of the polyeder should lie. The angle can be entered by clicking the mouse at any point in the drawing. The position of this point relative to the center determines the angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This determines the angle at which the first corner of the polyeder should lie.

A polyeder is stored internally as a surface. It can be manipulated with the same commands as a surface.

The surface is assigned to the current layer. It also contains a reference to the current pen.

Related commands:

 Polyeder>On Circle

 Polyeder>Circumcircle

 Polyeder>Diameter

 Polyeder>Side Length - Center

 Polyeder>Inner Radius - Center

 Surface

Polyeder>On Circle (Draw Menu)

This command draws a polyeder. A polyeder is a regular polygon whose corner points all lie on an imaginary circle. A polyeder is, as a result, a closed sequence of joined lines.

The polyeder is determined by identifying a reference circle which determines the imaginary circle on which all of the corner points lie.

1. *Enter number of sides*
   After choosing the command, a dialog appears in which the desired number of sides can be entered. Values between 3 and 99 inclusive are allowed. This number of sides remains valid until either a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify reference circle*
   Any circle which exists independently or as part of another object can be identified as the reference object. This can also be the arc of a circular segment or a circular arc within a surface.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

3. *Enter angle*
   After specifying the imaginary circle, the polyeder can be rotated. To do this, the angle (measured from the center of the circle) at which the first corner of the polyeder should lie. The angle can be entered by clicking the mouse at any point in the drawing. The position of this point relative to the center determines the angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This determines the angle at which the first corner of the polyeder should lie.

A polyeder is stored internally as a surface. It can be manipulated with the same commands as a surface.

The surface is assigned to the current layer. It also contains a reference to the current pen.

Related commands:

- Polyeder>Standard
- Polyeder>Circumcircle
- Polyeder>Diameter
- Polyeder>Side Length - Center
- Polyeder>Inner Radius - Center
- Surface

Polyeder>Circumcircle (Draw Menu)

This command draws a polyeder. A polyeder is a regular polygon whose corner points all lie on an imaginary circle. A polyeder is, as a result, a closed sequence of joined lines.

The polyeder is calculated by entering three points through which the imaginary circle, on which all the corners lie, runs.

1. *Enter number of sides*
   After choosing the command, a dialog appears in which the desired number of sides can be entered. Values between 3 and 99 inclusive are allowed. This number of sides remains valid until either a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Enter point 1 on circle*
   The first point through which the imaginary circle should run can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter point 2 on circle*
   The second point through which the imaginary circle should run can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter point 3 on circle*
   The third point through which the imaginary circle should run can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

5. *Enter angle*
   After specifying the imaginary circle, the polyeder can be rotated. To do this, the angle (measured from the center of the circle) at which the first corner of the polyeder should lie. The angle can be entered by clicking the mouse at any point in the drawing. The position of this point relative to the center determines the angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This determines the angle at which the first corner of the polyeder should lie.

A polyeder is stored internally as a surface. It can be manipulated with the same commands as a surface.

The surface is assigned to the current layer. It also contains a reference to the current pen.

Related commands:

 Polyeder>Standard

 Polyeder>On Circle

 Polyeder>Diameter

 Polyeder>Side Length - Center

Polyeder>Inner Radius - Center

Surface

Polyeder>Diameter (Draw Menu)

This command draws a polyeder. A polyeder is a regular polygon whose corner points all lie on an imaginary circle. A polyeder is, as a result, a closed sequence of joined lines.

The polyeder is calculated from two points which indicate the endpoints of the diameter of the imaginary circle on which the polyeder's corners lie.

1. *Enter number of sides*
   After choosing the command, a dialog appears in which the desired number of sides can be entered. Values between 3 and 99 inclusive are allowed. This number of sides remains valid until either a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Enter point 1 on circle*
   The first point through which the imaginary circle should run can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter point 2 on circle*
   The second point through which the imaginary circle should run can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter angle*
   After specifying the imaginary circle, the polyeder can be rotated. To do this, the angle (measured from the center of the circle) at which the first corner of the polyeder should lie. The angle can be entered by clicking the mouse at any point in the drawing. The position of this point relative to the center determines the angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This determines the angle at which the first corner of the polyeder should lie.

A polyeder is stored internally as a surface. It can be manipulated with the same commands as a surface.

The surface is assigned to the current layer. It also contains a reference to the current pen.

Related commands:

Polyeder>Standard

Polyeder>On Circle

Polyeder>Circumcircle

Polyeder>Side Length - Center

Polyeder>Inner Radius - Center

Surface

Polyeder>Side Length - Center (Draw Menu)

This command draws a polyeder. A polyeder is a regular polygon whose corner points all lie on an imaginary circle. A polyeder is, as a result, a closed sequence of joined lines.

The polyeder is calculated by entering the desired side length and the center of the imaginary circle on which all the corners lie.

1. *Enter side length and number of sides*
   After choosing the command, a dialog appears in which the desired number of sides and their length can be entered. Values between 3 and 99 inclusive are allowed. This number of sides remains valid until either a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Enter center point*
   The center point of the imaginary circle can be entered by clicking the mouse button anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter angle*
   After specifying the imaginary circle, the polyeder can be rotated. To do this, the angle (measured from the center of the circle) at which the first corner of the polyeder should lie. The angle can be entered by clicking the mouse at any point in the drawing. The position of this point relative to the center determines the angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This determines the angle at which the first corner of the polyeder should lie.

A polyeder is stored internally as a surface. It can be manipulated with the same commands as a surface.

The surface is assigned to the current layer. It also contains a reference to the current pen.

Related commands:

 Polyeder>Standard

 Polyeder>On Circle

 Polyeder>Circumcircle

 Polyeder>Diameter

 Polyeder>Inner Radius - Center

 Surface

Polyeder>Inner Radius - Center (Draw Menu) 

This command draws a polyeder. A polyeder is a regular polygon whose corner points all lie on an imaginary circle. A polyeder is, as a result, a closed sequence of joined lines.

The polyeder is calculated by entering the desired inner radius and center of the imaginary circle on which all the corners lie. The inner radius gives the radius of the largest circle which lies within the polyeder and touches its side.

The inner radius is exactly half of the usual width for six-sided screw heads. If the plan view of a six-sided screw with an 8 mm width is to be drawn, 4 mm should be entered as the inner radius.

1. *Enter number of sides and inner radius*
   After choosing the command a dialog appears in which the number of sides and inner radius for the polyeder can be entered. A number of sides between 3 and 99 can be entered. The number of sides and the inner radius remain valid until a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Enter center point*
   The center point of the imaginary circle can be entered by clicking the mouse button anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter angle*
   After specifying the imaginary circle, the polyeder can be rotated. To do this, the angle (measured from the center of the circle) at which the first corner of the polyeder should lie. The angle can be entered by clicking the mouse at any point in the drawing. The position of this point relative to the center determines the angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This determines the angle at which the first corner of the polyeder should lie.

A polyeder is stored internally as a surface. It can be manipulated with the same commands as a surface.

The surface is assigned to the current layer. It also contains a reference to the current pen.

Related commands:

 Polyeder>Standard

 Polyeder>On Circle

 Polyeder>Circumcircle

 Polyeder>Diameter

 Polyeder>Side Length - Center

 Surface

Circle>Standard (Draw Menu)

This command draws a circle. The circle is determined by entering its center and radius.

If a circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter center point*
   The center point of the circle can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter radius*
   The radius of the circle can be entered by clicking the mouse anywhere in the drawing at a point through which the circle should run. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter a specific radius. This gives the radius of the circle.

The circle is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

 Circle>On Circle

 Circle>Circumcircle

 Circle>Diameter

 Circle>Incircle

 Circle>Concentric

 Circle>Tangential

 Circle>Line - Line - Line

 Circle>Radius - Point - Point

 Circle>Radius - Object - Point

 Circle>Radius - Object - Object

Circle>On Circle (Draw Menu)

This command draws a circle based on another circle or a circle part.

This command is especially useful if a geometry circle is to be transferred into the drawing. It can also be used to draw a circle over an existing circle part.

If a circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Identify reference circle*
   Any circle which exists independently or as part of another object can be identified as the reference object. This can also be the arc of a circular segment or a circular arc within a surface.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

The circle is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

- Circle>Standard
- Circle>Circumcircle
- Circle>Diameter
- Circle>Incircle
- Circle>Concentric
- Circle>Tangential
- Circle>Line - Line - Line
- Circle>Radius - Point - Point
- Circle>Radius - Object - Point
- Circle>Radius - Object - Object

Circle>Circumcircle (Draw Menu)

This command draws a circle. The circle is calculated by entering three points through which it should run. This is the same as calculating the circle surrounding a triangle.

If a circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter point 1 on circle*
   The first point through which the circle should run can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point 2 on circle*
   The second point through which the circle should run can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter point 3 on circle*
   The third point through which the circle should run can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The circle is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

 Circle>Standard

 Circle>On Circle

 Circle>Diameter

 Circle>Incircle

 Circle>Concentric

 Circle>Tangential

 Circle>Line - Line - Line

 Circle>Radius - Point - Point

 Circle>Radius - Object - Point

 Circle>Radius - Object - Object

Circle>Diameter (Draw Menu)

This command draws a circle. The circle is calculated by entering two points, one at either end of its diameter.

If a circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter point 1 on circle*
   The first point through which the circle should run can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point 2 on circle*
   The second point through which the circle should run can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The circle is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

- Circle>Standard
- Circle>On Circle
- Circle>Circumcircle
- Circle>Incircle
- Circle>Concentric
- Circle>Tangential
- Circle>Line - Line - Line
- Circle>Radius - Point - Point
- Circle>Radius - Object - Point
- Circle>Radius - Object - Object

Circle>Incircle (Draw Menu)

This command draws a circle. The circle is calculated from the positions of the corner points of a triangle which it should lie within.

If the sides of the triangle are known, then the command Circle>Line - Line - Line can be used instead.

If a circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter corner point 1*
   The first corner point of the triangle can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter corner point 2*
   The second corner point of the triangle can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter corner point 3*
   The first corner point of the triangle can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The circle is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Circle>Standard


Circle>On Circle


Circle>Circumcircle


Circle>Diameter


Circle>Concentric


Circle>Tangential


Circle>Line - Line - Line


Circle>Radius - Point - Point


Circle>Radius - Object - Point


Circle>Radius - Object - Object

Circle>Concentric (Draw Menu)

This command draws a circle concentric with another circle or circle part.

If a circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Identify reference circle*
   Any circle which exists independently or as part of another object can be identified as the reference object. This can also be the arc of a circular segment or a circular arc within a surface.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

2. *Enter radius*
   The radius of the circle can be entered by clicking the mouse anywhere in the drawing at a point through which the circle should run. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter a specific radius. This gives the radius of the circle. The radius can be given relative to the radius of the reference circle.

The circle is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

 Circle>Standard

 Circle>On Circle

 Circle>Circumcircle

 Circle>Diameter

 Circle>Incircle

 Circle>Tangential

 Circle>Line - Line - Line

 Circle>Radius - Point - Point

 Circle>Radius - Object - Point

 Circle>Radius - Object - Object

Circle>Tangential (Draw Menu)

This command draws a circle which has a given center and is tangent to another line (or geometry line), circle (or geometry circle) or ellipse.

If a circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Identify reference object*
   Any part of an existing object can be used as a reference object. This can be a rectangle side, the arc of an ellipse segment or a circular arc within a surface.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The identified object is referred to as the "Reference Object". After identification, a straight line appears on which the circle's center point will lie. This is referred to as the "Reference Straight Line"

2. *Enter reference point*
   The position of the circle and the tangent opines of the objects can be entered by clicking the mouse at any point within the drawing. A perpendicular is dropped from this point to the reference object to calculate the tangent points.

   If a circle is identified as the reference object, the position of the reference point determines on which side of the reference circle the tangent point should lie. The intersection of the reference straight line with the reference circle nearer to the reference point is used. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If a circle is identified as the reference object, then you can enter an angle. This indicates the direction of the reference straight line.

3. *Enter center point*
   The center point of the circle can be entered by clicking the mouse anywhere in the drawing. A perpendicular is dropped from this point to the reference straight line to work out the center of the circle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter a specific radius. This gives the radius of the circle.

The circle is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Circle>Standard


Circle>On Circle


Circle>Circumcircle


Circle>Diameter


Circle>Incircle


Circle>Concentric


Circle>Line - Line - Line


Circle>Radius - Point - Point


Circle>Radius - Object - Point


Circle>Radius - Object - Object

Circle>Line - Line - Line (Draw Menu)

This command draws a circle which is tangent to three lines or geometry lines.

If a circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Ellipse>Distorted because the result on the page is not circular but elliptical.

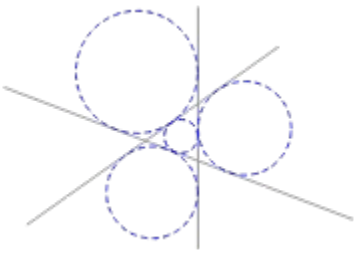
1. *Identify reference line 1*
   Any line in an existing object can be identified as the first reference line. This can also be a side of a rectangle or a leg of a circular segment.
   The line is identified by clicking on it with the mouse. If several possible lines lie close to the position clicked on, a small dialog appears near the bottom of the screen, from which the correct line can be chosen.

   The position at which the line is identified (the "ID point") does not affect the rest of the command. Because there are four possible results, choosing them based on the position of the ID points would be too complicated.

2. *Identify reference line 2*
   Any line in an existing object can be identified as the second reference line. This can also be a side of a rectangle or a leg of a circular segment.
   The line is identified by clicking on it with the mouse. If several possible lines lie close to the position clicked on, a small dialog appears near the bottom of the screen, from which the correct line can be chosen.

3. *Identify reference line 3*
   Any line in an existing object can be identified as the third reference line. This can also be a side of a rectangle or a leg of a circular segment.
   The line is identified by clicking on it with the mouse. If several possible lines lie close to the position clicked on, a small dialog appears near the bottom of the screen, from which the correct line can be chosen.

4. *Select object*
   After identifying all three reference lines, all the possible circles are calculated (the graphic shows one possible set of results).



   The 'right' circle can then be chosen with the help of the small dialog which appears at the bottom of the screen. Only one circle can be chosen at a time.

If all three reference lines are parallel to one another, then a circle can not be worked out and an error message appears. However the calculation is possible if only two of the reference lines are parallel.

The circle is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Circle>Standard


Circle>On Circle


Circle>Circumcircle


Circle>Diameter


Circle>Incircle


Circle>Concentric


Circle>Tangential


Circle>Radius - Point - Point


Circle>Radius - Object - Point


Circle>Radius - Object - Object

Circle>Radius - Point - Point (Draw Menu)

This command draws a circle with a given radius and which runs through two points.

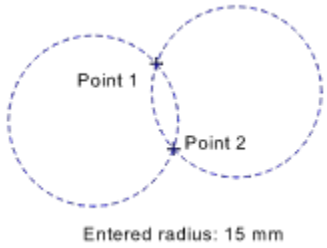
If a circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter radius*
   After choosing this command a dialog appears in which the radius of the circle can be entered. This radius remains valid until either a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Enter point 1*
   The first point through which the circle should run can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter point 2*
   The second point through which the circle should run can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Select object*
   After entering both points, both possible circles are calculated (the graphic shows one possible set of results).



   The 'right' circle can then be chosen with the help of the small dialog which appears at the bottom of the screen. Only one circle can be chosen at a time.

If the two points are further apart than twice the radius, it is not possible to calculate the circle, and an error message appears.

The circle is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

Circle>Standard

Circle>On Circle


Circle>Circumcircle


Circle>Diameter


Circle>Incircle


Circle>Concentric


Circle>Tangential


Circle>Line - Line - Line


Circle>Radius - Object - Point


Circle>Radius - Object - Object

Circle>Radius - Object - Point (Draw Menu)

This command draws a circle with a given radius which is tangent to a given line (or geometry line) and a given circle (or geometry circle) and runs through a given point.

If a circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter radius*
   After choosing this command a dialog appears in which the radius of the circle can be entered. This radius remains valid until either a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

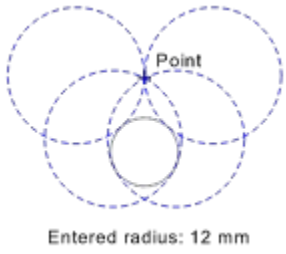
2. *Identify reference object*
   Any part of an existing object can be identified as a reference object. This can be a side of a rectangle or a circular arc within a surface. Ellipses and ellipse parts can not be identified.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The identified object is referred to as the "Reference Object".

   The position at which the object was identified (the "ID point") has no effect on the rest of the command. Because there are up to four possible results it would be too complicated to make a choice on the basis of the ID point's position.

3. *Enter point*
   The point through which the circle should run can be entered by clicking the mouse at any point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Select object*
   After identifying the reference object and entering the point, then all the possible circles are calculated (the graphic shows one possible result).



   One of the circles can be chosen with the help of a small dialog which appears near the bottom of the screen. Only one circle can be chosen.

If the points are further than twice the radius from the object then the circles can not be calculated. In this case an error message appears.

The circle is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held

down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Circle>Standard


Circle>On Circle


Circle>Circumcircle


Circle>Diameter


Circle>Incircle


Circle>Concentric


Circle>Tangential


Circle>Line - Line - Line


Circle>Radius - Point - Point


Circle>Radius - Object - Object

Circle>Radius - Object - Object (Draw Menu)

This command draws a circle with a given radius which is tangent to two specified lines (or geometry lines) or circles (or geometry circles)

If a circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter radius*
   After choosing this command a dialog appears in which the radius of the circle can be entered. This radius remains valid until either a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify reference object 1*
   Any part of an existing object can be identified as the first reference object. This can be a side of a rectangle or a circular arc within a surface. Ellipses and ellipse parts can not be identified.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

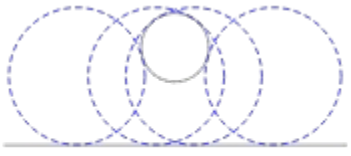
   The identified object is referred to as the "Reference Object".

   The position at which the object was identified (the "ID point") has no effect on the rest of the command. As there are up to eight possible results, a choice based on the position of the ID point would be too complex.

3. *Identify reference object 2*
   Any part of an existing object can be identified as the second reference object. This can be a side of a rectangle or a circular arc within a surface. Ellipses and ellipse parts can not be identified.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The identified object is referred to as the "Reference Object".

4. *Select object*
   After identifying both reference objects, all the possible circles are calculated (the graphic shows one possible resulting group).



   One of the circles can be chosen with the help of a small dialog which appears near the bottom of the screen. Only one circle can be chosen.

If the distance between the objects is more than double the radius or if two parallel lines are identified, the circles cannot be calculated and an error message appears.

The circle is assigned to the current layer. In addition, it contains a reference to the current pen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Circle>Standard


Circle>On Circle


Circle>Circumcircle


Circle>Diameter


Circle>Incircle


Circle>Concentric


Circle>Tangential


Circle>Line - Line - Line


Circle>Radius - Point - Point


Circle>Radius - Object - Point

Circular Arc>Standard (Draw Menu)

This command draws a standard circular arc. The circular arc is specified by entering the center, radius, start and end points.

If a circular arc is to be drawn while working in isometric or dimetric view, then the command Elliptical Arc>Distorted must be used, because the result on the page is elliptical not circular.

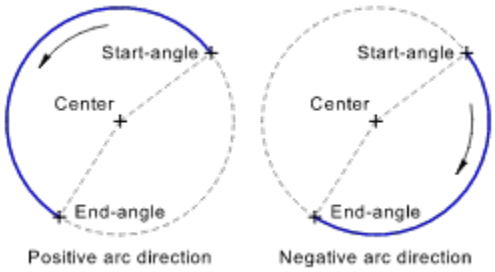
1. *Enter center point*
   The center of the arc can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter radius*
   The radius of the arc can be entered by clicking the mouse anywhere in the drawing at a point through which the arc should run. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter a specific radius. This gives the radius for the circular arc.

3. *Enter start angle*
   The arc's start angle can be entered by clicking the mouse anywhere in the drawing. The position of this point relative to the center determines the start angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the arc's start angle.

4. *Enter end angle*
   The arc's end angle can be entered by clicking the mouse anywhere in the drawing. The position of this point relative to the center determines the end angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the end angle of the arc. The angle can also be given relative to the previously entered start angle.

   The appearance of the arc is affected by the current arc direction setting. This is set using the command Toggle Arc Direction(F2), with the F2 key or the button in the panel. If the arc direction is positive, then the arc is drawn in a mathematically positive direction (i.e. anticlockwise) from the start angle to the end angle. If the arc direction is negative, then the arc is drawn in a mathematically negative direction (i.e. clockwise) from the start angle to the end angle (see graphic).



   The appearance of the arc is show while it is being entered and changes immediately if the current arc direction is changed.

The arc is assigned to the current layer. In addition, it contains a reference to the current pen.

The arc direction settings of existing arcs are not affected by changes to the current arc direction. To change the arc direction of an existing arc, use the command Transform Object to>Invert.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circular arcs will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Circular Arc>On Circle


Circular Arc>Circumcircle


Circular Arc>Diameter


Circular Arc>Concentric to Circle


Circular Arc>Concentric to Circle Part


Circular Arc>Radius - Point - Point


Circular Arc>Radius - Object - Point


Circular Arc>Radius - Object - Object

Circular Arc>On Circle (Draw Menu)

This command draws a circular arc on top of an existing circle. The arc is determined by identifying a circle and entering the start and end angles.

This command is particularly useful if a circular arc is to be drawn on a geometry element. In this case, the geometry circle must be identified and then the start and end angles are used as intersections within the geometry.

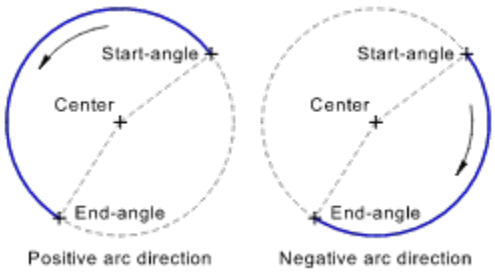
If a circular arc is to be drawn while working in isometric or dimetric view, then the command Elliptical Arc>Distorted must be used, because the result on the page is elliptical not circular.

1. *Identify reference circle*
   Any circle which exists independently or as part of another object can be identified as the reference object. This can also be the arc of a circular segment or a circular arc within a surface.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

2. *Enter start angle*
   The arc's start angle can be entered by clicking the mouse anywhere in the drawing. The position of this point relative to the center determines the start angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the arc's start angle.

3. *Enter end angle*
   The arc's end angle can be entered by clicking the mouse anywhere in the drawing. The position of this point relative to the center determines the end angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the end angle of the arc. The angle can also be given relative to the previously entered start angle.

   The appearance of the arc is affected by the current arc direction setting. This is set using the command Toggle Arc Direction(F2), with the F2 key or the button in the panel. If the arc direction is positive, then the arc is drawn in a mathematically positive direction (i.e. anticlockwise) from the start angle to the end angle. If the arc direction is negative, then the arc is drawn in a mathematically negative direction (i.e. clockwise) from the start angle to the end angle (see graphic).



   The appearance of the arc is show while it is being entered and changes immediately if the current arc direction is changed.

The arc is assigned to the current layer. In addition, it contains a reference to the current pen.

The arc direction settings of existing arcs are not affected by changes to the current arc direction. To change the arc direction of an existing arc, use the command Transform Object to>Invert.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circular arcs will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Circular Arc>Standard


Circular Arc>Circumcircle


Circular Arc>Diameter


Circular Arc>Concentric to Circle


Circular Arc>Concentric to Circle Part


Circular Arc>Radius - Point - Point


Circular Arc>Radius - Object - Point


Circular Arc>Radius - Object - Object

Circular Arc>Circumcircle (Draw Menu)

This command draws a circular arc which ends at two given points and runs through a third.

If a circular arc is to be drawn while working in isometric or dimetric view, then the command Elliptical Arc>Distorted must be used, because the result on the page is elliptical not circular.

1. *Enter start point*
   The start point can be entered by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter end point*
   The end point can be entered by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter point on arc*
   The third point through which the circular arc should run, can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

   The appearance of the arc is not dependent on the current arc direction, as the arc direction is determined implicitly by the position of the third point.

The arc is assigned to the current layer. In addition, it contains a reference to the current pen.

The arc direction settings of existing arcs are not affected by changes to the current arc direction. To change the arc direction of an existing arc, use the command Transform Object to>Invert.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circular arcs will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Circular Arc>Standard


Circular Arc>On Circle


Circular Arc>Diameter


Circular Arc>Concentric to Circle


Circular Arc>Concentric to Circle Part


Circular Arc>Radius - Point - Point


Circular Arc>Radius - Object - Point


Circular Arc>Radius - Object - Object

Circular Arc>Diameter (Draw Menu)

This command draws a circular arc. The circular arc is calculated by entering two points, one at either end of its diameter, plus start- and end-angle.

If a circular arc is to be drawn while working in isometric or dimetric view, then the command Elliptical Arc>Distorted must be used, because the result on the page is elliptical not circular.

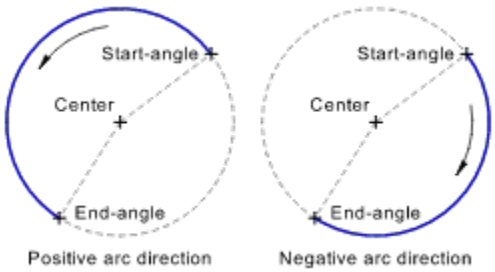
1. *Enter point 1 on circle*
   The first point through which the circle should run can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point 2 on circle*
   The second point through which the circle should run can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter start angle*
   The arc's start angle can be entered by clicking the mouse anywhere in the drawing. The position of this point relative to the center determines the start angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the arc's start angle.

4. *Enter end angle*
   The arc's end angle can be entered by clicking the mouse anywhere in the drawing. The position of this point relative to the center determines the end angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the end angle of the arc. The angle can also be given relative to the previously entered start angle.

   The appearance of the arc is affected by the current arc direction setting. This is set using the command Toggle Arc Direction(F2), with the F2 key or the button in the panel. If the arc direction is positive, then the arc is drawn in a mathematically positive direction (i.e. anticlockwise) from the start angle to the end angle. If the arc direction is negative, then the arc is drawn in a mathematically negative direction (i.e. clockwise) from the start angle to the end angle (see graphic).



   The appearance of the arc is show while it is being entered and changes immediately if the current arc direction is changed.

The arc is assigned to the current layer. In addition, it contains a reference to the current pen.

The arc direction settings of existing arcs are not affected by changes to the current arc direction. To change the arc direction of an existing arc, use the command Transform Object to>Invert.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circular arcs will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Circular Arc>Standard


Circular Arc>On Circle


Circular Arc>Circumcircle


Circular Arc>Concentric to Circle


Circular Arc>Concentric to Circle Part


Circular Arc>Radius - Point - Point


Circular Arc>Radius - Object - Point


Circular Arc>Radius - Object - Object

Circular Arc>Concentric to Circle (Draw Menu)

This command draws a circular arc which is concentric with another circle. The arc is determined by identifying the circle and entering the radius, start and end angles.

If a circular arc is to be drawn while working in isometric or dimetric view, then the command Elliptical Arc>Distorted must be used, because the result on the page is elliptical not circular.

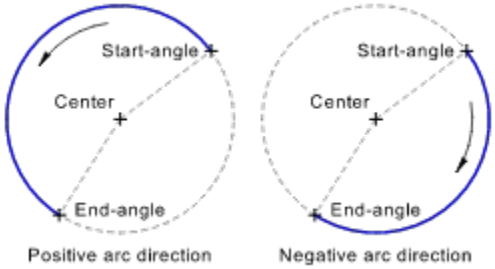
1. *Identify reference circle*
   Any circle which exists independently or as part of another object can be identified as the reference object. This can also be the arc of a circular segment or a circular arc within a surface.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

2. *Enter radius*
   The radius of the arc can be entered by clicking the mouse anywhere in the drawing at a point through which the arc should run. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter a specific radius. This gives the radius of the circle. The radius can be given relative to the radius of the reference circle.

3. *Enter start angle*
   The arc's start angle can be entered by clicking the mouse anywhere in the drawing. The position of this point relative to the center determines the start angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the arc's start angle.

4. *Enter end angle*
   The arc's end angle can be entered by clicking the mouse anywhere in the drawing. The position of this point relative to the center determines the end angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the end angle of the arc. The angle can also be given relative to the previously entered start angle.

   The appearance of the arc is affected by the current arc direction setting. This is set using the command Toggle Arc Direction(F2), with the F2 key or the button in the panel. If the arc direction is positive, then the arc is drawn in a mathematically positive direction (i.e. anticlockwise) from the start angle to the end angle. If the arc direction is negative, then the arc is drawn in a mathematically negative direction (i.e. clockwise) from the start angle to the end angle (see graphic).



   The appearance of the arc is show while it is being entered and changes immediately if the current arc direction is changed.

The arc is assigned to the current layer. In addition, it contains a reference to the current pen.

The arc direction settings of existing arcs are not affected by changes to the current arc direction. To change the arc direction of an existing arc, use the command Transform Object to>Invert.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circular arcs will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

 Circular Arc>Standard

 Circular Arc>On Circle

 Circular Arc>Circumcircle

 Circular Arc>Diameter

 Circular Arc>Concentric to Circle Part

 Circular Arc>Radius - Point - Point

 Circular Arc>Radius - Object - Point

 Circular Arc>Radius - Object - Object

Circular Arc>Concentric to Circle Part (Draw Menu)

This command draws a circular arc which runs concentric to another circular arc. After identifying the other circular arc, the arc is determined by entering its radius.

If a circular arc is to be drawn while working in isometric or dimetric view, then the command Elliptical Arc>Distorted must be used, because the result on the page is elliptical not circular.

1. *Identify reference circle part*
   Any circle part which exists independently or within other objects can be identified as a reference object. This can also be the arc of a circular segment or a circular arc within a surface.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

2. *Enter radius*
   The radius of the arc can be entered by clicking the mouse anywhere in the drawing at a point through which the arc should run. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter a specific radius. This gives the radius of the circle. The radius can be entered relative to the reference circle part's radius.

The start and end angle as well as the arc direction are taken from the reference circle part.

The arc is assigned to the current layer. In addition, it contains a reference to the current pen.

The arc direction settings of existing arcs are not affected by changes to the current arc direction. To change the arc direction of an existing arc, use the command Transform Object to>Invert.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circular arcs will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

- Circular Arc>Standard
- Circular Arc>On Circle
- Circular Arc>Circumcircle
- Circular Arc>Diameter
- Circular Arc>Concentric to Circle
- Circular Arc>Radius - Point - Point
- Circular Arc>Radius - Object - Point
- Circular Arc>Radius - Object - Object

Circular Arc>Radius - Point - Point (Draw Menu)

This command draws a circular arc which has a given radius and ends at two given points.

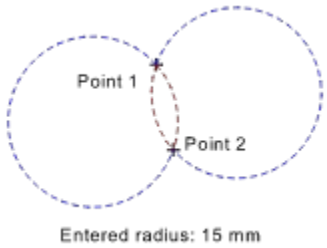
If a circular arc is to be drawn while working in isometric or dimetric view, then the command Elliptical Arc>Distorted must be used, because the result on the page is elliptical not circular.

1. *Enter radius*
   After choosing this command, a dialog appears in which a radius for the circular arc can be entered. This radius remains valid until a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Enter point 1*
   The first point, which the circular arc should end at, can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter point 2*
   The second point, which the circular arc should end at, can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Select object*
   After both points have been entered, all the possible circular arcs are calculated (the graphic shows one possible result).



   These are offered for choice. To do this, a small dialog from which the correct arc can be chosen, appears at the bottom of the screen. Only one circular arc can be chosen at a time.

If the distance between the two points is more than twice the radius, then the circular arc can not be calculated. In this case, an error message appears.

The arc is assigned to the current layer. In addition, it contains a reference to the current pen.

The arc direction settings of existing arcs are not affected by changes to the current arc direction. To change the arc direction of an existing arc, use the command Transform Object to>Invert.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circular arcs will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:

- Circular Arc>Standard
- Circular Arc>On Circle
- Circular Arc>Circumcircle
- Circular Arc>Diameter
- Circular Arc>Concentric to Circle
- Circular Arc>Concentric to Circle Part
- Circular Arc>Radius - Object - Point
- Circular Arc>Radius - Object - Object

Circular Arc>Radius - Object - Point (Draw Menu)

This command draws a circular arc which has a given radius, begins tangent to a given line (or geometry line) or circle (or geometry circle) and ends at a given point.

If a circular arc is to be drawn while working in isometric or dimetric view, then the command Elliptical Arc>Distorted must be used, because the result on the page is elliptical not circular.

1. *Enter radius*
   After choosing this command, a dialog appears in which a radius for the circular arc can be entered. This radius remains valid until a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

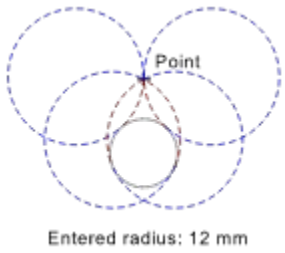
2. *Identify reference object*
   Any part of an existing object can be identified as a reference object. This can be the side of a rectangle or a circular arc within a surface. Ellipses and ellipse parts can not be identified.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The identified object is referred to as the "Reference Object".

   The position at which the object was identified (the "ID point") has no effect on the rest of the command. As there are up to eight possible results, it would be too complicated to make a choice on the basis of the ID point.

3. *Enter point*
   The point at which the arc should end can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Select object*
   After identifying the reference object and entering the point, all the possible arcs are calculated (the graphic shows one possible result).



   These are offered for choice. To do this, a small dialog from which the correct arc can be chosen, appears at the bottom of the screen. Only one circular arc can be chosen at a time.

If the distance from the point to the object is more than double the radius, then the arc can not be calculated. In this case, an error message appears.

The arc is assigned to the current layer. In addition, it contains a reference to the current pen.

The arc direction settings of existing arcs are not affected by changes to the current arc direction. To change the arc direction of an existing arc, use the command Transform Object to>Invert.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circular arcs will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Circular Arc>Standard


Circular Arc>On Circle


Circular Arc>Circumcircle


Circular Arc>Diameter


Circular Arc>Concentric to Circle


Circular Arc>Concentric to Circle Part


Circular Arc>Radius - Point - Point


Circular Arc>Radius - Object - Object

Circular Arc>Radius - Object - Object (Draw Menu)

This command draws a circle which has a given radius and is tangent to two given lines (or geometry lines) or circles (or geometry circles).

If a circular arc is to be drawn while working in isometric or dimetric view, then the command Elliptical Arc>Distorted must be used, because the result on the page is elliptical not circular.

1. *Enter radius*
   After choosing this command, a dialog appears in which a radius for the circular arc can be entered. This radius remains valid until a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify reference object 1*
   Any part object in an existing object can be identified as the first reference object. This can be a side of a rectangle or a circular arc within a surface. Ellipses and ellipse parts can not be identified.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

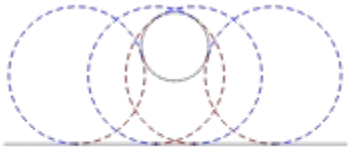
   The identified object is referred to as the "Reference Object".

   The position at which the object was identified (the "ID point") has no effect on the rest of the command. As there are up to eight possible results, it would be too complicated to make a choice on the basis of the ID point's position.

3. *Identify reference object 2*
   Any part object in an existing object can be identified as the second reference object. This can be a side of a rectangle or a circular arc within a surface. Ellipses and ellipse parts can not be identified.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

   The identified object is referred to as the "Reference Object".

4. *Select object*
   After identifying both reference objects, all possible circular arcs are calculated (the graphic shows one possible set of results).



   These are offered for choice. To do this, a small dialog from which the correct arc can be chosen, appears at the bottom of the screen. Only one circular arc can be chosen at a time.

If the distance between the two objects is more than double the radius, or if two parallel lines are identified, it is not possible to calculate the circular arc.

The arc is assigned to the current layer. In addition, it contains a reference to the current pen.

The arc direction settings of existing arcs are not affected by changes to the current arc direction. To change the arc direction of an existing arc, use the command Transform Object to>Invert.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circular arcs will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

Related commands:


Circular Arc>Standard


Circular Arc>On Circle


Circular Arc>Circumcircle


Circular Arc>Diameter


Circular Arc>Concentric to Circle


Circular Arc>Concentric to Circle Part


Circular Arc>Radius - Point - Point


Circular Arc>Radius - Object - Point

Ellipse>Rectangular (Draw Menu)

This command draws a rectangular ellipse. "Rectangular" means here that both of the ellipse's half-axes are at right angles to one another. In addition, when using this command they are horizontal and vertical. Horizontal and vertical refer to the position relative to the page and not the view. If the drawing is rotated, then the half-axes will be rotated correspondingly.

1. *Enter center point*
   The center point of the ellipse can be entered with the mouse by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point on frame*
   The point on the frame surrounding the ellipse can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). By entering relative values, both of the ellipse's half-axes can be entered precisely.

A rectangular ellipse is stored as a distorted ellipse. This means that it can be rotated or distorted later.

The ellipse is assigned to the current layer. It also contains a reference to the current pen.

Related commands:


Ellipse>Rotated


Ellipse>Distorted


Ellipse>On Ellipse

Ellipse>Rotated (Draw Menu)

This command draws a rotated rectangular ellipse. "Rectangular" means here that both of the ellipse's half-axes are at right angles to one another.

1. *Enter center point*
   The center point of the ellipse can be entered with the mouse by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point on frame*
   The point on the frame surrounding the ellipse can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). By entering relative values, both of the ellipse's half-axes can be entered precisely.

3. *Enter angle*
   The ellipse rotation angle can be entered with the mouse by clicking anywhere in the drawing. The position of this point relative to the center point determines the rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This determines the ellipse's rotation angle.

A rectangular ellipse is stored internally as a distorted ellipse. They can be further rotated or distorted later.

The ellipse is assigned to the current layer. It also contains a reference to the current pen.

Related commands:


Ellipse>Rectangular


Ellipse>Distorted


Ellipse>On Ellipse

Ellipse>Distorted (Draw Menu)

This command draws a distorted ellipse. "Distorted" means here that both half-axes are entered separately and can therefore be at any angle to one another.

This command is particularly useful when, in isometric or dimetric view, a circle has to be entered. To do this, both of the ellipse's half-axes have to run in the respective main directions.

1. *Enter center point*
   The center point of the ellipse can be entered with the mouse by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point 1 on ellipse*
   The first point through which the ellipse should run is the end point of the first half axis. It can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter point 2 on ellipse*
   The second point through which the ellipse should run is the end point of the second half axis. It can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The ellipse is assigned to the current layer. It also contains a reference to the current pen.

Related commands:


Ellipse>Rectangular


Ellipse>Rotated


Ellipse>On Ellipse

Ellipse>On Ellipse (Draw Menu)

This command draws an ellipse based on another ellipse or a ellipse part.

This command is especially useful if a geometry ellipse is to be transferred into the drawing. It can also be used to draw a ellipse over an existing ellipse part.

1. *Identify reference ellipse*
   Any ellipse which exists independently or as part of another object can be identified as the reference object. This can also be the arc of an elliptical segment.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

The ellipse is assigned to the current layer. In addition, it contains a reference to the current pen.

Related commands:

Ellipse>Rectangular

Ellipse>Rotated

Ellipse>Distorted

Elliptical Arc>Rectangular (Draw Menu)

This command draws a rectangular elliptic arc. "Rectangular" means here that both half-axes of the underling ellipse are at right angles to one another. With this command they are in addition horizontal and vertical. Horizontal and vertical refer to the position relative to the page, not the view. If the drawing is displayed rotated then the half axes are displayed correspondingly rotated.

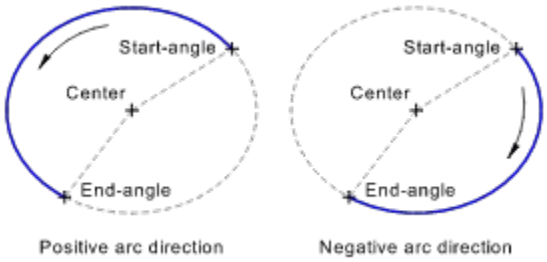
1. *Enter center point*
   The center point of the ellipse can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point on frame*
   The point on the frame surrounding the ellipse can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). By entering relative values, both of the ellipse's half-axes can be entered precisely.

3. *Enter start angle*
   The start angle of the elliptical arc can be entered with the mouse by clicking anywhere in the drawing. The position of this point relative to the center determines the start angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the start angle of the elliptical arc.

4. *Enter end angle*
   The end angle of the elliptical arc can be entered with the mouse by clicking anywhere in the drawing. The position of this point relative to the center determines the end angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the end angle of the elliptical arc. The angle can also be given relative to the previously entered start angle.

   The appearance of the elliptical arc is determined by the current arc direction setting. This is set using the command Toggle Arc Direction(F2), with the F2 key or the button in the panel. If the arc direction is positive, the elliptical arc is drawn in a mathematically positive (i.e. anti-clockwise) direction from the start angle to the end angle. If the arc direction is negative, the elliptical arc is drawn in a mathematically negative (i.e. clockwise) direction from the start angle to the end angle (see graphic).



   The later appearance of the elliptical arc is shown during entry and changes immediately if the current arc direction is changed during entry.

A rectangular elliptical arc is stored as a distorted elliptical arc. It can therefore be further rotated or distorted later.

The elliptical arc is assigned to the current layer. In addition, it contains a reference to the current pen.

The arc direction settings saved in an elliptical arc are not affected by changing the current arc direction. To change the arc direction of an existing arc, use the command Transform Object to>Invert.

Related commands:

Elliptical Arc>Distorted

Elliptical Arc>On Ellipse

Elliptical Arc>Distorted (Draw Menu)

This command draws a distorted elliptical arc. "Distorted" means here that both half-axes of the underlying ellipse are entered separately and can be at any angle to one another.

This command is particularly useful if, in isometric or dimetric view, a circular arc has to be entered. To do this, both half-axes of the underling ellipse must run in the main directions.

1. *Enter center point*
   The center point of the ellipse can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

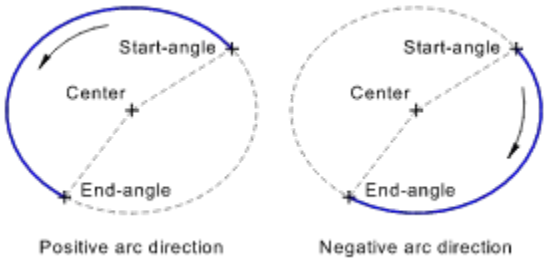
2. *Enter point 1 on ellipse*
   The first point through which the ellipse should run is the end point of the first half axis. It can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter point 2 on ellipse*
   The second point through which the ellipse should run is the end point of the second half axis. It can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter start angle*
   The start angle of the elliptical arc can be entered with the mouse by clicking anywhere in the drawing. The position of this point relative to the center determines the start angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the start angle of the elliptical arc.

5. *Enter end angle*
   The end angle of the elliptical arc can be entered with the mouse by clicking anywhere in the drawing. The position of this point relative to the center determines the end angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the end angle of the elliptical arc. The angle can also be given relative to the previously entered start angle.

   The appearance of the elliptical arc is determined by the current arc direction setting. This is set using the command Toggle Arc Direction(F2), with the F2 key or the button in the panel. If the arc direction is positive, the elliptical arc is drawn in a mathematically positive (i.e. anti-clockwise) direction from the start angle to the end angle. If the arc direction is negative, the elliptical arc is drawn in a mathematically negative (i.e. clockwise) direction from the start angle to the end angle (see graphic).



   The later appearance of the elliptical arc is shown during entry and changes immediately if the current

arc direction is changed during entry.

The elliptical arc is assigned to the current layer. In addition, it contains a reference to the current pen.

The arc direction settings saved in an elliptical arc are not affected by changing the current arc direction. To change the arc direction of an existing arc, use the command Transform Object to>Invert.

Related commands:

Elliptical Arc>Rectangular

Elliptical Arc>On Ellipse

Elliptical Arc>On Ellipse (Draw Menu)

This command draws an elliptical arc on top of an existing ellipse. The arc is determined by identifying an ellipse and entering the start and end angles.

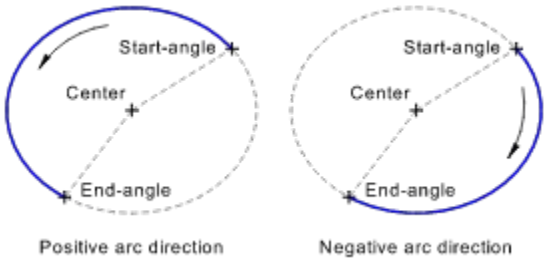
This command is particularly useful if an elliptical arc is to be drawn on a geometry element. In this case, the geometry ellipse must be identified and then the start and end angles are used as intersections within the geometry.

1. *Identify reference ellipse*
   Any ellipse which exists independently or as part of another object can be identified as the reference object. This can also be the arc of an elliptical segment.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

2. *Enter start angle*
   The arc's start angle can be entered by clicking the mouse anywhere in the drawing. The position of this point relative to the center determines the start angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the arc's start angle.

3. *Enter end angle*
   The arc's end angle can be entered by clicking the mouse anywhere in the drawing. The position of this point relative to the center determines the end angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter an angle directly into a dialog. This gives the end angle of the arc. The angle can also be given relative to the previously entered start angle.

   The appearance of the elliptical arc is affected by the current arc direction setting. This is set using the command Toggle Arc Direction(F2), with the F2 key or the button in the panel. If the arc direction is positive, then the elliptical arc is drawn in a mathematically positive direction (i.e. anticlockwise) from the start angle to the end angle. If the arc direction is negative, then the elliptical arc is drawn in a mathematically negative direction (i.e. clockwise) from the start angle to the end angle (see graphic).



   The appearance of the arc is show while it is being entered and changes immediately if the current arc direction is changed.

The elliptical sector is assigned to the current layer. In addition, it contains a reference to the current pen.

The arc direction setting saved in the elliptical sector is not affected by changing the current arc direction. To change the arc direction of an existing elliptical sector, use the command Transform Object to>Invert.

Related commands:

Elliptical Arc>Rectangular

Elliptical Arc>Distorted

Arc Mode>Arc (Draw Menu)

This command sets the current arc mode to "Arc", i.e. all subsequently created circle and ellipse parts will appear as arcs.

The state of an existing circle or ellipse part can be altered by means of the following commands:

- Transform Object to>Circular Arc
- Transform Object to>Circular Sector
- Transform Object to>Circular Segment
- Transform Object to>Elliptic Arc
- Transform Object to>Elliptic Sector
- Transform Object to>Elliptic Segment


Optionally, the arc mode can be changed by clicking the left mouse button on the button in the panel shown to the left. The button shows the current arc mode.

Related commands:

- Arc Mode>Sector
- Arc Mode>Segment
- Arc Mode>Toggle Arc Mode
- Arc Mode>Toggle Arc Direction

Arc Mode>Sector (Draw Menu)

This command sets the current arc mode to "Sector", i.e. all subsequently created circle and ellipse parts will appear as sectors.

The state of an existing circle or ellipse part can be altered by means of the following commands:

- Transform Object to>Circular Arc
- Transform Object to>Circular Sector
- Transform Object to>Circular Segment
- Transform Object to>Elliptic Arc
- Transform Object to>Elliptic Sector
- Transform Object to>Elliptic Segment

Optionally, the arc mode can be changed by clicking the left mouse button on the button in the panel shown to the left. The button shows the current arc mode.

Related commands:

- Arc Mode>Arc
- Arc Mode>Segment
- Arc Mode>Toggle Arc Mode
- Arc Mode>Toggle Arc Direction

Arc Mode>Segment (Draw Menu)

This command sets the current arc mode to "Segment", i.e. all subsequently created circle and ellipse parts will appear as segment.

The state of an existing circle or ellipse part can be altered by means of the following commands:

- Transform Object to>Circular Arc
- Transform Object to>Circular Sector
- Transform Object to>Circular Segment
- Transform Object to>Elliptic Arc
- Transform Object to>Elliptic Sector
- Transform Object to>Elliptic Segment

Optionally, the arc mode can be changed by clicking the left mouse button on the button in the panel shown to the left. The button shows the current arc mode.

Related commands:

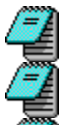
- Arc Mode>Arc
- Arc Mode>Sector
- Arc Mode>Toggle Arc Mode
- Arc Mode>Toggle Arc Direction

Arc Mode>Toggle Arc Mode (Draw Menu)

This command can be used to change the current arc mode. Each call of this command cyclically switches to the next arc mode, in the order "Arc" - "Sector" - "Segment". All subsequently created circle and ellipse parts will use this new arc mode.

The state of an existing circle or ellipse part can be altered by means of the following commands:

- Transform Object to>Circular Arc
- Transform Object to>Circular Sector
- Transform Object to>Circular Segment
- Transform Object to>Elliptic Arc
- Transform Object to>Elliptic Sector
- Transform Object to>Elliptic Segment



Optionally, the arc mode can be changed by clicking the left mouse button on the button in the panel shown to the left. The button shows the current arc mode.

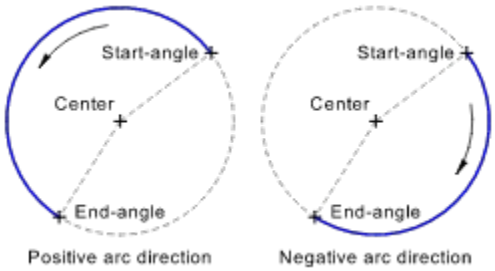
Related commands:

- Arc Mode>Arc
- Arc Mode>Sector
- Arc Mode>Segment
- Arc Mode>Toggle Arc Direction

Arc Mode>Toggle Arc Direction (Draw Menu)

This command can be used to change the current arc direction. The arc direction particularly affects the drawing of circle and ellipse parts. It determines whether the arc should be drawn in a mathematically positive (I.e. anticlockwise) or negative (i.e. clockwise) direction from the start angle to the end angle.

This is an example of the effect of the arc direction command on a circular arc.




Optionally, the arc direction can be changed by clicking the left mouse button on the button in the panel shown to the left. The button shows the current arc direction.

Related commands:

Arc Mode>Arc

Arc Mode>Sector

Arc Mode>Segment

Arc Mode>Toggle Arc Mode

Freehand Line (Draw Menu)

This command draws a freehand line. Like a polyline, a freehand line is entered as an open series of connected lines. After entering all lines, the program calculates a "freehand" curve that runs through all entered points.

1. *Enter start point*
   The start point of a freehand line can be entered by clicking the mouse anywhere within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point*
   After entering the start point, several points can be entered, one after another. The freehand line begins at the start point and runs through all the subsequently-entered points. These points can be entered by clicking the mouse at any point in the drawing for each point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter the length and/or angle of the next section relative to the previous section. This is an easy way, for example, to enter bends with a given angle.

   Freehand line input is ended by clicking the right mouse button. Unlike most other commands, this does not lead to the previous entry being ignored; instead the freehand line is inserted into the drawing. If required, this can be undone using the Undo command.

A freehand line can consist of up to 1000 segments. If this limit is reached during entry, further points can not be entered.

A freehand line is stored internally as a curve. It can be modified with the same commands as curves.

The curve is assigned to the active layer. It also contains a reference to the current pen.

When choosing the Change Parameters (+ESC) command, a dialog appears where parameters for the generation of freehand lines can be entered. These parameters are used for all subsequently drawn freehand lines.
The parameters consist of an accuracy and a waviness. The accuracy determines the average length of the lines that are produced to build the freehand line, the waviness determines how much each of these lines may deverge from the reference line.

Previously drawn freehand lines are not affected by changing the current accuracy. The accuracy of existing freehand lines cannot be changed.

All point entries can be undone step by step using Previous Point (ESC) or by pressing the ESC key. Incorrect entries can be easily corrected without having to reenter the whole freehand line.

Related commands:

Curve

Surface


Spline


Line>Polyline


Polygon>Arbitrary

Spline (Draw Menu)

This command draws a spline. A spline is a curve which runs through a number of points, joining them as smoothly as possible.

1. *Enter start point*
   The start point of the spline can be entered using the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point*
   After entering the start point, more points can be entered one after another. The spline begins at the start point and runs through all subsequently entered points. These points can be entered using the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition you can specify the length and/or angle of the next section relative to the previously entered section.

   Spline entry is ended by clicking the right mouse button. Unlike most other commands this does not lead to the previous entry being ignored, but instead inserts the spline. If this is not wanted, the Undo command must be chosen.

A spline can consist of up to 1000 points. If the limit is reached, further entry is not allowed.

Because a spline needs a lot of calculation, splines should only be used sparingly. If shallow curves are needed which do not have to be splines, the use of Bézier curves is preferable. Bézier curves can be drawn with the Curve and Surface commands.

The spline is assigned to the current layer. It also contains a reference to the current pen.

Related commands:

 Curve

 Surface

 Line>Polyline

 Polygon>Arbitrary

 Freehand Line

Curve (Draw Menu)

This command draws a curve. A curve is made up of lines, circular arcs and Bézier curves. They can be combined to make a complex line

According to the complexity of this object type, its entry can be very complicated. The all-encompassing process described here is only needed in a few special cases. Mostly, the same thing can be achieved more simply.
If a curve is to be made up only of lines, it can be entered using the command Line>Polyline. In many other cases it is better to draw all the part objects and then transform them to curves using the command Transform Objects>To Curves and then merge them together using the command Trim Curve>Extent.

After choosing the command nothing happens until the start point of the curve is entered.

1. *Enter start point*
   The start point of the curve can be entered with the mouse by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After entering the start point, the Curve Entry dialog appears, in which the procedure to be used during surface entry can be specified. The way entry is carried out depends on which buttons are activated in this dialog.

The sequence of point entry specified will be repeated until input is interrupted by pressing the right mouse button. This recalls the dialog where a new choice can be made.

The surface is assigned to the current layer. It also contains a reference to the current pen.

A surface can consist of up to 1000 points, where each line requires one point, a circular arc two points and a Bézier curve three points. If the limit is reached, further entry is not allowed. These 1000 points can be distributed on up to 100 outlines in the surface.

Related commands:

 Surface

 Spline

 Line>Polyline

 Polygon>Arbitrary

 Freehand Line

Surface (Draw Menu)

This command draws a surface. A surface is made up of lines, circular arcs and Bézier curves. These can be fitted together to make a closed surface, known as an outline. A surface can be made up of more than one closed outline.

According to the complexity of this object type, its entry can be very complicated. The all-encompassing process described here is only needed in a few special cases. Mostly, the same thing can be achieved more simply.
A surface made up only of lines can be entered using the command Polygon>Arbitrary. In many other cases it is sensible to draw part surfaces and combine them using Trim Surface>Union (A=A+B), Trim Surface>Intersection (A=A×B) and Trim Surface>Difference (A=A-B).

After choosing the command nothing happens until the start point of the first outline is entered.

1. *Enter start point*
   The start point of the first outline can be entered with the mouse by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After entering the start point, the Curve Entry dialog appears, in which the procedure to be used during surface entry can be specified. The way entry is carried out depends on which buttons are activated in this dialog.

The sequence of point entry specified will be repeated until input is interrupted by pressing the right mouse button. This recalls the dialog where a new choice can be made.

The surface is assigned to the current layer. It also contains a reference to the current pen.

A surface can consist of up to 1000 points, where each line requires one point, a circular arc two points and a Bézier curve three points. If the limit is reached, further entry is not allowed. These 1000 points can be distributed on up to 100 outlines in the surface

Related commands:

- Trim Surface>Generate Surface
- Curve
- Spline
- Line>Polyline
- Polygon>Arbitrary
- Freehand Line

Hatching>Objects (Draw Menu)

This command can be used to hatch a surface defined by the union of all chosen objects.

1. *Choose objects*
   The objects which are to be hatched must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing the objects, the union surface of the chosen object is hatched. Only those objects which have a closed outline and thus form a surface are regarded. The hatching type chosen with the command Hatching>Hatching Type List is used. The fixed point of block-based hatchings can be placed using the command Hatching>Set Fixed Point.

Use the command Change Parameters (+ESC) to edit this command's parameters.

The hatching consists of one or more "hatching" objects. These objects are not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The pen assignments from the current hatching type are used.

Related command:

Hatching>Generated Surface

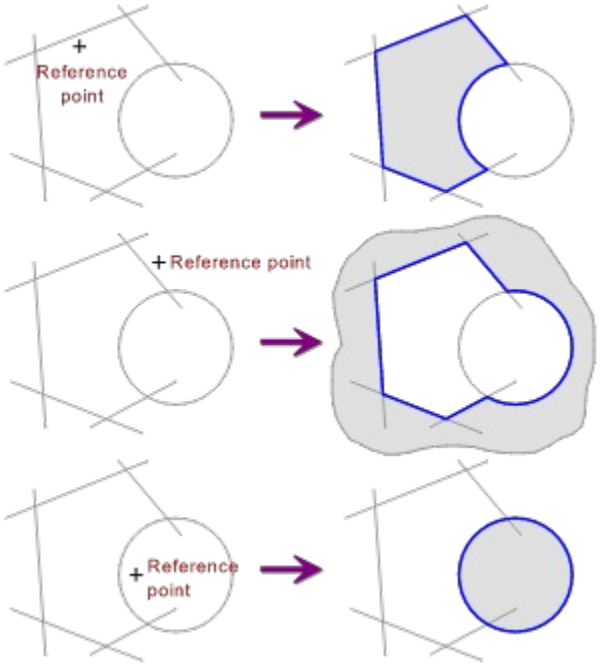
Hatching>Generated Surface (Draw Menu)

This command can be used to hatch a surface that is generated out of chosen objects.

1. *Choose objects*
   The objects whose outlines are to be included in the surface are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

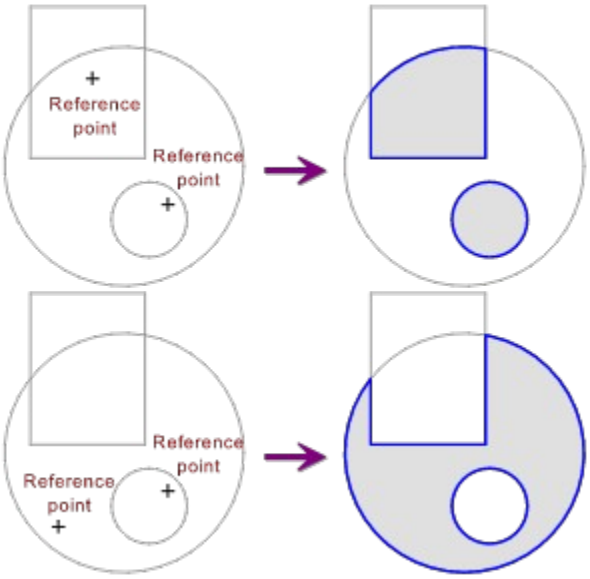
2. *Enter reference point*
   After choosing the objects, several reference points can be entered, one after another. These points can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

   The position of the reference points determines where and how the surface should be generated. Each reference point should be close to one of the outline elements to be incorporated in the surface. It shows with which element the search for a closed outline should begin. Through its position relative to this element, it also determines whether the "largest possible" or "smallest possible" surface should be worked out (see graphic).



   The effect of the reference point can be explained (simply) as follows: Imagine emptying a can of paint at the reference point. The paint spreads out in all directions until it reaches an object.
   The result is one area which is covered in paint, and one area which is not. The boundary between these two areas is the outline of the newly-generated surface.

   If multiple reference points are entered, one outline will be calculated for each of those reference points accordings to the method described above. The resulting outlines will then be combined to one surface. This way it's easy to generate complex surface (see graphic).



Reference point input is ended by clicking the right mouse button. Unlike most other commands, this does not lead to the previous entry being ignored; instead the command is processed with all previously entered reference points. If required, this can be undone using the Undo command.

After entering the reference points, the surface is generated. If an error message appears although it ought to be possible to generate the surface, try increasing the tolerance.

Use the command Change Parameters (+ESC) to edit this command's parameters.

If a surface has been generated successfully, the surface is hatched. The hatching type chosen with the command Hatching>Hatching Type List is used. The fixed point of block-based hatchings can be placed using the command Hatching>Set Fixed Point.

The hatching consists of one or more "hatching" objects. These objects are not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The pen assignments from the current hatching type are used.

As the result is a surface, only its elements (lines, circular arcs and Bézier curves) can be used when calculating the result. Many object types (above all, ellipses and ellipse parts) must be converted to polylines first, which inevitably causes loss of quality. The accuracy of the conversion can be specified using the command Settings>Screen under the heading "Accuracy - Transform".

Related command:

Hatching>Objects

Hatching>Hatching Type List (Draw Menu)

This command can be used to display a list of all currently defined hatching types. In this list, a new "active" hatching type can be chosen.

Affected commands:

Hatching>Objects

Hatching>Generated Surface

Hatching>Edit Hatching Types (Draw Menu)

This command can be used to manage hatching types. They can be defined, edited, deleted or made current.

When choosing this command, the Hatching Types dialog appears.

Affected commands:

Hatching>Objects

Hatching>Generated Surface

Hatching>Edit Line Sequences (Draw Menu)

This command can be used to manage line sequences. They can be defined, edited or deleted. Line sequences are mainly used inside hatching types, but can also be used in connection with the Multiline option (see Duplicate / Multiline (F7)).

When choosing this command, the Line Sequences dialog appears.

Affected commands:

Hatching>Objects

Hatching>Generated Surface

Hatching>Set Fixed Point (Draw Menu)

This command is used to place the fixed point for block-based hatching. The point entered here will be used as a base point for the grid of blocks created during the hatching.

1. *Set origin*
   The fixed point can be entered by clicking the mouse anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

Affected commands:


Hatching>Objects


Hatching>Generated Surface

Geometry Menu

- Geometry Line
  - Standard
  - On line
  - Horizontal
  - Vertical
  - Mid-Perpendicular
  - Perpendicular
  - Parallel
  - Parallel, Numerical
  - Angle to Line
  - Bisector of Angle
  - Center Cross
- Geometry Tangent
  - Object - Point
  - Object, Horizontal
  - Object, Vertical
  - Angle - Object
  - Object - Object
- Geometry Circle
  - Standard
  - On Circle
  - Circumcircle
  - Diameter
  - Incircle
  - Concentric
  - Tangential
  - Line - Line - Line
  - Radius - Point - Point
  - Radius - Object - Point
  - Radius - Object - Object
- Geometry Ellipse
  - Rectangular
  - Rotated
  - Distorted
  - On Ellipse
- Display Geometry
- Freeze Geometry
- Marking
- Division
  - Distance
  - Object
  - Array
- Multiple Copy
  - Markings
  - Rotation

Gradual
Distance
Object
Array

Geometry Line>Standard (Geometry Menu) 

This command is used to draw a line in the geometry.

1. *Enter start point*
   The start point of the geometry line can be entered with the mouse by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter end point*
   The end point of the geometry line can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

 Geometry Line>On Line

 Geometry Line>Horizontal

 Geometry Line>Vertical

 Geometry Line>Mid-Perpendicular

 Geometry Line>Perpendicular

 Geometry Line>Parallel

 Geometry Line>Parallel, Numerical

 Geometry Line>Angle to Line

 Geometry Line>Bisector of Angle

Geometry Line>On Line (Geometry Menu) 

This command draws a geometry line on an extension of an existing line.

1. *Identify reference line*
   Any line in an existing object can be identified as the reference line. This can also be a side of a rectangle or a part of a circular segment.

   The line is identified by clicking on it with the mouse. If more than one line is close to the position clicked on, a small dialog appears at the bottom of the screen with the help of which the correct line can be chosen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

 Geometry Line>Standard

 Geometry Line>Horizontal

 Geometry Line>Vertical

 Geometry Line>Mid-Perpendicular

 Geometry Line>Perpendicular

 Geometry Line>Parallel

 Geometry Line>Parallel, Numerical

 Geometry Line>Angle to Line

 Geometry Line>Bisector of Angle

Geometry Line>Horizontal (Geometry Menu) 

This command draws a horizontal geometry line. "Horizontal" relates to the position on the page and not to the view. If the drawing is rotated, then the line will be rotated correspondingly.

1. *Enter start point*
   The start point of the geometry line can be entered with the mouse by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

 Geometry Line>Standard

 Geometry Line>On Line

 Geometry Line>Vertical

 Geometry Line>Mid-Perpendicular

 Geometry Line>Perpendicular

 Geometry Line>Parallel

 Geometry Line>Parallel, Numerical

 Geometry Line>Angle to Line

 Geometry Line>Bisector of Angle

Geometry Line>Vertical (Geometry Menu) 

This command draws a vertical geometry line. "Vertical" relates to the position on the page and not to the view. If the drawing is rotated, then the line will be rotated correspondingly.

1. *Enter start point*
   The start point of the geometry line can be entered with the mouse by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

 Geometry Line>Standard

 Geometry Line>On Line

 Geometry Line>Horizontal

 Geometry Line>Mid-Perpendicular

 Geometry Line>Perpendicular

 Geometry Line>Parallel

 Geometry Line>Parallel, Numerical

 Geometry Line>Angle to Line

 Geometry Line>Bisector of Angle

Geometry Line>Mid-Perpendicular (Geometry Menu)

This command draws a geometry line which runs along the mid-perpendicular of another line. The geometry line is so constructed, that the right angle is given in normal view. If working with a distorted view (e.g. when using dimetric view) the visible angle is not 90 degrees.

1. *Identify reference line*
   Any line in an existing object can be identified as the reference line. This can also be a side of a rectangle or a part of a circular segment. Geometry lines cannot be used because they have no "middle".
   The line is identified by clicking on it with the mouse. If more than one line is close to the position clicked on, a small dialog appears at the bottom of the screen with the help of which the correct line can be chosen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

- Geometry Line>Standard
- Geometry Line>On Line
- Geometry Line>Horizontal
- Geometry Line>Vertical
- Geometry Line>Perpendicular
- Geometry Line>Parallel
- Geometry Line>Parallel, Numerical
- Geometry Line>Angle to Line
- Geometry Line>Bisector of Angle

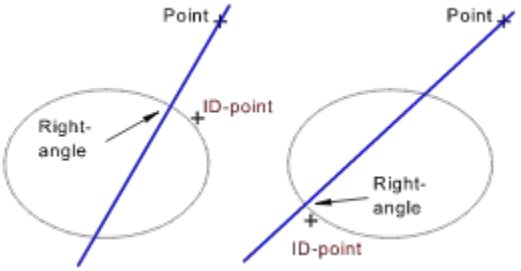
Geometry Line>Perpendicular (Geometry Menu)

This command draws a geometry line which is perpendicular to another line (or geometry line), circle (or geometry circle) or an ellipse. The geometry line is so constructed, that the right angle is given in normal view. If working with a distorted view (e.g. when using dimetric view) the visible angle is not 90 degrees.

1. *Identify reference object*
   Any part object in an existing object can be identified as the reference object. This can be a side of a rectangle, the arc of an ellipse segment or a circular segment within a surface.
   The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

   The identified (part) object is referred to as the "Reference Object".

   If an ellipse part was identified, the point at which it was clicked on (the "ID point") affects the way the rest of the command is carried out. If a perpendicular is dropped on to an ellipse, it can end on either side of the ellipse. The point closer to the ID point is used as the end point of the perpendicular (see graphic).



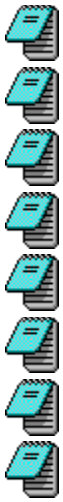
2. *Enter point*
   The point through which the perpendicular should run can be entered with the mouse by clicking anywhere in the drawing. A perpendicular is dropped from this point to the reference object. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If a circle is identified as the reference object, then it is possible to enter the length and angle of the perpendicular. The angle indicates the direction of the perpendicular (beginning with the reference object), the length gives the length of the perpendicular. If the ID point is on the "wrong" side of the circle, the perpendicular is drawn through the circle. In this case, its length is the specified length plus the diameter of the circle.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

Geometry Line>Standard



Geometry Line>On Line

Geometry Line>Horizontal

Geometry Line>Vertical

Geometry Line>Mid-Perpendicular

Geometry Line>Parallel

Geometry Line>Parallel, Numerical

Geometry Line>Angle to Line

Geometry Line>Bisector of Angle

Geometry Line>Parallel (Geometry Menu) 

This command draws a geometry line which runs parallel to another line or geometry line.

The distance is not entered numerically, but is implied by the entry of a point on the line If a known numeric distance is to be entered, use the command Geometry Line>Parallel, Numerical.

1. *Identify reference line*
   Any line in an existing object can be identified as the reference line. This can also be a side of a rectangle or a part of a circular segment.
   The line is identified by clicking on it with the mouse. If more than one line is close to the position clicked on, a small dialog appears at the bottom of the screen with the help of which the correct line can be chosen.

   After identification a straight line which runs parallel to the identified line and moves with the mouse appears. This line is referred to as the "Reference Straight Line".

2. *Enter point*
   After identifying the reference line, further points can be entered. Each point defines a parallel which runs through that point. This point can be entered with the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

   Parallel entry is ended by clicking the right mouse button. Unlike other commands, this does not lead to the last entry being ignored, but inserts the parallel(s). If this is not desired, the command Undo must be chosen next.

Up to 1000 parallels can be entered at one time. If this limit is reached, then no further entries are accepted.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Point entry can be undone, step by step, using the command Previous Point (ESC) or the ESC key. This makes it easy to correct incorrect entries without having to re-enter all the previously entered parallels.

Related commands:

 Geometry Line>Standard

 Geometry Line>On Line

 Geometry Line>Horizontal


Geometry Line>Vertical


Geometry Line>Mid-Perpendicular


Geometry Line>Perpendicular


Geometry Line>Parallel, Numerical


Geometry Line>Angle to Line


Geometry Line>Bisector of Angle

Geometry Line>Parallel, Numerical (Geometry Menu)

This command draws a geometry line which runs parallel to another line or geometry line.

The suffix "Numerical" indicates that the distance is specified by entering a specific value. If the size of the distance is not known and the distance should be entered implicitly using the mouse, use the command Geometry Line>Parallel.

1. *Enter distance*
   After choosing the command, a dialog appears into which the distance can be entered. This distance remains valid until a new command is chosen or the current command is restarted using Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify reference line*
   Any line in an existing object can be identified as the reference line. This can also be a side of a rectangle or a part of a circular segment.
   The line is identified by clicking on it with the mouse. If more than one line is close to the position clicked on, a small dialog appears at the bottom of the screen with the help of which the correct line can be chosen.

   After identification a straight line appears which moves when the mouse is moved. It always moves in multiples of the distance. This line is referred to as the "Reference Straight Line".

3. *Enter point*
   After identifying the reference line, further points can be entered. Each point defines a parallel which runs through that point and is a multiple of the specified distance from the reference line. This point can be entered with the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

   Parallel entry is ended by clicking the right mouse button. Unlike other commands, this does not lead to the last entry being ignored, but inserts the parallel(s). If this is not desired, the command Undo must be chosen next.

Up to 1000 parallels can be entered at one time. If this limit is reached, then no further entries are accepted.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Point entry can be undone, step by step, using the command Previous Point (ESC) or the ESC key. This makes it easy to correct incorrect entries without having to re-enter all the previously entered parallels.

Related commands:

- Geometry Line>Standard
- Geometry Line>On Line
- Geometry Line>Horizontal
- Geometry Line>Vertical
- Geometry Line>Mid-Perpendicular
- Geometry Line>Perpendicular
- Geometry Line>Parallel
- Geometry Line>Angle to Line
- Geometry Line>Bisector of Angle

Geometry Line>Angle to Line (Geometry Menu)

This command draws a geometry line which runs at a specified angle to another line or geometry line.

1. *Enter angle*
   After choosing this command, a dialog appears where a specific angle can be entered. This angle remains valid until either another command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify reference line*
   Any line in an existing object can be identified as the reference line. This can also be a side of a rectangle or a part of a circular segment.
   The line is identified by clicking on it with the mouse. If more than one line is close to the position clicked on, a small dialog appears at the bottom of the screen with the help of which the correct line can be chosen.

   After identification a straight line appears which runs at the specified angle to the identified line. This line is referred to as the "Reference Straight Line".

3. *Enter point*
   The point through which the line should run can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

 Geometry Line>Standard

 Geometry Line>On Line

 Geometry Line>Horizontal

 Geometry Line>Vertical

 Geometry Line>Mid-Perpendicular

 Geometry Line>Perpendicular

 Geometry Line>Parallel

 Geometry Line>Parallel, Numerical

 Geometry Line>Bisector of Angle

Geometry Line>Bisector of Angle (Geometry Menu)

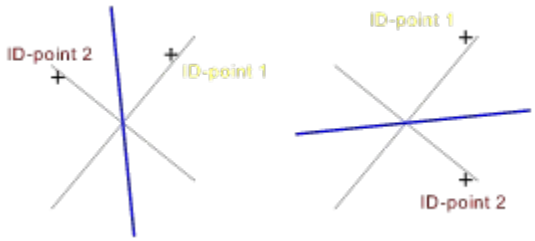
This command draws a geometry line which runs along the angle bisector of two other lines or geometry lines.

1. *Identify reference line 1*
   Any line in an existing object can be identified as the first reference line. This can also be a side of a rectangle or a part of a circular segment.
   The line is identified by clicking on it with the mouse. If more than one line is close to the position clicked on, a small dialog appears at the bottom of the screen with the help of which the correct line can be chosen.

   The position at which the line was identified (the "ID point") affects the way in which the rest of the command is carried out. Together with the position at which the second line was identified, it determines which of the two possible angle bisectors should be used (see below).

2. *Identify reference line 2*
   Any line in an existing object can be identified as the second reference line. This can also be a side of a rectangle or a part of a circular segment.
   The line is identified by clicking on it with the mouse. If more than one line is close to the position clicked on, a small dialog appears at the bottom of the screen with the help of which the correct line can be chosen.

   The position at which the line was identified (the "ID point") affects the way in which the rest of the command is carried out. Together with the position at which the first line was identified, it determines which of the two possible angle bisectors should be used (see graphic).



If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

Geometry Line>Standard

Geometry Line>On Line

Geometry Line>Horizontal

Geometry Line>Vertical

Geometry Line>Mid-Perpendicular

Geometry Line>Perpendicular

Geometry Line>Parallel

Geometry Line>Parallel, Numerical

Geometry Line>Angle to Line

Geometry Line>Center Cross (Geometry Menu) 

This command draws to geometry lines. One runs horizontally and one runs vertically. They cross at the center of an identified circle or ellipse.

1. *Identify reference circle / ellipse*
   Any circle or ellipse which exists on its own or as part of another object can be identified as a reference object. This can also be the arc of an elliptical segment or a circular arc within a surface. The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:


Geometry Line>Horizontal

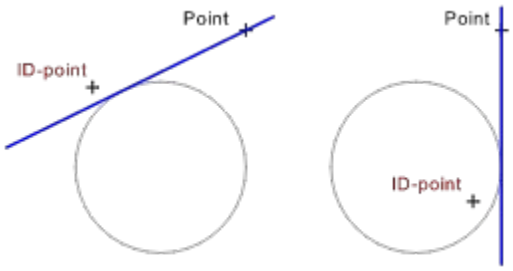

Geometry Line>Vertical

Geometry Tangent>Object - Point (Geometry Menu)

This command draws a geometry line which begins at a given point and runs tangent to an ellipse or circle.

1. *Identify reference circle / ellipse*
   Any circle or ellipse which exists on its own or as part of another object can be identified as a reference object. This can also be the arc of an elliptical segment or a circular arc within a surface. The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

   The position at which the (part) object is identified (the "ID point) affects the way in which the rest of the command is carried out. A tangent to a circle or ellipse can be drawn to either side of the circle or ellipse. The tangent point which is nearer to the ID point is used as the end point for the line.



2. *Enter point*
   The point through which the tangent should run can be entered with the mouse by clicking anywhere in the drawing. The tangent is dropped from this point to the object to work out the second point of the tangent. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

   This command does not function if the entered point is on the object outline or inside the object. In this case point entry is refused.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

Geometry Tangent>Object, Horizontal

Geometry Tangent>Object, Vertical

Geometry Tangent>Angle - Object

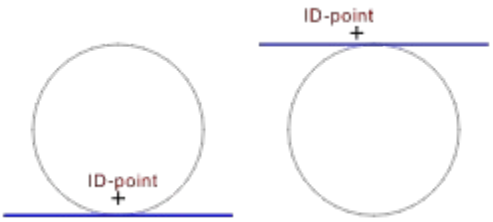
Geometry Tangent>Object - Object

Geometry Tangent>Object, Horizontal (Geometry Menu)

This command draws a horizontal geometry line which is tangent to a circle or ellipse. "Horizontal" relates to the position on the page and not to the view. If the drawing is rotated, then the line will be rotated correspondingly.

1. *Identify reference circle / ellipse*
   Any circle or ellipse which exists on its own or as part of another object can be identified as a reference object. This can also be the arc of an elliptical segment or a circular arc within a surface. The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

   The position at which the (part) object is identified (the "ID point) affects the way in which the rest of the command is carried out. If a horizontal tangent is placed on a circle or ellipse, the tangent can be placed above or below the circle or ellipse. The tangent point nearer to the ID point is used as the point through which the line should run (see graphic).



If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

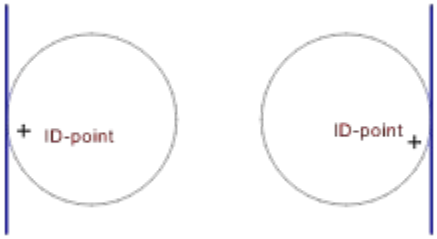
- Geometry Tangent>Object - Point
- Geometry Tangent>Object, Vertical
- Geometry Tangent>Angle - Object
- Geometry Tangent>Object - Object

Geometry Tangent>Object, Vertical (Geometry Menu)

This command draws a vertical geometry line which runs tangent to a circle or ellipse. "Vertical" relates to the position on the page and not to the view. If the drawing is rotated, then the line will be rotated correspondingly.

1. *Identify reference circle / ellipse*
   Any circle or ellipse which exists on its own or as part of another object can be identified as a reference object. This can also be the arc of an elliptical segment or a circular arc within a surface. The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

   The position at which the (part) object is identified (the "ID point) affects the way in which the rest of the command is carried out. If a vertical tangent is placed on a circle or ellipse, it can be placed at the right or left of the circle or ellipse. The tangent point nearer to the ID point is used as the point through which the line should run (see graphic).



If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

Geometry Tangent>Object - Point

Geometry Tangent>Object, Horizontal

Geometry Tangent>Angle - Object

Geometry Tangent>Object - Object

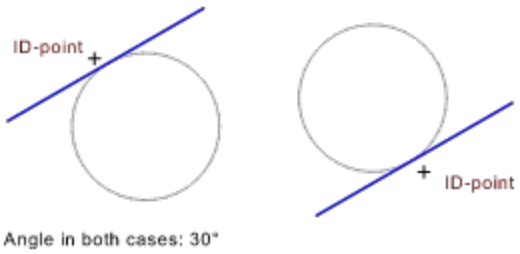
Geometry Tangent>Angle - Object (Geometry Menu)

This command draws a geometry line which is tangent to a given circle at a specified angle.

1. *Enter angle*
   After choosing this command, a dialog appears where a specific angle can be entered. This angle remains valid until either another command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify reference circle / ellipse*
   Any circle or ellipse which exists on its own or as part of another object can be identified as a reference object. This can also be the arc of an elliptical segment or a circular arc within a surface. The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

   The position at which the (part) object is identified (the "ID point) affects the way in which the rest of the command is carried out. If a tangent is to be placed at a specified angle to a circle or ellipse, it can be placed on either side of the circle or ellipse. The tangent whose tangent point is nearer to the ID point is used (see graphic).



If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

Geometry Tangent>Object - Point

Geometry Tangent>Object, Horizontal

Geometry Tangent>Object, Vertical

Geometry Tangent>Object - Object

Geometry Tangent>Object - Object (Geometry Menu)

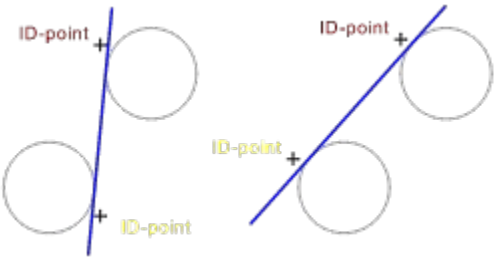
This command draws a geometry line which is tangent to two circles or ellipses.

1. *Identify reference circle / ellipse 1*
   Any circle or ellipse which exists independently or as part of another object can be identified as the first reference object. This can also be the arc of an ellipse segment or a circular arc within a surface. The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

   The position at which the (part) object is identified (the "ID point) affects the way in which the rest of the command is carried out. If a tangent is placed on two circles, there are up to four possible results. Together with the second ID point, this ID point determines which of the four possible tangents should be used (see below).

2. *Identify reference circle / ellipse 2*
   Any circle or ellipse which exists independently or as part of another object can be identified as the second reference object. This can also be the arc of an ellipse segment or a circular arc within a surface.
   The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

   The position at which the (part) object is identified (the "ID point) affects the way in which the rest of the command is carried out. If a tangent is placed on two circles, there are up to four possible results. Together with the first ID point, this ID point determines which of the four possible tangents should be used (see below).



If one circle or ellipse lies wholly within another, it is not possible to work out a line tangent to both objects. In this case, a corresponding error message appears.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple parallel lines will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the line is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

- Geometry Tangent>Object - Point
- Geometry Tangent>Object, Horizontal
- Geometry Tangent>Object, Vertical
- Geometry Tangent>Angle - Object

Geometry Circle>Standard (Geometry Menu)

This command draws a geometry circle. The geometry circle is specified by entering its center point and radius.

If a geometry circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Geometry Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter center point*
   The center point of the geometry cycle can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter radius*
   The radius of the geometry circle can be entered with the mouse by clicking anywhere in the drawing at a point through which the circle should run. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition it is possible to enter a specific radius. This gives the radius of the geometry circle.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the circle is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

- Geometry Circle>On Circle
- Geometry Circle>Circumcircle
- Geometry Circle>Diameter
- Geometry Circle>Incircle
- Geometry Circle>Concentric
- Geometry Circle>Tangential
- Geometry Circle>Line - Line - Line
- Geometry Circle>Radius - Point - Point
- Geometry Circle>Radius - Object - Point
- Geometry Circle>Radius - Object - Object

Geometry Circle>On Circle (Geometry Menu)

This command draws a geometry circle on top of an existing circle or circle part.

This command is particularly useful if a circle or circular arc which currently only exists in the drawing is to be used as a geometry circle.

If a geometry circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Geometry Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Identify reference circle*
   Any circle which exists on its own or within an existing object can be identified as a reference object. This can be the arc of a circular segment or a circular arc within a surface.
   The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the circle is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

- Geometry Circle>Standard
- Geometry Circle>Circumcircle
- Geometry Circle>Diameter
- Geometry Circle>Incircle
- Geometry Circle>Concentric
- Geometry Circle>Tangential
- Geometry Circle>Line - Line - Line
- Geometry Circle>Radius - Point - Point
- Geometry Circle>Radius - Object - Point
- Geometry Circle>Radius - Object - Object

Geometry Circle>Circumcircle (Geometry Menu)

This command draws a geometry circle. The geometry circle is determined by three points through which it should run. This is the same as calculating a triangle's circumcircle.

If a geometry circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Geometry Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter point 1 on circle*
   The first point through which the geometry circle should run can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point 2 on circle*
   The second point through which the geometry circle should run can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter point 3 on circle*
   The third point through which the geometry circle should run can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the circle is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

- Geometry Circle>Standard
- Geometry Circle>On Circle
- Geometry Circle>Diameter
- Geometry Circle>Incircle
- Geometry Circle>Concentric
- Geometry Circle>Tangential
- Geometry Circle>Line - Line - Line
- Geometry Circle>Radius - Point - Point
- Geometry Circle>Radius - Object - Point

Geometry Circle>Radius - Object - Object

Geometry Circle>Diameter (Geometry Menu)

This command draws a geometry circle. The geometry circle is specified by entering two points which are the ends of its diameter.

If a geometry circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Geometry Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter point 1 on circle*
   The first point through which the geometry circle should run can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point 2 on circle*
   The second point through which the geometry circle should run can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the circle is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

- Geometry Circle>Standard
- Geometry Circle>On Circle
- Geometry Circle>Circumcircle
- Geometry Circle>Incircle
- Geometry Circle>Concentric
- Geometry Circle>Tangential
- Geometry Circle>Line - Line - Line
- Geometry Circle>Radius - Point - Point
- Geometry Circle>Radius - Object - Point
- Geometry Circle>Radius - Object - Object

Geometry Circle>Incircle (Geometry Menu)

This command draws a geometry circle. The geometry circle is specified by entering the three corners of a triangle whose in circle it should be.

If the sides of the circle are known instead of its corner points, the command Geometry Circle>Line - Line - Line can be used instead.

If a geometry circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Geometry Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter corner point 1*
   The triangle's first corner point can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter corner point 2*
   The triangle's second corner point can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter corner point 3*
   The triangle's third corner point can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the circle is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

- Geometry Circle>Standard
- Geometry Circle>On Circle
- Geometry Circle>Circumcircle
- Geometry Circle>Diameter
- Geometry Circle>Concentric
- Geometry Circle>Tangential
- Geometry Circle>Line - Line - Line
- Geometry Circle>Radius - Point - Point
- Geometry Circle>Radius - Object - Point

Geometry Circle>Radius - Object - Object

Geometry Circle>Concentric (Geometry Menu)

This command draws a geometry circle which runs concentric to another circle or circle part.

If a geometry circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Geometry Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Identify reference circle*
   Any circle which exists on its own or within an existing object can be identified as a reference object. This can be the arc of a circular segment or a circular arc within a surface.
   The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

2. *Enter radius*
   After identifying the reference line, further points can be entered. Each point defines a concentric circle which runs though this point. This point can be entered with the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition it is possible to enter a specific radius. This gives the radius of the geometry circle. The radius can be entered relative to the radius of the reference circle.

   Concentric circle entry is ended by clicking the right mouse button. In contrast to most commands, clicking the right mouse button does not lead to the previous entry being ignored, but to all concentric circles being inserted. If this is not desired, the command Undo must be chosen next.

Up to 1000 concentric circles can be entered in one procedure. If this limit is reached, then no further entries are accepted.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the circle is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Point entry can be undone, step by step, using the command Previous Point (ESC) or the ESC key. This means that incorrect entries can easily be corrected, without having to redo all of the previously entered concentric circles.

Related commands:

Geometry Circle>Standard

Geometry Circle>On Circle

Geometry Circle>Circumcircle

Geometry Circle>Diameter

Geometry Circle>Incircle

Geometry Circle>Tangential

Geometry Circle>Line - Line - Line

Geometry Circle>Radius - Point - Point

Geometry Circle>Radius - Object - Point

Geometry Circle>Radius - Object - Object

Geometry Circle>Tangential (Geometry Menu) 

This command draws a geometry circle which has a given center point and is tangent to another line (or geometry line), circle (or geometry circle) or ellipse.

If a geometry circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Geometry Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Identify reference object*
   Any part object in an existing object can be identified as the reference object. This can be a side of a rectangle, the arc of an ellipse segment or a circular segment within a surface.
   The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

   The identified (part) object is referred to as the "Reference Object". After identification, a straight line appears on which the circle's center point will lie. This is known as the "Reference Straight Line".

2. *Enter reference point*
   The position of the circle and therefore the tangent point of the objects can be entered using the mouse, by clicking anywhere in the drawing. A perpendicular is dropped from this position to the reference object in order to work out the tangent point.

   If a circle is identified as the reference object, the position of the reference point determines which side of the reference circle the tangent point should lie. The intersection of the reference straight line with the reference cirle nearer to the reference point is used as the tangent point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If a circle is identified as the reference object, you can enter an angle. This determines the direction of the reference straight line.

3. *Enter center point*
   The center point of the circle can be entered using the mouse, by clicking anywhere in the drawing. A perpendicular is dropped from this point to the reference straight line to work out the center of the circle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition it is possible to enter a specific radius. This determines the radius of the circle.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the circle is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

 Geometry Circle>Standard

 Geometry Circle>On Circle

 Geometry Circle>Circumcircle


Geometry Circle>Diameter


Geometry Circle>Incircle


Geometry Circle>Concentric


Geometry Circle>Line - Line - Line


Geometry Circle>Radius - Point - Point


Geometry Circle>Radius - Object - Point


Geometry Circle>Radius - Object - Object

Geometry Circle>Line - Line - Line (Geometry Menu)

This command draws a geometry circle which is tangent to three lines or geometry lines.

If a geometry circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Geometry Ellipse>Distorted because the result on the page is not circular but elliptical.

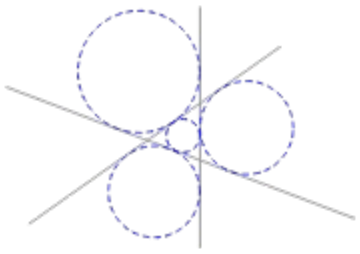
1. *Identify reference line 1*
   Any line in an existing object can be identified as the first reference line. This can also be a side of a rectangle or a part of a circular segment.
   The line is identified by clicking on it with the mouse. If more than one line is close to the position clicked on, a small dialog appears at the bottom of the screen with the help of which the correct line can be chosen.

   The position at which the line is identified (the "ID point") has no effect in this command. Because there are up to eight possible results, making a choice on the basis of the ID point's position would be to complicated.

2. *Identify reference line 2*
   Any line in an existing object can be identified as the second reference line. This can also be a side of a rectangle or a part of a circular segment.
   The line is identified by clicking on it with the mouse. If more than one line is close to the position clicked on, a small dialog appears at the bottom of the screen with the help of which the correct line can be chosen.

3. *Identify reference line 3*
   Any line in an existing object can be identified as the third reference line. This can also be a side of a rectangle or a part of a circular segment.
   The line is identified by clicking on it with the mouse. If more than one line is close to the position clicked on, a small dialog appears at the bottom of the screen with the help of which the correct line can be chosen.

4. *Choose object*
   After identifying all three reference lines, all the possible geometry circles are calculated (the graphic shows one possible set of results)



   They are presented so that a choice can be made. To do this, a small dialog appears at the bottom of the screen from which the correct geometry circle can be chosen. Only one circle can be chosen at any one time.

If all three reference lines are parallel to one another, it is not possible to calculate the geometry circles. In this case, a corresponding error message appears. It does however work if only two of the reference lines

are parallel to one another.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the circle is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

 Geometry Circle>Standard

 Geometry Circle>On Circle

 Geometry Circle>Circumcircle

 Geometry Circle>Diameter

 Geometry Circle>Incircle

 Geometry Circle>Concentric

 Geometry Circle>Tangential

 Geometry Circle>Radius - Point - Point

 Geometry Circle>Radius - Object - Point

 Geometry Circle>Radius - Object - Object

Geometry Circle>Radius - Point - Point (Geometry Menu)

This command draws a geometry circle, which has a given radius and runs through two points.

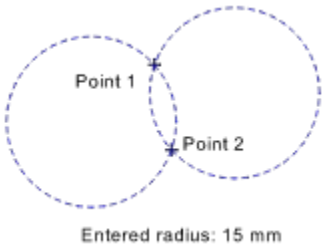
If a geometry circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Geometry Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter radius*
   After choosing the command, a dialog appears in which the geometry circle's radius can be specified. This radius remains valid until a new command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Enter point 1*
   The first point through which the geometry circle should run can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter point 2*
   The second point through which the geometry circle should run can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Choose object*
   After entering both points, all possible geometry circles are worked out (the graphic shows one possible set of results).



   They are presented so that a choice can be made. To do this, a small dialog appears at the bottom of the screen from which the correct geometry circle can be chosen. Only one circle can be chosen at any one time.

If the distance between the points is more than double the radius, it is not possible to calculate the geometry circles. In this case, a corresponding error message appears.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the circle is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

- Geometry Circle>Standard
- Geometry Circle>On Circle
- Geometry Circle>Circumcircle
- Geometry Circle>Diameter
- Geometry Circle>Incircle
- Geometry Circle>Concentric
- Geometry Circle>Tangential
- Geometry Circle>Line - Line - Line
- Geometry Circle>Radius - Object - Point
- Geometry Circle>Radius - Object - Object

Geometry Circle>Radius - Object - Point (Geometry Menu)

This command draws a geometry circle, which has a given radius, is tangent to a given line (or geometry line) or circle (or geometry circle) and runs through a given point.

If a geometry circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Geometry Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter radius*
   After choosing the command, a dialog appears in which the geometry circle's radius can be specified. This radius remains valid until a new command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

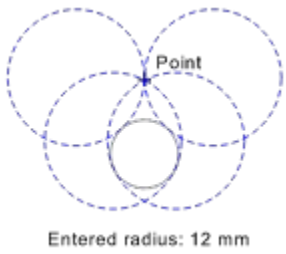
2. *Identify reference object*
   Any part object in an existing object can be identified as the reference object. This can be a side of a rectangle or a circular arc within a surface. Ellipses and ellipse parts cannot be identified.
   The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

   The identified (part) object is referred to as the "Reference Object".

   The position at which the object is identified (the "ID point") has no effect on the rest of this command. As there are up to four possible results, it would be to complicated to make a choice based on the ID point's position.

3. *Enter point*
   The point through which the geometry circle should run can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Choose object*
   After identifying the reference object and entering the point, all possible geometry circles are worked out (the graphic shows one possible set of results).



   They are presented so that a choice can be made. To do this, a small dialog appears at the bottom of the screen from which the correct geometry circle can be chosen. Only one circle can be chosen at any one time.

If the distance between the point and the object is more than twice the radius, it is not possible to calculate the geometry circles. In this case, a corresponding error message appears.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the circle is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

 Geometry Circle>Standard

 Geometry Circle>On Circle

 Geometry Circle>Circumcircle

 Geometry Circle>Diameter

 Geometry Circle>Incircle

 Geometry Circle>Concentric

 Geometry Circle>Tangential

 Geometry Circle>Line - Line - Line

 Geometry Circle>Radius - Point - Point

 Geometry Circle>Radius - Object - Object

Geometry Circle>Radius - Object - Object (Geometry Menu)

This command draws a geometry circle, which has a given radius and is tangent to two given lines (or geometry lines) or circles (or geometry circles).

If a geometry circle is to be drawn while working in isometric or dimetric views, this *must* be done with the command Geometry Ellipse>Distorted because the result on the page is not circular but elliptical.

1. *Enter radius*
   After choosing the command, a dialog appears in which the geometry circle's radius can be specified. This radius remains valid until a new command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify reference object 1*
   Any partial object in an existing object can be identified as the first reference object. This can be a side of a rectangle or a circular arc within a surface. Ellipses and ellipse parts cannot be identified. The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

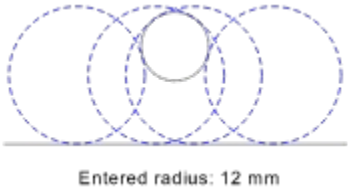
   The identified (part) object is referred to as the "Reference Object".

   The position at which the object is identified (the "ID point") has no effect on the rest of this command. As there are up to eight possible results, it would be to complicated to make a choice on the basis of the ID point's position.

3. *Identify reference object 2*
   Any partial object in an existing object can be identified as the second reference object. This can be a side of a rectangle or a circular arc within a surface. Ellipses and ellipse parts cannot be identified. The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

   The identified (part) object is referred to as the "Reference Object".

4. *Choose object*
   After identifying both reference objects, all possible geometry circles are worked out (the graphic shows one possible set of results).



   They are presented so that a choice can be made. To do this, a small dialog appears at the bottom of the screen from which the correct geometry circle can be chosen. Only one circle can be chosen at any one time.

If the distance between both objects is more than double the radius, or if two parallel lines are identified,

it is not possible to calculate the geometry circles. In this case, a corresponding error message appears.

If the multiline function is active during the final point's entry (for example if the CTRL key was held down), multiple concentric circles will be produced according to the current multiline settings. To alter the multiline settings, use the Change Parameters (+ESC) command.

If the geometry is not displayed when the circle is drawn, it will be displayed now. The geometry display can be toggled on and off using the command Display Geometry (F9), the F9 key or the corresponding button in the panel.

Related commands:

 Geometry Circle>Standard

 Geometry Circle>On Circle

 Geometry Circle>Circumcircle

 Geometry Circle>Diameter

 Geometry Circle>Incircle

 Geometry Circle>Concentric

 Geometry Circle>Tangential

 Geometry Circle>Line - Line - Line

 Geometry Circle>Radius - Point - Point

 Geometry Circle>Radius - Object - Point

Geometry Ellipse>Rectangular (Geometry Menu)

This command draws a rectangular geometry ellipse. "Rectangular" means here that both of the ellipse's half-axes are at right angles to one another. In addition, when using this command they are horizontal and vertical. Horizontal and vertical refer to the position relative to the page and not the view. If the drawing is rotated, then the half-axes will be rotated correspondingly.

1. *Enter center point*
   The center point of the ellipse can be entered with the mouse by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point on frame*
   The point on the frame surrounding the ellipse can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). By entering relative values, both of the ellipse's half-axes can be entered precisely.

A rectangular ellipse is stored as a distorted ellipse. This means that it can be rotated or distorted later.

If the geometry is not displayed when the circle is drawn, it will be displayed now.

The geometry can be displayed using the command Display Geometry (F9), or turned on/off with the F9 key or the corresponding button in the panel.

Related commands:

Geometry Ellipse>Rotated

Geometry Ellipse>Distorted

Geometry Ellipse>On Ellipse

Geometry Ellipse>Rotated (Geometry Menu) 

This command draws a rotated rectangular geometry ellipse. "Rectangular" means here that both of the ellipse's half-axes are at right angles to one another.

1. *Enter center point*
   The center point of the ellipse can be entered with the mouse by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point on frame*
   The point on the frame surrounding the ellipse can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). By entering relative values, both of the ellipse's half-axes can be entered precisely.

3. *Enter angle*
   The ellipse rotation angle can be entered with the mouse by clicking anywhere in the drawing. The position of this point relative to the center point determines the rotation angle. You can also enter an angle directly into a dialog. This determines the ellipse's rotation angle.

A rectangular ellipse is stored internally as a distorted ellipse. They can be further rotated or distorted later.

If the geometry is not displayed when the circle is drawn, it will be displayed now.

The geometry can be displayed using the command Display Geometry (F9), or turned on/off with the F9 key or the corresponding button in the panel.

Related commands:

 Geometry Ellipse>Rectangular

 Geometry Ellipse>Distorted

 Geometry Ellipse>On Ellipse

Geometry Ellipse>Distorted (Geometry Menu)

This command draws a distorted geometry ellipse. "Distorted" means here that both half-axes are entered separately and can therefore be at any angle to one another.

This command is particularly useful when, in isometric or dimetric view, a circle has to be entered. To do this, both of the ellipse's half-axes have to run in the respective main directions.

1. *Enter center point*
   The center point of the ellipse can be entered with the mouse by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter point 1 on ellipse*
   The first point through which the ellipse should run is the end point of the first half axis. It can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter point 2 on ellipse*
   The second point through which the ellipse should run is the end point of the second half axis. It can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the geometry is not displayed when the circle is drawn, it will be displayed now.

The geometry can be displayed using the command Display Geometry (F9), or turned on/off with the F9 key or the corresponding button in the panel.

Related commands:

Geometry Ellipse>Rectangular

Geometry Ellipse>Rotated

Geometry Ellipse>On Ellipse

Geometry Ellipse>On Ellipse (Geometry Menu)

This command draws a geometry ellipse based on another ellipse or a ellipse part.

This command is especially useful if an ellipse is to be transferred into a geometry ellipse. It can also be used to draw a geometry ellipse over an existing ellipse part.

1. *Identify reference ellipse*
   Any ellipse which exists independently or as part of another object can be identified as the reference object. This can also be the arc of an elliptical segment.
   Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

If the geometry is not displayed when the circle is drawn, it will be displayed now.

The geometry can be displayed using the command Display Geometry (F9), or turned on/off with the F9 key or the corresponding button in the panel.

Related commands:

Geometry Ellipse>Rectangular

Geometry Ellipse>Rotated

Geometry Ellipse>Distorted

Display Geometry (Geometry Menu)

This command can be used to switch the display of all geometry objects on and off.

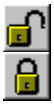
 The geometry display can also be changed by clicking the left mouse button on the button in the panel shown to the left. The button also shows the current status.

Related command:

 Freeze Geometry

Freeze Geometry (Geometry Menu)

This command can be used to freeze all geometry objects or to melt them again.



The geometry freezing can also be changed by clicking the left mouse button on the button in the panel shown to the left. The button also shows the current status.

Related command:

Display Geometry

Marking (Geometry Menu)

This command generates a marking. A marking is a point in the drawing marked with a marking symbol ( ). Markings can be used to save and recall particular positions.

1. *Enter position*
   The position for the marking can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The marking is assigned to a layer, like every other object, which can be frozen and/or blended out. Markings are not allocated to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer.

Markings are normally only displayed on the screen. They are ignored by output to a printer or plotter, to the Clipboard, a metafile or bitmap graphic.
If markings are to be included in output, the relevant check box must be enabled in the Settings>Output check box. The appearance of markings when output can be specified here.

Related commands:

 Division>Distance

 Division>Object

 Division>Array

Division>Distance (Geometry Menu)

This command can be used to divide up a distance according to various mathematical processes. A marking is placed between each division.

1. *Enter start point*
   The start point of the distance to be divided can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter end point*
   The end point of the distance to be divided can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After entering the points at either end of the length, the Division in which the division parameters can be specified. After entering the division parameters, markings are placed at all the calculated division points.

The generated markings are, like any other object, assigned to a layer which can be frozen and/or hidden. All the markings make up one object, i.e. they can only be edited or deleted together.
Markings are not allocated to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer.

Markings are normally only displayed on the screen. They are ignored by output to a printer or plotter, to the Clipboard, a metafile or bitmap graphic.
If markings are to be included in output, the relevant check box must be enabled in the Settings>Output check box. The appearance of markings when output can be specified here.

Related commands:


Marking


Division>Object


Division>Array

Division>Object (Geometry Menu)

This command divides the outline of an object according to a specified mathematical principle. A marking is inserted at the end of each division.

1. *Identify reference object*
   Any part object in an existing object can be identified as the reference object. This can be a side of a rectangle, the arc of an ellipse segment or a circular segment within a surface.
   The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

   The identified (part) object is referred to as the "Reference Object".

The outline of the object is divided up during division. In general, this is referred to as the "Distance"

Which point is interpreted as the start point depends on the reference object type. The rotation direction of the division is determined by the current arc direction. This is specified using the command Toggle Arc Direction (F2), with the F2 key or the button in the panel. If the arc direction is positive, the circular arc is divided beginning from the start angle in a mathematically positive sense (i.e. anticlockwise). If the arc direction is negative, the circular arc is divided beginning from the start angle in a mathematically negative sense (i.e. clockwise).

Ellipses and elliptical arcs are treated in a special way. As ellipses and elliptical arcs are mostly used in technical drawing to show circles in isometric or dimetric view, they are treated correspondingly. They are treated like circles during the division process.

After identifying the reference object, the Division dialog appears, in which the division parameters can be specified. After entering the division parameters markings are placed on all the division points calculated.

The generated markings are, like any other object, assigned to a layer which can be frozen and/or hidden. All the markings make up one object, i.e. they can only be edited or deleted together.
Markings are not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer.

Markings are normally only displayed on the screen. They are ignored by output to a printer or plotter, to the Clipboard, a metafile or bitmap graphic.
If markings are to be included in output, the relevant check box must be enabled in the Settings>Output check box. The appearance of markings when output can be specified here.

Related commands:

Marking

Division>Distance

Division>Array

Division>Array (Geometry Menu)

This command can be used to divide up a two-dimensional array using various mathematical procedures. A marking is inserted at each division point.

The array is entered in the form of a parallelogram. A parallelogram is a quadrangle whose opposite sides are parallel to one another. It can also be seen as a distorted rectangle.

1. *Enter corner point 1*
   The first corner point of the parallelogram can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter corner point 2*
   The second corner point of the parallelogram can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter corner point 3*
   The third corner point of the parallelogram can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The lengths Point 1-Point 2 and Point 1-Point 3 are used for the division process. In general, these are referred to as "Lengths".

After entering the three points the Division dialog appears twice, so that the division parameters for both lengths can be entered. After entering the division parameters, a marking is placed at each division point.

The generated markings are, like any other object, assigned to a layer which can be frozen and/or hidden. All the markings make up one object, i.e. they can only be edited or deleted together.
Markings are not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer.

Markings are normally only displayed on the screen. They are ignored by output to a printer or plotter, to the Clipboard, a metafile or bitmap graphic.
If markings are to be included in output, the relevant check box must be enabled in the Settings>Output check box. The appearance of markings when output can be specified here.

Related commands:


Marking


Division>Distance


Division>Object

Multiple Copy>Markings (Geometry Menu)

This command can be used to make multiple copies of selected groups of objects. The copies are inserted at the positions where there is a selected marking.

1. *Choose objects*
   The objects to be copied must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The reference point within the objects can be entered using the mouse, by clicking anywhere in the drawing. The reference point indicates exactly where markings should be placed. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After choosing the objects and entering the reference point, the chosen objects are copied to all currently selected markings. If there are no selected markings, an error message appears.

Related commands:

 Multiple Copy>Rotation

 Multiple Copy>Gradual

 Multiple Copy>Distance

 Multiple Copy>Object

 Multiple Copy>Array

Multiple Copy>Rotation (Geometry Menu)

This command can be used to make multiple copies of chosen groups of objects. The copies are arranged in a circle around a given center point. The objects are rotated during copying.

1. *Choose objects*
   The objects to be copied must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The reference point, i.e. the point at which the copies will be placed, can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter number of steps*
   After choosing the objects and entering the reference point, a dialog appears in which the number of steps to be used can be entered. Values between 2 and 250 are allowed. This number of steps is divided equally around the whole rotation.

The original objects are retained. If 10 steps are specified, nine extra copies of the object are created and distributed, with the original object, around a circle centered around the reference point.

Related commands:

 Multiple Copy>Markings

 Multiple Copy>Gradual

 Multiple Copy>Distance

 Multiple Copy>Object

 Multiple Copy>Array

Multiple Copy>Gradual (Geometry Menu)

This command can be used to make multiple copies of chosen groups of objects. The copies are each moved by a specified distance.

1. *Choose objects*
   The objects to be copied must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The copy's reference point, i.e. the start point of the step vector, can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter target point*
   The target point for the copy, i.e. the end point of the step vector, can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). The movement can be easily entered in this way by entering polar or relative coordinates.

3. *Enter number of steps*
   After choosing the objects and entering the reference and target points a dialog, in which the number of steps to be used can be entered, appears. Values between 1 and 250 are allowed.

The original objects are retained. If 10 steps were entered, 10 copies of the objects are made and moved in the direction of the given step vector.

Related commands:

 Multiple Copy>Markings

 Multiple Copy>Rotation

 Multiple Copy>Distance

 Multiple Copy>Object

 Multiple Copy>Array

Multiple Copy>Distance (Geometry Menu)

This command can be used to make multiple copies of chosen groups of objects. The copies are placed onto the division points of a given distance, worked out according to different mathematical processes.

1. *Choose objects*
   The objects to be copied must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The reference point within the objects can be entered using the mouse, by clicking anywhere in the drawing. The reference point indicates the place where the division point should be placed. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter start point*
   The start point of the distance to be divided can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter end point*
   The end point of the distance to be divided can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After choosing the objects and entering the distance's start and end points, the Division dialog appears, in which the division parameters can be specified.

After entering the division parameters the objects are copied to all the distance's division points.

Related commands:

 Multiple Copy>Markings

 Multiple Copy>Rotation

 Multiple Copy>Gradual

 Multiple Copy>Object

 Multiple Copy>Array

Multiple Copy>Object (Geometry Menu)

This command can be used to make multiple copies of chosen groups of objects. The copies are placed on the division points of an identified object, worked out by various mathematical processes.

1. *Choose objects*
   The objects to be copied must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The reference point within the objects can be entered using the mouse, by clicking anywhere in the drawing. The reference point indicates the place where the division point should be placed. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Identify reference object*
   Any part object in an existing object can be identified as the reference object. This can be a side of a rectangle, the arc of an ellipse segment or a circular segment within a surface.
   The (part) object is identified by clicking on it with the mouse. If several (part) objects lie close to the point clicked on, then a small dialog appears at the bottom of the screen with the help of which the correct (part) object can be chosen.

   The identified (part) object is referred to as the "Reference Object".

The outline of the identified object is divided up during the division. In general, this is referred to as the "Distance"

Which point is interpreted as the start point depends on the reference object type. The rotation direction of the division is determined by the current arc direction. This is specified using the command Toggle Arc Direction (F2), with the F2 key or the button in the panel. If the arc direction is positive, the circular arc is divided beginning from the start angle in a mathematically positive sense (i.e. anticlockwise). If the arc direction is negative, the circular arc is divided beginning from the start angle in a mathematically negative sense (i.e. clockwise).

Ellipses and elliptical arcs are treated in a special way. As ellipses and elliptical arcs are mostly used in technical drawings to show circles in isometric or dimetric view, they are treated correspondingly. They are treated like circles during the division process.

After choosing the objects and identifying the reference object, the Division dialog appears, in which the division parameters can be specified.

After entering the division parameters the objects are copied to all the reference object's division points (if need be they are rotated).

Related commands:

Multiple Copy>Markings

Multiple Copy>Rotation


Multiple Copy>Gradual


Multiple Copy>Distance


Multiple Copy>Array

Multiple Copy>Array (Geometry Menu)

This command can be used to make multiple copies of chosen groups of objects. The copies are placed on the division points of a two-dimensional array, which can be calculated according to various mathematical processes.

The arrray is entered in the form of a parallelogram. A parallelogram is a quadrangle whose opposite sides are parallel to one another. It can also be seen as a distorted rectangle.

1. *Choose objects*
   The objects to be copied must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The reference point within the objects can be entered using the mouse, by clicking anywhere in the drawing. The reference point indicates the place where the division point should be placed. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter corner point 1*
   The first corner point of the parallelogram can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

4. *Enter corner point 2*
   The second corner point of the parallelogram can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

5. *Enter corner point 3*
   The third corner point of the parallelogram can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The lengths Point 1 - Point 2 and Point 1 - Point 3 are used in the procedure. These are generally referred to as "Distances".

After choosing the objects and entering the reference point and the three points, the Division dialog appears in which the division parameters for each of the two lengths can be specified.

After entering the division parameters, the objects are copied to all the calculated division points in the array.

Related commands:

 Multiple Copy>Markings

 Multiple Copy>Rotation

 Multiple Copy>Gradual

 Multiple Copy>Distance

 Multiple Copy>Object

Trimming Menu

- Trim Object
  - Cut Out
  - Split
  - Resolve Completely
  - Length / Radius to Point
  - Length / Radius to Object
  - Angle to Point
  - Angle to Object
- Trim Curve
  - Edit
  - Extend
  - Split
- Trim Surface
  - Generate Surface
  - Union (A=A+B)
  - Intersection (A=A×B)
  - Difference (A=A-B)
  - Combine Outlines
  - Split into Outlines
- Trim Edge
- Chamfer Edge
  - Object - Object
  - One Edge
  - Complete
- Round Edge, Outer Arc
  - Object - Object
  - One Edge
  - Complete
- Round Edge, Inner Arc
  - Object - Object
  - One Edge
  - Complete
- Transform Object to
  - Invert
  - Zigzag Line
  - Circle
  - Circular Arc
  - Circular Sector
  - Circular Segment
  - Ellipse
  - Elliptic Arc
  - Elliptic Sector
  - Elliptic Segment
  - Curve
  - Surface

Clipping
- Create, Inside
- Create, Outside
- Resolve

Trim Object>Cut Out (Trimming Menu)

This command can be used to cut a piece from an existing object. Where possible, the end of an object can be capped.

1. *Identify the object to be trimmed*
   Any standard object can be identified as the object to be trimmed. Lines, circles, circle parts, ellipses and ellipse parts. The objects are always treated as wholes; sectors and segments are handled like arcs.
   To identify an object, click on it with the mouse. If the identification is ambiguous, i.e. if more than one object is very close to the position clicked on, then a dialog with whose help the correct object can be chosen appears at the bottom of the screen.

   The identified object is referred to in the rest of this section as the "trim object".

   The position at which the trim object is identified (the "ID Point") affects the rest of the command. It determines which part of the object should be trimmed or capped. Refer to the graphics at the end of this section.

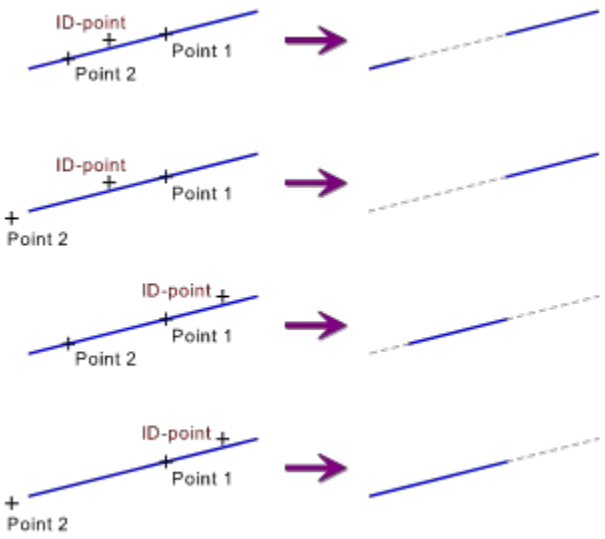
2. *Enter point 1*
   The first separation point can be specified by clicking on any point within the drawing. A perpendicular is dropped from here to the trim object to work out the exact separation point. In most cases it is a good idea to snap to the intersection of the trim object with another object. To do this, hold down the SHIFT key during point entry. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If the reference object is a circle, circle part, ellipse or ellipse part then it is possible to specify an angle, measured from the midpoint of the trim object.

3. *Enter point 2*
   The second separation point can be specified by clicking on any point within the drawing. A perpendicular is dropped from here to the trim object to work out the exact separation point. In most cases it is a good idea to snap to the intersection of the trim object with another object. To do this, hold down the SHIFT key during point entry. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If the trim object is a line, then you can specify a length. This is the length of the line from a previously-specified starting point. In all other cases, it is possible to specify an angle, measured from the midpoint of the trim object.

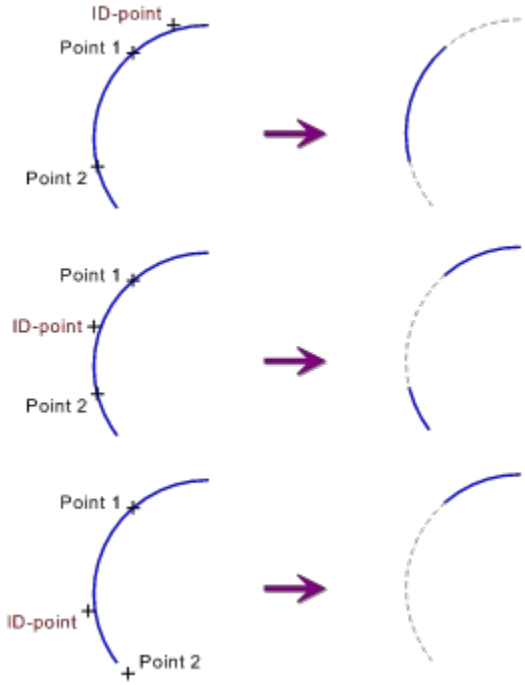
If the duplicate function is on while the last point is specified (e.g. if the CTRL key is held down), the trim object remains unchanged; a duplicate is generated and trimmed.

After specifying both separation points, the trim object is trimmed. If both separation points lie on the object outline, then a portion is removed or both outstanding ends are separated. If a one separation point is outside the outline, then one end of the object is capped. The segment nearest to the ID point is always removed (see diagram).

Trimming lines:



Trimming circle and ellipse parts.



Note a basic rule for trimming: the object is always changed where it is clicked on!

If both points lie outside the object outline, an error message appears.

If a non-standard object (e.g. a surface or a curve) is to be trimmed, it must be resolved to its constituent parts first using the command Trim Object>Resolve Completely. These can then be trimmed.

Related commands:

- Trim Object>Split
- Trim Object>Length / Radius to Point
- Trim Object>Length / Radius to Object
- Trim Edge

Trim Object>Split (Trimming Menu)

This command splits an existing object into two parts.

1. *Identify the object to be trimmed*
   Any standard object can be identified as the object to be trimmed. Lines, circles, circle parts, ellipses and ellipse parts. The objects are always treated as wholes; sectors and segments are handled like arcs.
   To identify an object, click on it with the mouse. If the identification is ambiguous, i.e. if more than one object is very close to the position clicked on, then a dialog with whose help the correct object can be chosen appears at the bottom of the screen.

   The identified object is referred to in the rest of this section as the "trim object".

2. *Enter point 1*
   The first separation point can be specified by clicking on any point within the drawing. A perpendicular is dropped from here to the trim object to work out the exact separation point. In most cases it is a good idea to snap to the intersection of the trim object with another object. To do this, hold down the SHIFT key during point entry. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If the reference object is a circle, circle part, ellipse or ellipse part then it is possible to specify an angle, measured from the midpoint of the trim object.

If the trim object is a circle or ellipse, a second separation point has to be specified. In other cases, this is not necessary.

3. *Enter point 2*
   The second separation point can be specified by clicking on any point within the drawing. A perpendicular is dropped from here to the trim object to work out the exact separation point. In most cases it is a good idea to snap to the intersection of the trim object with another object. To do this, hold down the SHIFT key during point entry. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If the reference object is a circle, circle part, ellipse or ellipse part then it is possible to specify an angle, measured from the midpoint of the trim object.

If the duplicate function is on while the last point is specified (e.g. if the CTRL key is held down), the trim object remains unchanged; a duplicate is generated and trimmed.

After specifying the separation points, the trim object is split. If the separation point lies outside the object outline, then an error message appears.

If a non-standard object (e.g. a surface or a curve) is to be trimmed, it must be resolved to its constituent parts first using the command Trim Object>Resolve Completely. These can then be trimmed.

Related commands:

Trim Object>Cut Out

Trim Object>Resolve Completely

Trim Curve>Split

Trim Edge

Trim Object>Resolve Completely (Trimming Menu)

This command breaks existing objects into their constituent basic elements. These basic elements are lines, circles and circular arcs.

This command is useful if complicated objects are to be trimmed. As the trimming functions only work with standard objects, objects such as curves, surfaces and hatchings have to be resolved into basic elements first.

1. *Choose objects*
   The objects which are to be broken up are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

If the duplicate function is active during object choosing, a copy of the chosen objects is separated.

If a text is to be split up, they must be transformed to surfaces first using the command Text>Resolve. These can then be split up using the commands described here. The same goes for groups and instances, which must also be resolved first.

Related commands:

 Trim Object>Cut Out

 Trim Object>Split

 Trim Curve>Split

 Group>Ungroup

 Text>Resolve

 Resolve Instances

Trim Object>Length / Radius to Point (Trimming Menu)

This command can be used to modify an existing object. If the object is a line, then its length can be changed. If, however, it is a circle or circle part, then its radius can be altered. The size of the alteration is determined by entering a point.

1. *Identify the object to be trimmed*
   Any standard object can be identified and trimmed. In this case, standard objects are lines, circles and circle parts. The objects are always treated as a whole, sectors and segments are treated like arcs. Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

   The identified object is referred to as the "trim object"

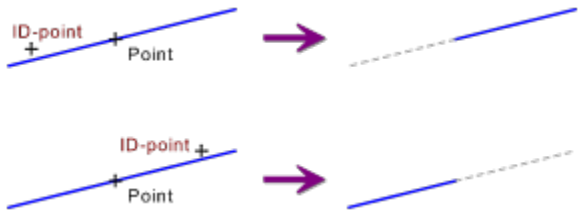
   The point at which the line was identified (the ID point) affects the rest of the command. It determines which end of the line should be moved. Refer to the graphics at the end of this section.

   If the trim object is a line, a straight line appears, extending the line. It is called the "reference line", and can be used to snap to its intersections with other objects.

2. *Enter point*
   The point to which the length or radius should be altered, can be entered using the mouse, by clicking anywhere in the drawing. In most cases, it makes sense to use the intersection of the trim object or reference straight line with other objects as a snapping point. To do this, the SHIFT key is kept held down during point entry. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If the trim object is a line, then a length can be entered. The length can be entered relative to the current line length. It determines the resulting line length. The end point of the line nearest to the ID point is then moved until this length is reached. If on the other hand the trim object is a circle or circle part you can enter a radius. The radius can be entered relative to the current circle or circle part"s radius. It determines the resulting radius of the trim object.

If the duplicate function is active during the entry of the last point (e.g. because the CTRL key is held down), the trim object itself is not trimmed, but a copy is made and the copy is trimmed.

After entering the point, the trim object is trimmed. If the trim object is a line, the end point of the line nearest to the ID point is moved (see graphic).



Note this basic rule for trimming: the object is always changed where it is clicked on.

If a non-standard object (e.g. a curve or surface) is to be trimmed, then the command Trim Object>Resolve Completely must be used to break it into its constituent parts. These can then be trimmed.

Related commands:

- Trim Object>Cut Out
- Trim Object>Split
- Trim Object>Length / Radius to Object
- Trim Object>Angle to Point
- Trim Object>Angle to Object
- Trim Edge

Trim Object>Length / Radius to Object (Trimming Menu)

This command can be used to modify an existing object. If it is a line, the length of the line can be changed. If, however, it is a circle or circle part, then its radius can be altered. The size of the change is determined by the intersection or tangent with a given object

1. *Identify the object to be trimmed*
   Any standard object can be identified and trimmed. In this case, standard objects are lines, circles and circle parts. The objects are always treated as a whole, sectors and segments are treated like arcs. Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

   The identified object is referred to as the "trim object"

   If the trim object is a line, then the position at which the line was identified (the "ID point"), affects the rest of the command. It determines which end of the line should be moved. Refer to the graphics at the end of this section.

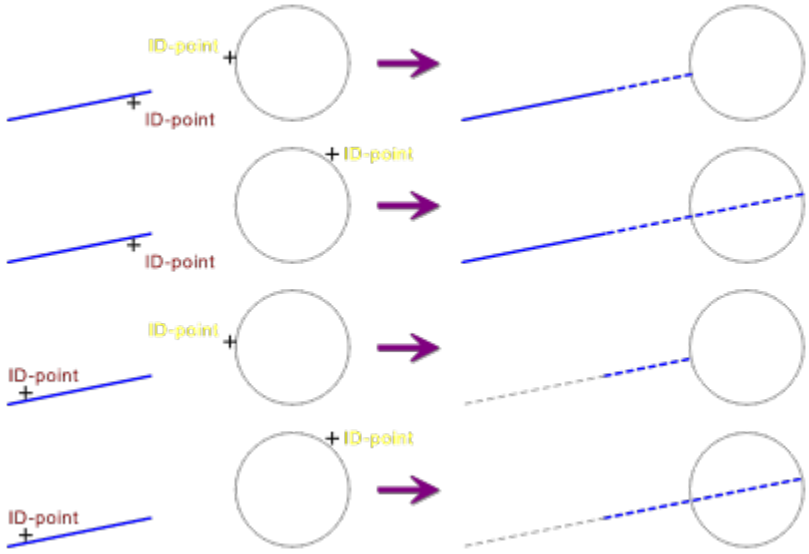
2. *Identify reference object*
   Any part object in an existing object can be identified as a reference object. This may be an edge of a rectangle, the arc of an elliptic segment or a circular arc within a surface.
   To identify it, click on the (part) object with the mouse. If more than one possible (part) object lies close to the point clicked on, then a small dialog appears at the bottom of the screen from which the "right" object can be chosen.

   The identified object is referred to as the "reference object"

   The position at which the (part) object was identified (the "ID point"), affects the rest of the command. If the trim object has several possible intersections or tangent points with the reference object, the point closer to the ID point is chosen.

If the duplicate function is active while the reference object is selected (e.g. because the CTRL key was held down), the trim object itself is not trimmed, but a copy is generated and the copy is trimmed.

After entering the point, the trim object is trimmed. If the trim object is a line, the end point of the line nearest to the ID point is moved (see graphic).



Note a basic rule for trimming: The object changes, wherever it is clicked on.

If a non-standard object (e.g. a curve or surface) is to be trimmed, then the command Trim Object>Resolve Completely must be used to break it into its constituent parts. These can then be trimmed.

Related commands:

- Trim Object>Cut Out
- Trim Object>Split
- Trim Object>Length / Radius to Point
- Trim Object>Angle to Point
- Trim Object>Angle to Object
- Trim Edge

Trim Object>Angle to Point (Trimming Menu)

This command can be used to modify an existing object. If the object is a line, the line direction is changed. If on the other hand it is a circle part, the start or end angle is changed. The size of the alteration is determined by entering a point.

1. *Identify the object to be trimmed*
   Any standard object can be identified and trimmed. Standard objects are in this case lines, circle parts and ellipse parts. The objects are always treated as a whole, sectors and segments are treated like arcs. Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

   The identified object is referred to as the "trim object"

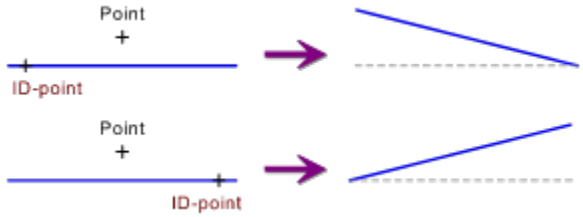
   The position at which the object is identified (the "ID point"), affects the rest of the command. It determines which end point of the line or which arc angle should be altered.

   After identification, a circle or ellipse (the "reference object") appears. This indicates the possible new end point positions.

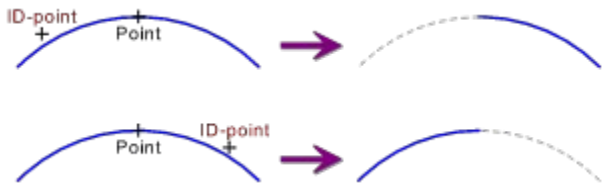
2. *Enter point*
   The point to which the angle should be changed, can be entered using the mouse, by clicking anywhere in the drawing. A perpendicular is dropped from this point to the reference object. In most cases it makes sense to use the intersection of the reference object with another object as a snapping reference. To do this, the SHIFT key is kept held down during point entry. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, an angle can be entered. The angle can be entered relative to the current angle. It determines the resulting trim object angle.

If the duplicate function is active during the entry of the last point (e.g. because the CTRL key is held down), the trim object itself is not trimmed, but a copy is generated and the copy is trimmed.

After entering the point, the trim object is trimmed. If the trim object is a line, the end point of the line nearest to the ID point is moved (see graphic).



If the trim object is a circle or ellipse part, then the angle at the end of the arc closer to the ID point is altered (see graphic).



Note a basic rule for trimming: The object changes, wherever it is clicked on.

If a non-standard object (e.g. a curve or surface) is to be trimmed, then the command Trim Object>Resolve Completely must be used to break it into its constituent parts. These can then be trimmed.

Related commands:

- Trim Object>Cut Out
- Trim Object>Split
- Trim Object>Length / Radius to Point
- Trim Object>Length / Radius to Object
- Trim Object>Angle to Object

Trim Object>Angle to Object (Trimming Menu)

This command can be used to modify an existing object. If it is a line, the line direction is altered. If however it is a circle or ellipse part, then the start or end angle is changed. The size of the change is given by the intersection or tangent point with a given object.

1. *Identify the object to be trimmed*
   Any standard object can be identified and trimmed. Standard objects are in this case lines, circle parts and ellipse parts. The objects are always treated as a whole, sectors and segments are treated like arcs. Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

   The identified object is referred to as the "trim object"

   The position at which the object is identified (the "ID point"), affects the rest of the command. It determines which end point of the line or which arc angle should be altered.

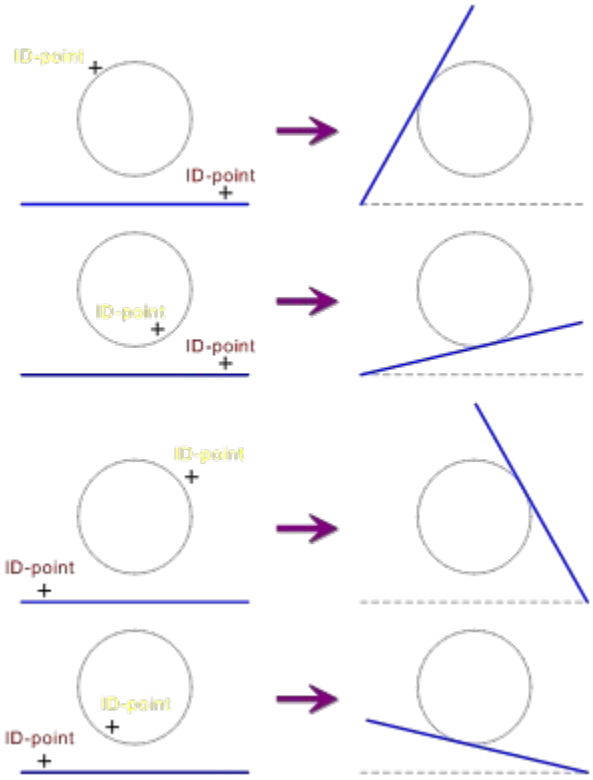
2. *Identify reference object*
   Any part object in an existing object can be identified as a reference object. This may be an edge of a rectangle, the arc of an elliptic segment or a circular arc within a surface.
   To identify it, click on the (part) object with the mouse. If more than one possible (part) object lies close to the point clicked on, then a small dialog appears at the bottom of the screen from which the "right" object can be chosen.

   The identified object is referred to as the "reference object"

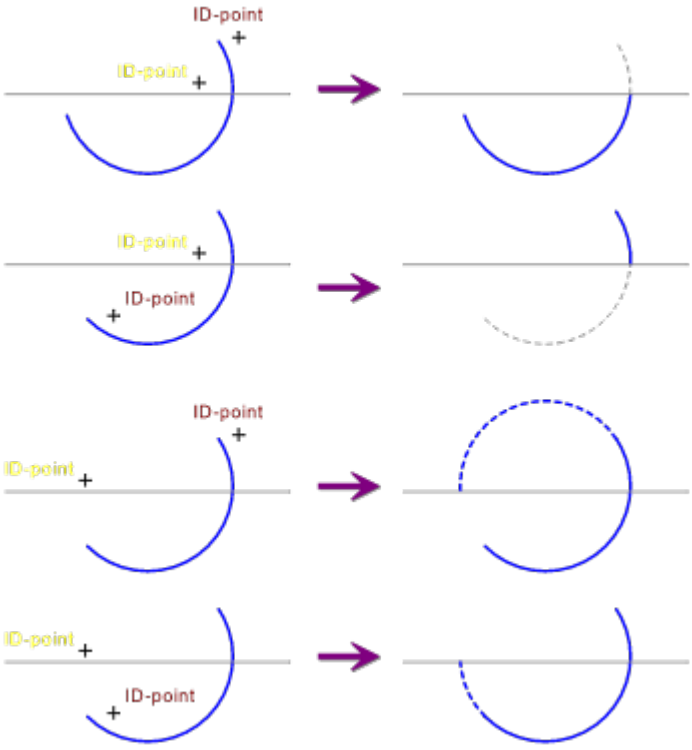
   The position at which the (part) object was identified (the "ID point"), affects the rest of the command. If the trim object has several possible intersections or tangent points with the reference object, the point closer to the ID point is chosen.

If the duplicate function is active while the reference object is selected (e.g. because the CTRL key was held down), the trim object itself is not trimmed, but a copy is made and the copy is trimmed.

After entering the point, the trim object is trimmed. If the trim object is a line, the end point of the line nearest to the ID point is moved (see graphic).



If the trim object is a circle or ellipse part, then the arc end angle closer to the ID point is changed (see graphic).



Note a basic rule for trimming: The object changes, wherever it is clicked on.

If a non-standard object (e.g. a curve or surface) is to be trimmed, then the command Trim Object>Resolve Completely must be used to break it into its constituent parts. These can then be trimmed.

Related commands:


Trim Object>Cut Out


Trim Object>Split


Trim Object>Length / Radius to Point


Trim Object>Length / Radius to Object


Trim Object>Angle to Point

Trim Curve>Edit (Trimming Menu)

An existing curve can be edited using this command. Elements can be inserted, transformed or deleted.

If a curve is to be extended from one of its ends to join an existing element, this can be done with the command Trim Curve>Extend. To divide a curve into two points, the command Trim Curve>Split can be used

Single points on a curve (End points, corner points and angel points) can be moved using the command Move Single Point.

1. *Identify curve*
   To identify it, click on the curve with the mouse. If several curves lie close to the position clicked on, a small dialog appears at the bottom of the window, with the help of which the "right" curve can be chosen.

   The position at which the curve is identified (the "ID point"), affects the rest of the command. It determines which end of the curve should be edited or at which point an element should be inserted.

After identifying the curve, the Edit Curve dialog appears, in which the action can be chosen which will be carried out with the element or joint of the curve which is nearest to the ID point. The further progress of the entry depends on which button is chosen in this dialog.

If the duplicate function is active during selection, a copy of the curve is generated and the copy, not the original, is altered.

Polylines and freehand lines are stored internally as curves and can therefore be manipulated with this command. All other objects must, if necessary, be converted to a curve using the command Transform Objects to>Curve.

Related commands:

Trim Curve>Extend

Trim Curve>Split

Trim Curve>Extend (Trimming Menu)

An existing curve can be extended to another, existing, object using this command.

If a curve is to be split into two parts, then the command Trim Curve>Split can be used. With the command Trim Curve>Edit individual elements of a curve can be altered or deleted.

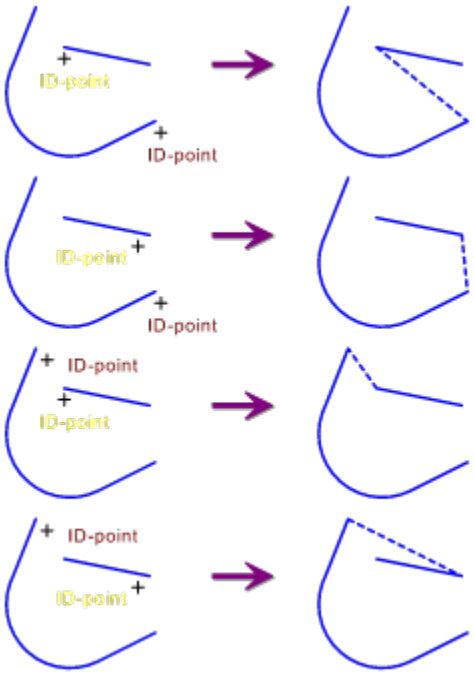
1. *Identify curve*
   To identify it, click on the curve with the mouse. If several curves lie close to the position clicked on, a small dialog appears at the bottom of the window, with the help of which the "right" curve can be chosen.

   The position at which the curve is identified (the "ID point"), affects the rest of the command. It determines which end of the curve the object identified next should be attached to.

2. *Identify object*
   The object to which the curve should be extended can be a line, a circular arc or another curve. To identify the object, click on it with the mouse. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

   The position at which the object is identified (the "ID point"), affects the rest of the command. It determines which end of the object the previously identified curve should be extended to.

After identifying both objects, the extended curve is generated. The endpoints nearer to the ID point of the two objects previously identified are joined by a line (see graphic).



If the duplicate function was active while selecting the second object (e.g. because the CTRL key was held down), the identified objects remain intact. Otherwise they are deleted and only the new, extended curve remains.

If, for example, two lines are to be joined, then one of them must first be transformed to a curve using the command Transform Objects to>Curve. This can then be extended to the second line. The same goes for circular arcs and the combinations of lines and circular arcs. Polylines are already curves and can be used directly.

Related commands:

Trim Curve>Edit

Trim Curve>Split

Trim Curve>Split (Trimming Menu)

This command can be used to split an existing curve at one of its junctions.

If a curve is to be extended from one of its ends to join an existing element, this can be done with the command Trim Curve>Extend. The command Trim Curve>Edit can be used to alter or delete individual elements of a curve.

1. *Identify curve*
   To identify it, click on the curve with the mouse. If several curves lie close to the position clicked on, a small dialog appears at the bottom of the window, with the help of which the "right" curve can be chosen.

   The position at which the curve is identified (the "ID point"), affects the rest of the command. It determines which end of the curve the object identified next should be attached to.

If the duplicate function was active when the curve was selected (e.g. because the CTRL key was held down), a copy of the curve is made and the copy is cut up, the original curve remains intact.

Polylines are stored internally as curves and can be directly manipulated with this command. All other objects must, if necessary, be converted to curves using the command Transform Objects to>Curve.

Related commands:

- Trim Curve>Edit
- Trim Curve>Extend
- Trim Object>Cut Out
- Trim Object>Split
- Trim Object>Resolve Completely

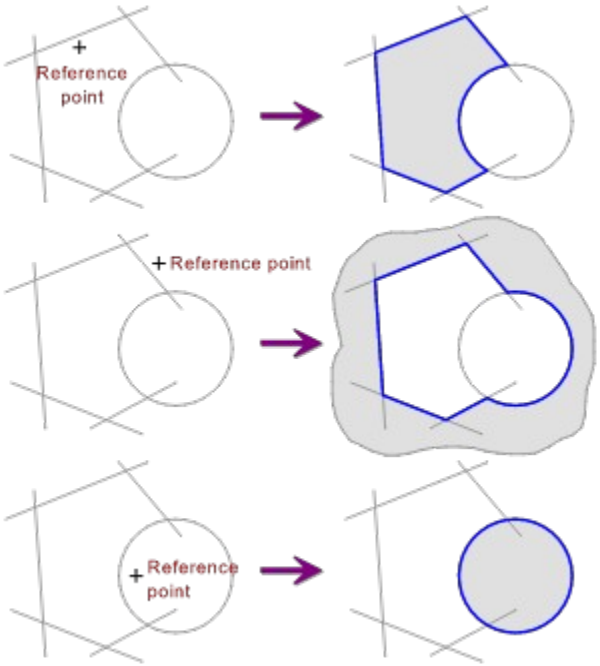
Trim Surface>Generate Surface (Trimming Menu)

This command can be used to generate a surface from individual outline elements. This is particularly useful to hatch areas which are not a surface, but are made up of the space between other objects.

1. *Choose objects*
   The objects whose outlines are to be included in the new surface are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

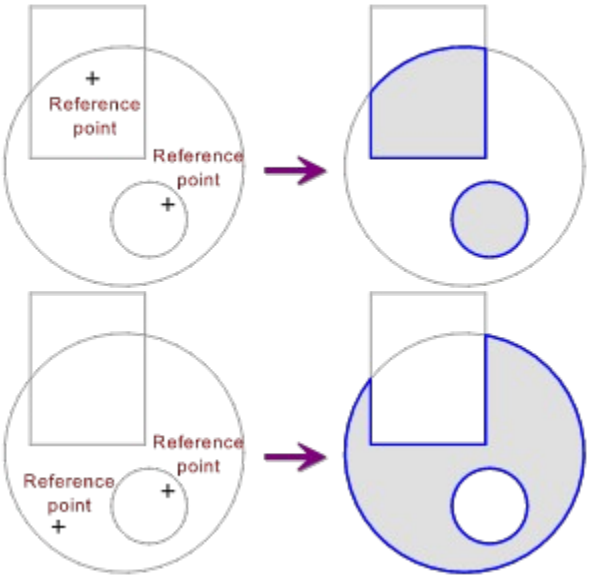
2. *Enter reference point*
   After choosing the objects, several reference points can be entered, one after another. These points can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

   The position of the reference points determines where and how the surface should be generated. Each reference point should be close to one of the outline elements to be incorporated in the surface. It shows with which element the search for a closed outline should begin. Through its position relative to this element, it also determines whether the "largest possible" or "smallest possible" surface should be worked out (see graphic).



   The effect of the reference point can be explained (simply) as follows: Imagine emptying a can of paint at the reference point. The paint spreads out in all directions until it reaches an object.
   The result is one area which is covered in paint, and one area which is not. The boundary between these two areas is the outline of the newly-generated surface.

   If multiple reference points are entered, one outline will be calculated for each of those reference points accordings to the method described above. The resulting outlines will then be combined to one surface. This way it's easy to generate complex surface (see graphic).



Reference point input is ended by clicking the right mouse button. Unlike most other commands, this does not lead to the previous entry being ignored; instead the command is processed with all previously entered reference points. If required, this can be undone using the Undo command.

After entering the reference points, the surface is generated and inserted. The result can be influenced by the parameters specified with the command Change Parameters (+ESC). If an error message appears although it ought to be possible to generate the surface, try increasing the tolerance.

Generated surfaces are not automatically allocated to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for the command Pens>Defaults and pen assignments.

As the result is a surface, only its elements (lines, circular arcs and Bézier curves) can be used when calculating the result. Many object types (above all, ellipses and ellipse parts) must be converted to polylines first, which inevitably causes loss of quality. The accuracy of the conversion can be specified using the command Settings>Screen under the heading "Accuracy - Transform".
The select objects, on the basis of which the surface is generated are neither deleted nor altered. If they are to be deleted, the easiest way is to use the command Delete Objects or press the DELETE key. The required selection can easily be made by pressing the F11 Key. This deletes exactly the objects used to generate the surface.

Related commands:

Trim Surface>Union (A=A+B)

Trim Surface>Intersection (A=A×B)

Trim Surface>Difference (A=A-B)

Trim Surface>Combine Outlines

Trim Surface>Split into Outlines

Trim Surface>Union (A=A+B) (Trimming Menu)

Using this command existing flat objects can be combined into one surface which covers all areas of the drawing which were covered by at least one of the surfaces. This is the same as creating a union of two surfaces according to group theory.

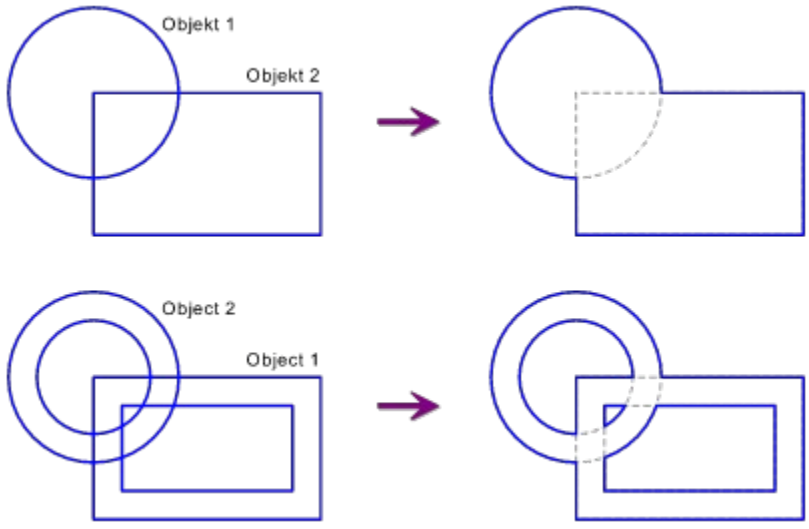
The following object types are classed as "flat": circles, circular sectors, circular segments, ellipses, elliptical sectors, elliptical segments and surfaces. Ellipses and ellipse parts are converted into polylines and are therefore "only" contained as polylines in the result.

1. *Identify areal object 1*
   The first object is identified by clicking on it with the mouse. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

2. *Identify areal object 2*
   The second object is identified by clicking on it with the mouse. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active while selecting the second object (e.g. because the CTRL key was held down), the original objects remain intact. Otherwise they are deleted.

The result in each case is one "surface" object, which may however be made up of more than one surface part. A graphic with examples of the union follows:



The order in which the objects were identified does not influence the result.

This command works with all objects which have a closed outline. These are automatically converted to surfaces for the calculation.
Many object types can not be converted into a surface without loss. Ellipses and ellipse parts must be converted into polylines, which invariably leads to a loss of quality. The accuracy of the conversion can

be specified using the command Settings>Screen under the heading "Accuracy - Transform".
The original objects (which remain unchanged if the duplicate function is active), are not influenced by this conversion. Surfaces with "holes" can also be used.
There can be problems if both objects share (part) of an edge. In this case there can, in spite of calculation to 16 decimal places, be incorrect interpretations. Do not rely on overlapping surfaces *always* or *never* being combined. This varies from case to case. The result may consist of one object which contains both the original surfaces, unaltered, as part surfaces.

Related commands:


Trim Surface>Intersection (A=A×B)


Trim Surface>Difference(A=A-B)


Trim Surface>Combine Outlines


Trim Surface>Split into Outlines


Trim Surface>Generate Surface

Trim Surface>Intersection (A=A×B) (Trimming Menu)

This command can be used to combine existing flat objects into one surface which covers all the parts of the drawing previously covered by both objects. This is the same as the intersection of two objects according to the group theory.

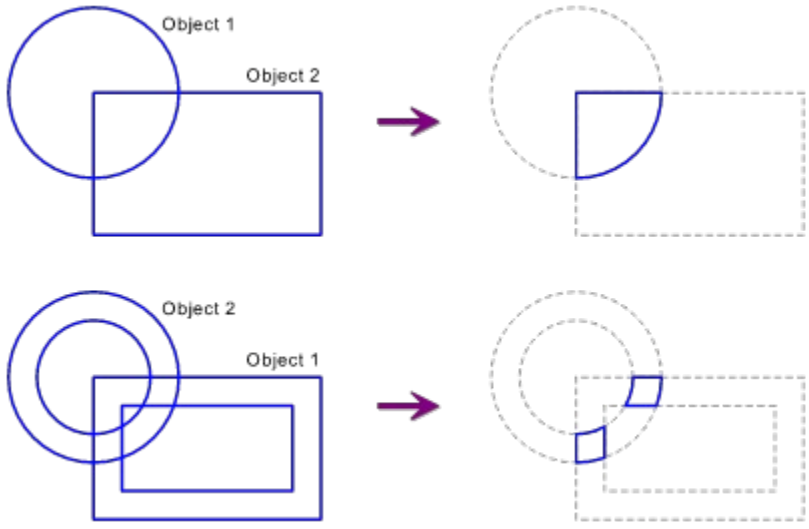
The following object types are classed as "flat": circles, circular sectors, circular segments, ellipses, elliptical sectors, elliptical segments and surfaces. Ellipses and ellipse parts are converted into polylines and are therefore "only" contained as polylines in the result.

1. *Identify areal object 1*
   The first object is identified by clicking on it with the mouse. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

2. *Identify areal object 2*
   The second object is identified by clicking on it with the mouse. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active while selecting the second object (e.g. because the CTRL key was held down), the original objects remain intact. Otherwise they are deleted.

The result in each case is one "surface" object, which may however be made up of more than one surface part. There follows a graphic showing examples of intersections:



The order in which the objects were identified does not influence the result.

This command works with all objects which have a closed outline. These are automatically converted to surfaces for the calculation.
Many object types can not be converted into a surface without loss. Ellipses and ellipse parts must be converted into polylines, which invariably leads to a loss of quality. The accuracy of the conversion can

be specified using the command Settings>Screen under the heading "Accuracy - Transform".
The original objects (which remain unchanged if the duplicate function is active), are not influenced by this conversion. Surfaces with "holes" can also be used.
There can be problems if both objects share (part) of an edge. In this case there can, in spite of calculation to 16 decimal places, be incorrect interpretations. Do not rely on the sides of overlapping objects always or never remaining unaltered. This varies from case to case.

Related commands:


Trim Surface>Union (A=A+B)


Trim Surface>Difference (A=A-B)


Trim Surface>Combine Outlines


Trim Surface>Split into Outlines


Trim Surface>Generate Surface

Trim Surface>Difference (A=A-B) (Trimming Menu)

This command can be used to combine existing flat objects into one surface which covers all parts of the drawing previously covered only by the first object and not by the second object. This is the same as the difference of two surfaces according to the group theory.

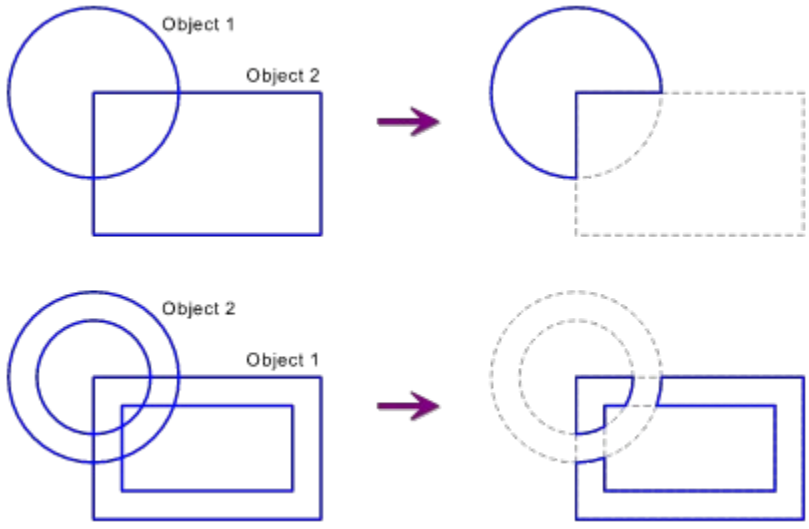
The following object types are classed as "flat": circles, circular sectors, circular segments, ellipses, elliptical sectors, elliptical segments and surfaces. Ellipses and ellipse parts are converted into polylines and are therefore "only" contained as polylines in the result.

1. *Identify areal object 1*
   The first object is identified by clicking on it with the mouse. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

2. *Identify areal object 2*
   The second object is identified by clicking on it with the mouse. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active while selecting the second object (e.g. because the CTRL key was held down), the original objects remain intact. Otherwise they are deleted.

The result in each case is one "surface" object, which may however be made up of more than one surface part. There follows a graphic showing examples of differences::



Note that the order in which the objects are identified is important. The last identified object is always "subtracted" from the object identified first.

This command works with all objects which have a closed outline. These are automatically converted to surfaces for the calculation.
Many object types can not be converted into a surface without loss. Ellipses and ellipse parts must be

converted into polylines, which invariably leads to a loss of quality. The accuracy of the conversion can be specified using the command Settings>Screen under the heading "Accuracy - Transform".
The original objects (which remain unchanged if the duplicate function is active), are not influenced by this conversion. Surfaces with "holes" can also be used.
There can be problems if both objects share (part) of an edge. In this case there can, in spite of calculation to 16 decimal places, be incorrect interpretations. Do not rely on the sides of overlapping objects always or never remaining unaltered. This varies from case to case.

Related commands:


Trim Surface>Union (A=A+B)


Trim Surface>Intersection (A=A×B)


Trim Surface>Combine Outlines


Trim Surface>Split into Outlines


Trim Surface>Generate Surface

Trim Surface>Combine Outlines (Trimming Menu)

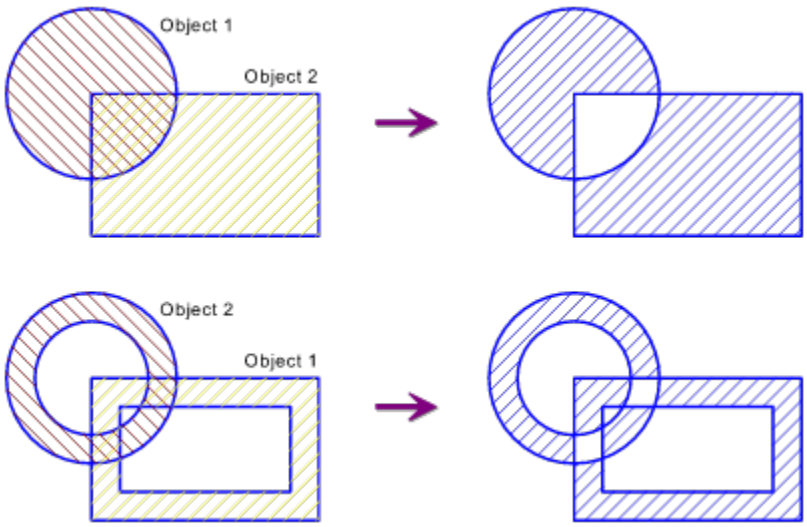
This command can be used to combine existing flat objects to a single surface. The outlines of all objects are simply included in one surface.

The following object types are regarded as "flat": circle, circular sector, circular segment, ellipse, elliptical sector, elliptical segment and surface. Ellipses and ellipse parts are converted into polylines first, and are therefore "only" present in the result as polylines.

1. *Choose objects*
   The objects to be combined in one surface are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

If the duplicate function is on during choosing, the original objects remain unchanged. Otherwise, all the objects used in the new surface are deleted. If objects which cannot be used (e.g. lines, texts, measurements) were chosen, they remain unchanged.

The result in each case is one "surface" object, which may however be made up of more than one surface part. A graphic showing examples of the combinations follows. The differently-colored hatchings show which areas are regarded as "part of the surface". All other areas are "outside the surface". This closes up any holes.



The areas of the objects shown hatched are hatched or filled if the relevant object is hatched or filled. The effects of this combination can be expressed as: (Object 1 - Object 2) × (Object 2 - Object 1).

This command works with all objects which have a closed outline. These are automatically converted to surfaces for the calculation.
Many object types can not be converted into a surface without loss. Ellipses and ellipse parts must be converted into polylines, which invariably leads to a loss of quality. The accuracy of the conversion can be specified using the command Settings>Screen under the heading "Accuracy - Transform".
The original objects (which remain unchanged if the duplicate function is active), are not influenced by this conversion. Surfaces with "holes" can also be used.

Related commands:

- Trim Surface>Union (A=A+B)
- Trim Surface>Intersection (A=A×B)
- Trim Surface>Difference (A=A-B)
- Trim Surface>Split into Outlines
- Trim Surface>Generate Surface

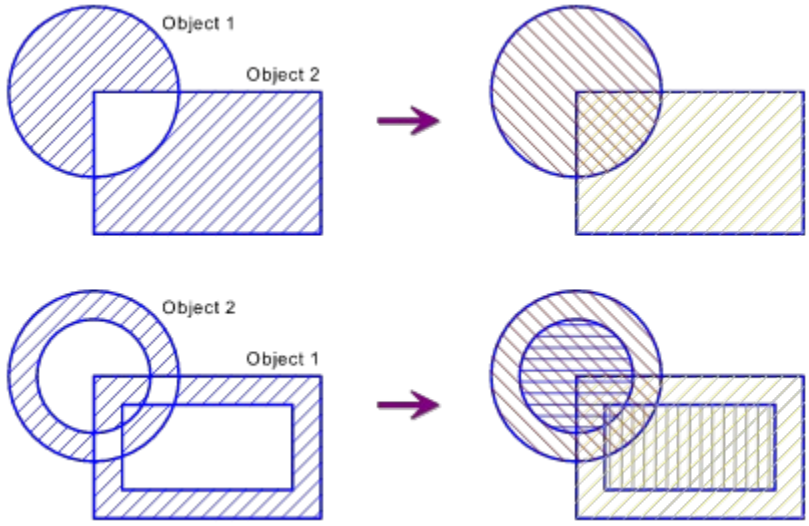
Trim Surface>Split into Outlines (Trimming Menu)

This command can be used to break existing surfaces which contain more than one outline into their individual outlines

1. *Choose objects*
   The objects to be broken into their outlines must be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

If the duplicate function is on during choosing, the original objects remain unchanged. Otherwise they are deleted.

The results are several "surface" objects which each have one outline only. A graphic showing an example follows. The differently-colored hatchings show which areas are regarded as "part of the surface". All other areas are "outside the surface". This closes up any holes.



The areas of the objects which are shown hatched, are hatched or filled when the corresponding object is hatched or filled.

Related commands:

- Trim Surface>Union (A=A+B)
- Trim Surface>Intersection (A=A×B)
- Trim Surface>Difference (A=A-B)
- Trim Surface>Combine Outlines
- Trim Surface>Generate Surface

Trim Edge (Trimming Menu)

This command can be used to trim two lines so that they form a corner.

1. *Identify line 1*
   Click on the first line with the mouse to identify it. If more than one possible line lies close to the position clicked on, then a small dialog appears at the bottom of the screen, from which the "right" line can be selected.

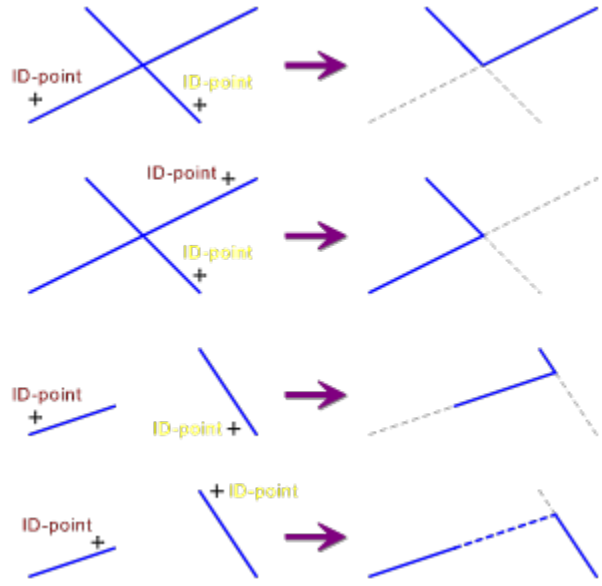
   The position at which the line was identified (the "ID point") affects the rest of the command. It determines which end of the line will be altered. Refer to the graphics at the end of this section.

2. *Identify line 2*
   Click on the second line with the mouse to identify it. If more than one possible line lies close to the position clicked on, then a small dialog appears at the bottom of the screen, from which the "right" line can be selected.

   The position at which the line was identified (the "ID point") affects the rest of the command. It determines which end of the line will be altered. Refer to the graphics at the end of this section.

If the duplicate function was active while selecting the second object (e.g. because the CTRL key was held down), the original lines remain unaltered. Otherwise they are deleted.

The position of the ID points determine which end of the lines will be altered The end of the line closer to the ID point is always the one altered. In this way, any of up to four possible corners can be generated (see graphic).



Note a basic rule for trimming: The object changes, wherever it is clicked on.

If the lines identified are parallel, then an error message appears.

Related commands:


Trim Object>Split


Trim Object>Cut Out

Chamfer Edge>Object - Object (Trimming Menu)

This command can be used to trim two lines so that they form a corner which is immediately chamfered.

1. *Enter angle and length*
   After choosing the command, a dialog appears, in which the chamfer angle and length can be entered. The angle and length remain valid until either a new command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify line 1*
   Click on the first line with the mouse to identify it. If more than one possible line lies close to the position clicked on, then a small dialog appears at the bottom of the screen, from which the "right" line can be selected.

   The position at which the line was identified (the "ID point") affects the rest of the command. It determines which end of the line will be altered. Refer to the graphics at the end of this section.

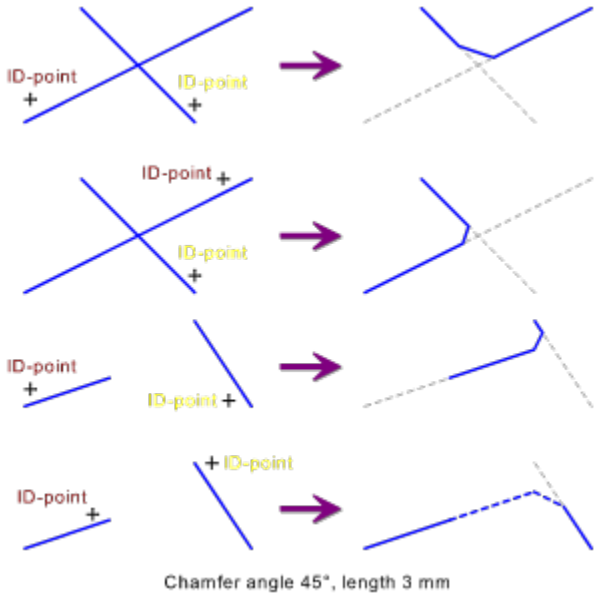
3. *Identify line 2*
   Click on the second line with the mouse to identify it. If more than one possible line lies close to the position clicked on, then a small dialog appears at the bottom of the screen, from which the "right" line can be selected.

   The position at which the line was identified (the "ID point") affects the rest of the command. It determines which end of the line will be altered. Refer to the graphics at the end of this section.

If the duplicate function was active while selecting the second object (e.g. because the CTRL key was held down), the original lines remain unaltered. Otherwise they are deleted.

The chamfer length is the distance between the chamfer start point and the intersection of the identified lines, it is laid on the line which was identified first.

The position of the ID points determine which end of the lines will be altered. The end of the line closer to the ID point is always the one altered. In this way, any of the four possible corners can be created and chamfered (see graphic).



Chamfer angle 45°, length 3 mm

Note a basic rule for trimming: The object changes, wherever it is clicked on.

If the lines identified are parallel, then an error message appears.

Corners within curves or surfaces can be chamfered using the commands Chamfer Edge>One Edge and Chamfer Edge>Complete.

Related commands:

- Chamfer Edge>One Edge
- Chamfer Edge>Complete
- Round Edge, Outer Arc>Object - Object
- Round Edge, Outer Arc>One Edge
- Round Edge, Outer Arc>Complete
- Round Edge, Inner Arc>Object - Object
- Round Edge, Inner Arc>One Edge
- Round Edge, Inner Arc>Complete

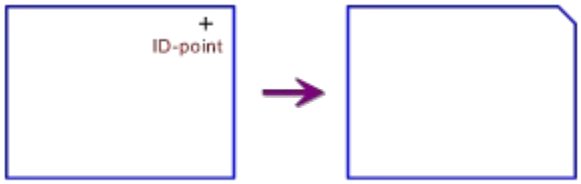
Chamfer Edge>One Edge (Trimming Menu)

This command can be used to chamfer a corner within a curve or surface. Only corners made up of two lines can be chamfered.

1. *Enter angle and length*
   After choosing the command, a dialog appears, in which the chamfer angle and length can be entered. The angle and length remain valid until either a new command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify curve / surface*
   To identify the curve or surface, click on its outline with the mouse. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

   The position at which the object was identified (the "ID point") affects how the command is carried out. It determines which corner will be chamfered (see graphic).



Chamfer angle 45°, length 3 mm

Note a basic rule for trimming: The object changes, wherever it is clicked on.

If the duplicate function was held down while the curve or surface was selected (e.g. because the CTRL key was held down), the object itself is not chamfered, but a copy is generated and that is chamfered.

If the two lines of the "corner" are parallel, then an error message appears.

A corner made from two individual lines can be chamfered using the command Chamfer Edge>Object - Object. All the corners of a curve or surface can be chamfered at once using the command Chamfer Edge>Complete.

Related commands:

- Chamfer Edge>Object - Object
- Chamfer Edge>Complete
- Round Edge, Outer Arc>Object - Object
- Round Edge, Outer Arc>One Edge
- Round Edge, Outer Arc>Complete


Round Edge, Inner Arc>Object - Object


Round Edge, Inner Arc>One Edge

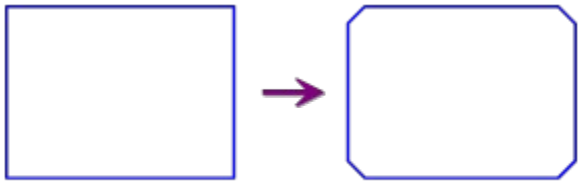

Round Edge, Inner Arc>Complete

Chamfer Edge>Complete (Trimming Menu)

This command can be used to chamfer all the corners in a curve or surface at the same time. Only corners made of two lines will be chamfered.

1. *Enter angle and length*
   After choosing the command, a dialog appears, in which the chamfer angle and length can be entered. The angle and length remain valid until either a new command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify curve / surface*
   To identify the curve or surface, click on its outline with the mouse. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.



Chamfer angle 45°, length 3 mm

If the duplicate function was held down while the curve or surface was selected (e.g. because the CTRL key was held down), the object itself is not chamfered, but a copy is generated and that is chamfered.

A corner made up of two individual lines can be chamfered using the command Chamfer Edge>Object - Object. A single corner of the curve or surface can be chamfered using Chamfer Edge>Complete.

Related commands:

- Chamfer Edge>Object - Object
- Chamfer Edge>One Edge
- Round Edge, Outer Arc>Object - Object
- Round Edge, Outer Arc>One Edge
- Round Edge, Outer Arc>Complete
- Round Edge, Inner Arc>Object - Object
- Round Edge, Inner Arc>One Edge
- Round Edge, Inner Arc>Complete

Round Edge, Outer Arc>Object - Object (Trimming Menu)

This command can be used to trim two lines so that they form a corner which is then rounded using a convex circular arc.

1. *Enter radius*
   After choosing the command, a dialog appears, in which the rounding angle to be used can be entered. This radius remains valid either until another command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify line 1*
   Click on the first line with the mouse to identify it. If more than one possible line lies close to the position clicked on, then a small dialog appears at the bottom of the screen, from which the "right" line can be selected.

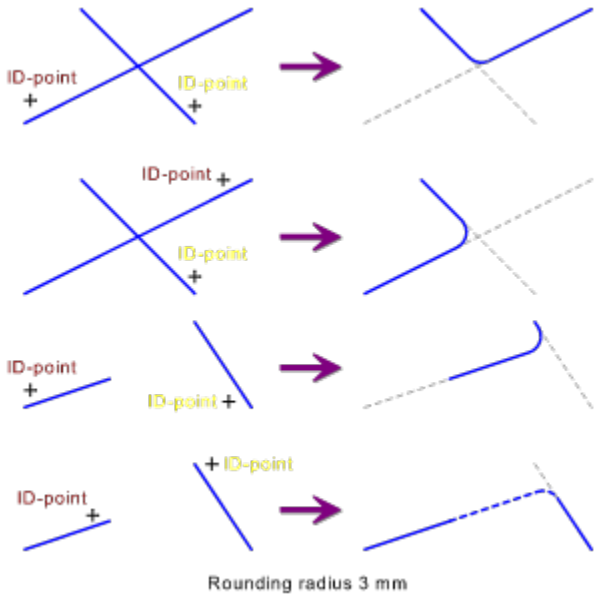
   The position at which the line was identified (the "ID point") affects the rest of the command. It determines which end of the line will be altered. Refer to the graphics at the end of this section.

3. *Identify line 2*
   Click on the second line with the mouse to identify it. If more than one possible line lies close to the position clicked on, then a small dialog appears at the bottom of the screen, from which the "right" line can be selected.

   The position at which the line was identified (the "ID point") affects the rest of the command. It determines which end of the line will be altered. Refer to the graphics at the end of this section.

If the duplicate function was active while selecting the second object (e.g. because the CTRL key was held down), the original lines remain unaltered. Otherwise they are deleted.

The position of the ID points determine which end of the lines will be altered. The end of the line closer to the ID point is always the one altered. In this way, any of the four possible corners can be generated and rounded (see graphic).



Note a basic rule for trimming: The object changes, wherever it is clicked on.

If the lines identified are parallel, then an error message appears.

If a corner in a surface or curve is to be rounded, this can be done with the commands Round Edge, Outer Arc>One Edge und Round Edge, Outer Arc>Complete.

Related commands:

- Chamfer Edge>Object - Object
- Chamfer Edge>One Edge
- Chamfer Edge>Complete
- Round Edge, Outer Arc>One Edge
- Round Edge, Outer Arc>Complete
- Round Edge, Inner Arc>Object - Object
- Round Edge, Inner Arc>One Edge
- Round Edge, Inner Arc>Complete

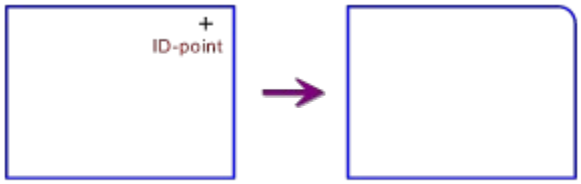
Round Edge, Outer Arc>One Edge (Trimming Menu)

This command can be used to round a corner using a convex arc. Only corners consisting of two lines can be rounded.

1. *Enter radius*
   After choosing the command, a dialog appears, in which the rounding angle to be used can be entered. This radius remains valid either until another command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify curve / surface*
   To identify the curve or surface, click on its outline with the mouse. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

   The position at which the object was identified (the "ID point") affects how the command is carried out. It determines which corner of the object is to be rounded (see graphic).



Rounding radius 3 mm

Note a basic rule for trimming: The object changes, wherever it is clicked on.

If the duplicate function was held down while the curve or surface was selected (e.g. because the CTRL key was held down), a copy of the object is generated and the copy, not the original, is changed.

If the two lines of the "corner" are parallel, then an error message appears.

If a corner made up of two individual lines is to be rounded, this can be done with the command Round Edge, Outer Arc>Object - Object. All the corners in a curve or surface can be chamfered at one using the command Round Edge, Outer Arc>Complete.

Related commands:

- Chamfer Edge>Object - Object
- Chamfer Edge>One Edge
- Chamfer Edge>Complete
- Round Edge, Outer Arc>Object - Object
- Round Edge, Outer Arc>Complete


Round Edge, Inner Arc>Object - Object


Round Edge, Inner Arc>One Edge

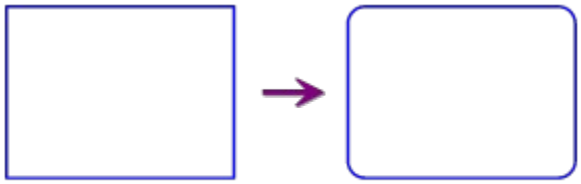

Round Edge, Inner Arc>Complete

Round Edge, Outer Arc>Complete (Trimming Menu)

This command can be used to round all the corners in a curve or surface at the same time using a convex arc. Only corners made up of two lines are rounded.

1. *Enter radius*
   After choosing the command, a dialog appears, in which the rounding angle to be used can be entered. This radius remains valid either until another command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify curve / surface*
   To identify the curve or surface, click on its outline with the mouse. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.



If the duplicate function was held down while the curve or surface was selected (e.g. because the CTRL key was held down), a copy of the object is generated and the copy, not the original, is changed.

A corner made up of two individual lines can be rounded with the command Round Edge, Outer Arc>Object - Object. A single corner in a curve or surfaces can be rounded using Round Edge, Outer Arc>Complete.

Related commands:

- Chamfer Edge>Object - Object
- Chamfer Edge>One Edge
- Chamfer Edge>Complete
- Round Edge, Outer Arc>Object - Object
- Round Edge, Outer Arc>One Edge
- Round Edge, Inner Arc>Object - Object
- Round Edge, Inner Arc>One Edge
- Round Edge, Inner Arc>Complete

Round Edge, Inner Arc>Object - Object (Trimming Menu)

This command can be used to trim two lines so that they form a corner which is immediately rounded with a concave arc

1. *Enter radius*
   After choosing the command, a dialog appears, in which the rounding angle to be used can be entered. This radius remains valid either until another command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify line 1*
   Click on the first line with the mouse to identify it. If more than one possible line lies close to the position clicked on, then a small dialog appears at the bottom of the screen, from which the "right" line can be selected.

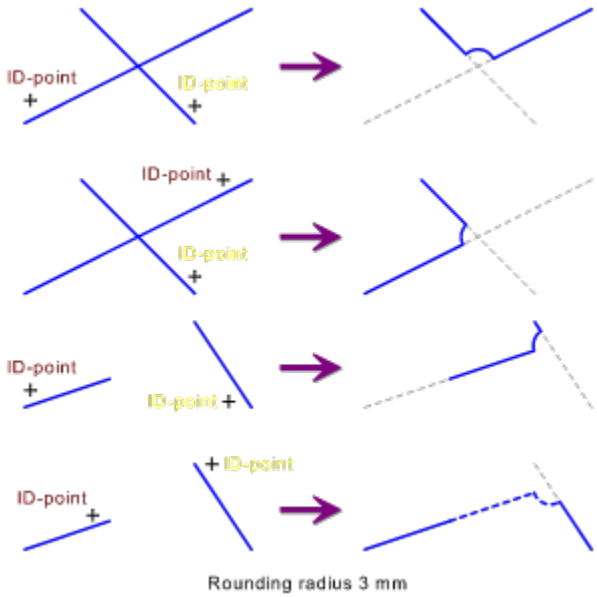
   The position at which the line was identified (the "ID point") affects the rest of the command. It determines which end of the line will be altered. Refer to the graphics at the end of this section.

3. *Identify line 2*
   Click on the second line with the mouse to identify it. If more than one possible line lies close to the position clicked on, then a small dialog appears at the bottom of the screen, from which the "right" line can be selected.

   The position at which the line was identified (the "ID point") affects the rest of the command. It determines which end of the line will be altered. Refer to the graphics at the end of this section.

If the duplicate function was active while selecting the second object (e.g. because the CTRL key was held down), the original lines remain unaltered. Otherwise they are deleted.

The position of the ID points determine which end of the lines will be altered. The end of the line closer to the ID point is always the one altered. In this way, any of the four possible corners can be generated and rounded (see graphic).



Note a basic rule for trimming: The object changes, wherever it is clicked on.

If the lines identified are parallel, then an error message appears.

A corner within a curve or surface can be rounded using the commands Round Edge, Inner Arc>One Edge and Round Edge, Inner Arc>Complete.

Related commands:


Chamfer Edge>Object - Object


Chamfer Edge>One Edge


Chamfer Edge>Complete


Round Edge, Outer Arc>Object - Object


Round Edge, Outer Arc>One Edge


Round Edge, Outer Arc>Complete


Round Edge, Inner Arc>One Edge


Round Edge, Inner Arc>Complete

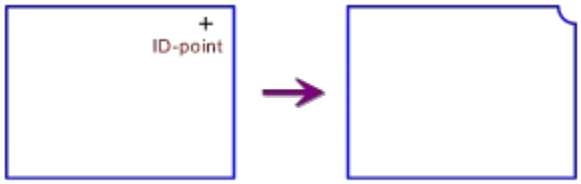
Round Edge, Inner Arc>One Edge (Trimming Menu)

This command can be used to round a corner within a curve or surface with a concave arc. Only corners consisting of two lines can be rounded.

1. *Enter radius*
   After choosing the command, a dialog appears, in which the rounding angle to be used can be entered. This radius remains valid either until another command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify curve / surface*
   To identify the curve or surface, click on its outline with the mouse. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

   The position at which the object was identified (the "ID point") affects how the command is carried out. It determines which corner of the object should be rounded (see graphic).



Rounding radius 3 mm

Note a basic rule for trimming: The object changes, wherever it is clicked on.

If the duplicate function was held down while the curve or surface was selected (e.g. because the CTRL key was held down), a copy of the object is generated and the copy, not the original, is changed.

If the two lines of the "corner" are parallel, then an error message appears.

A corner made up of two individual lines can be rounded using the command Round Edge, Inner Arc>Object - Object. All the corners in a curve or surface can be rounded at the same time using Round Edge, Inner Arc>Complete.

Related commands:

- Chamfer Edge>Object - Object
- Chamfer Edge>One Edge
- Chamfer Edge>Complete
- Round Edge, Outer Arc>Object - Object
- Round Edge, Outer Arc>One Edge

Round Edge, Outer Arc>Complete

Round Edge, Inner Arc>Object - Object

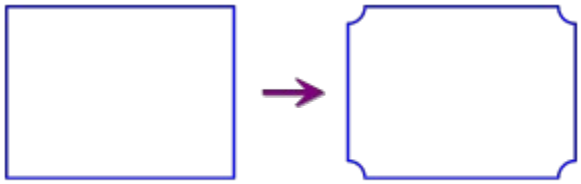
Round Edge, Inner Arc>Complete

Round Edge, Inner Arc>Complete (Trimming Menu)

This command can be used to round all the corners in a surface at the same time, using a concave arc. Only those corners which consist of two lines are rounded.

1. *Enter radius*
   After choosing the command, a dialog appears, in which the rounding angle to be used can be entered. This radius remains valid either until another command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Identify curve / surface*
   To identify the curve or surface, click on its outline with the mouse. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.



If the duplicate function was held down while the curve or surface was selected (e.g. because the CTRL key was held down), a copy of the object is generated and the copy, not the original, is changed.

A corner which is made up of two individual lines can be rounded using the command Round Edge, Inner Arc>Object - Object. A single corner in the curve or surface can be rounded using Round Edge, Inner Arc>Complete.

Related commands:

- Chamfer Edge>Object - Object
- Chamfer Edge>One Edge
- Chamfer Edge>Complete
- Round Edge, Outer Arc>Object - Object
- Round Edge, Outer Arc>One Edge
- Round Edge, Outer Arc>Complete
- Round Edge, Inner Arc>Object - Object
- Round Edge, Inner Arc>One Edge

Transform Object to>Invert (Trimming Menu)

This command can be used to invert a circle or ellipse part. This means that the exact opposite of the previous arc is drawn afterwards.

1. *Identify object*
   In principle, any object can be identified for transformation. However, transformation only makes sense for some objects.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the object itself is not altered, but a copy is generated and then altered.

If a circular arc within a curve is to be inverted, this can be done with the command Trim Curve>Edit. If the circular arc is in a surface, the surface must first be converted to a curve using Transform Objects To>Curve in order to allow the previously-named command to edit it.

Related commands:

 Transform Objects to>Zigzag Line

 Transform Objects to>Circle

 Transform Objects to>Circular Arc

 Transform Objects to>Circular Sector

 Transform Objects to>Circular Segment

 Transform Objects to>Ellipse

 Transform Objects to>Elliptic Arc

 Transform Objects to>Elliptic Sector

 Transform Objects to>Elliptic Segment

 Transform Objects to>Curve

 Transform Objects to>Surface

Transform Object to>Zigzag Line (Trimming Menu) 

This command can be used to transform a normal line into a zigzag line.

1. *Identify object*
   In principle, any object can be identified for transformation. However, transformation only makes sense for some objects.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the object itself is not altered, but a copy is generated and then altered.

Related commands:

 Transform Objects to>Invert

 Transform Objects to>Circle

 Transform Objects to>Circular Arc

 Transform Objects to>Circular Sector

 Transform Objects to>Circular Segment

 Transform Objects to>Ellipse

 Transform Objects to>Elliptic Arc

 Transform Objects to>Elliptic Sector

 Transform Objects to>Elliptic Segment

 Transform Objects to>Curve

 Transform Objects to>Surface

Transform Object to>Circle (Trimming Menu) 

This command can be used to convert a circle part to a full circle.

1. *Identify object*
   In principle, any object can be identified for transformation. However, transformation only makes sense for some objects.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the object itself is not altered, but a copy is generated and then altered.

Related commands:

 Transform Objects to>Invert

 Transform Objects to>Zigzag Line

 Transform Objects to>Circular Arc

 Transform Objects to>Circular Sector

 Transform Objects to>Circular Segment

 Transform Objects to>Ellipse

 Transform Objects to>Elliptic Arc

 Transform Objects to>Elliptic Sector

 Transform Objects to>Elliptic Segment

 Transform Objects to>Curve

 Transform Objects to>Surface

Transform Object to>Circular Arc (Trimming Menu)

This command can be used to convert a circle part into a circular arc.

1. *Identify object*
   In principle, any object can be identified for transformation. However, transformation only makes sense for some objects.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the object itself is not altered, but a copy is generated and then altered.

If a circle is to be transformed into a circular arc, this can be done with the command Trim Object>Cut Out.

Related commands:

- Transform Objects to>Invert
- Transform Objects to>Zigzag Line
- Transform Objects to>Circle
- Transform Objects to>Circular Sector
- Transform Objects to>Circular Segment
- Transform Objects to>Ellipse
- Transform Objects to>Elliptic Arc
- Transform Objects to>Elliptic Sector
- Transform Objects to>Elliptic Segment
- Transform Objects to>Curve
- Transform Objects to>Surface

Transform Object to>Circular Sector (Trimming Menu)

A circle part can be converted to a circular segment using this command.

1. *Identify object*
   In principle, any object can be identified for transformation. However, transformation only makes sense for some objects.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the object itself is not altered, but a copy is generated and then altered.

If a circle is to be transformed to a circular sector, this can be done with the command Trim Object>Cut Out.

Related commands:

- Transform Objects to>Invert
- Transform Objects to>Zigzag Line
- Transform Objects to>Circle
- Transform Objects to>Circular Arc
- Transform Objects to>Circular Segment
- Transform Objects to>Ellipse
- Transform Objects to>Elliptic Arc
- Transform Objects to>Elliptic Sector
- Transform Objects to>Elliptic Segment
- Transform Objects to>Curve
- Transform Objects to>Surface

Transform Object to>Circular Segment (Trimming Menu)

This command can be used to transform a circle part to a circular segment.

1. *Identify object*
   In principle, any object can be identified for transformation. However, transformation only makes sense for some objects.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the object itself is not altered, but a copy is generated and then altered.

If a circle is to be transformed into a circular segment, this can be done with the command Trim Object>Cut Out.

Related commands:

- Transform Objects to>Invert
- Transform Objects to>Zigzag Line
- Transform Objects to>Circle
- Transform Objects to>Circular Arc
- Transform Objects to>Circular Sector
- Transform Objects to>Ellipse
- Transform Objects to>Elliptic Arc
- Transform Objects to>Elliptic Sector
- Transform Objects to>Elliptic Segment
- Transform Objects to>Curve
- Transform Objects to>Surface

Transform Object to>Ellipse (Trimming Menu)

This command can be used to convert a circle, circle part or ellipse part to an ellipse.

1. *Identify object*
   In principle, any object can be identified for transformation. However, transformation only makes sense for some objects.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the object itself is not altered, but a copy is generated and then altered.

During the transformation of a circle or circle part to an ellipse, a right-angled ellipse whose half axes have the length of the original circle"s radius and run vertically or horizontally is generated.

Related commands:

- Transform Objects to>Invert
- Transform Objects to>Zigzag Line
- Transform Objects to>Circle
- Transform Objects to>Circular Arc
- Transform Objects to>Circular Sector
- Transform Objects to>Circular Segment
- Transform Objects to>Elliptic Arc
- Transform Objects to>Elliptic Sector
- Transform Objects to>Elliptic Segment
- Transform Objects to>Curve
- Transform Objects to>Surface

Transform Object to>Elliptic Arc (Trimming Menu)

This command can be used to transform a circle or ellipse part to an elliptic arc.

1. *Identify object*
   In principle, any object can be identified for transformation. However, transformation only makes sense for some objects.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the object itself is not altered, but a copy is generated and then altered.

During the transformation of a circle or circle part to an ellipse, a right-angled ellipse whose half axes have the length of the original circle"s radius and run vertically or horizontally is generated.

If an ellipse is to be converted to an elliptic arc, this can be done with the command Trim Object>Cut Out.

Related commands:

- Transform Objects to>Invert
- Transform Objects to>Zigzag Line
- Transform Objects to>Circle
- Transform Objects to>Circular Arc
- Transform Objects to>Circular Sector
- Transform Objects to>Circular Segment
- Transform Objects to>Ellipse
- Transform Objects to>Elliptic Sector
- Transform Objects to>Elliptic Segment
- Transform Objects to>Curve
- Transform Objects to>Surface

Transform Object to>Elliptic Sector (Trimming Menu)

This command can be used to transform a circle part or ellipse part to an elliptic sector.

1. *Identify object*
   In principle, any object can be identified for transformation. However, transformation only makes sense for some objects.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the object itself is not altered, but a copy is generated and then altered.

During the transformation of a circle or circle part to an ellipse, a right-angled ellipse whose half axes have the length of the original circle"s radius and run vertically or horizontally is generated.

If an ellipse is to be transformed to an elliptic sector, this can be done with the command Trim Object>Cut Out.

Related commands:

- Transform Objects to>Invert
- Transform Objects to>Zigzag Line
- Transform Objects to>Circle
- Transform Objects to>Circular Arc
- Transform Objects to>Circular Sector
- Transform Objects to>Circular Segment
- Transform Objects to>Ellipse
- Transform Objects to>Elliptic Arc
- Transform Objects to>Elliptic Segment
- Transform Objects to>Curve
- Transform Objects to>Surface

Transform Object to>Elliptic Segment (Trimming Menu)

This command can be used to transform a circle part or ellipse part to an elliptic segment.

1. *Identify object*
   In principle, any object can be identified for transformation. However, transformation only makes sense for some objects.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the object itself is not altered, but a copy is generated and then altered.

During the transformation of a circle or circle part to an ellipse, a right-angled ellipse whose half axes have the length of the original circle"s radius and run vertically or horizontally is generated.

If an ellipse is to be transformed to an elliptic segment, this can be done with the command Trim Object>Cut Out.

Related commands:

- Transform Objects to>Invert
- Transform Objects to>Zigzag Line
- Transform Objects to>Circle
- Transform Objects to>Circular Arc
- Transform Objects to>Circular Sector
- Transform Objects to>Circular Segment
- Transform Objects to>Ellipse
- Transform Objects to>Elliptic Arc
- Transform Objects to>Elliptic Sector
- Transform Objects to>Curve
- Transform Objects to>Surface

Transform Objects to>Curve (Trimming Menu)

This command can be used to convert an object to a curve. A curve is an "open" object, i.e. it does not have a closed outline and therefore can not be filled. If filled objects are converted to curves, they will no longer be available, but will however still be present.

1. *Identify object*
   In principle, any object can be identified for transformation. However, transformation only makes sense for some objects.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the object itself is not altered, but a copy is generated and then altered.

Many object types cannot be converted cleanly to a curve. Zigzag lines, splines, ellipses and ellipse parts must be converted into polylines, which invariably leads to a loss of quality. The accuracy of the conversion can be specified using the command Settings>Screen under the heading "Accuracy - Transform".

A surface can only be converted to a curve if it has a single outline. If it consists of multiple outlines, then the command Trim Surface>Split into Outlines must be used to split it into individual outlines

Related commands:

 Transform Objects to>Invert

 Transform Objects to>Zigzag Line

 Transform Objects to>Circle

 Transform Objects to>Circular Arc

 Transform Objects to>Circular Sector

 Transform Objects to>Circular Segment

 Transform Objects to>Ellipse

 Transform Objects to>Elliptic Arc

 Transform Objects to>Elliptic Sector

 Transform Objects to>Elliptic Segment

 Transform Objects to>Surface

 Text>Resolve

Transform Object to>Surface (Trimming Menu)

This command can be used to transform an object to a surface.

1. *Identify object*
   In principle, any object can be identified for transformation. However, transformation only makes sense for some objects.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the object itself is not altered, but a copy is generated and then altered.

Many object types can not be converted cleanly into a surface. Ellipses and ellipse parts must be converted into polylines, which invariably leads to a loss of quality. The accuracy of the conversion can be specified using the command Settings>Screen under the heading "Accuracy - Transform".

Related commands:

 Transform Objects to>Invert

 Transform Objects to>Zigzag Line

 Transform Objects to>Circle

 Transform Objects to>Circular Arc

 Transform Objects to>Circular Sector

 Transform Objects to>Circular Segment

 Transform Objects to>Ellipse

 Transform Objects to>Elliptic Arc

 Transform Objects to>Elliptic Sector

 Transform Objects to>Elliptic Segment

 Transform Objects to>Curve

 Text>Resolve

Clipping>Create, Inside (Trimming Menu)

This command can be used to create a clipping surface that leaves only its inside visible.

1. *Choose objects*
   The objects which are to be clipped are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Identify aerial object*
   Any fillable object can be identified as clipping surface.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the objects themselves are not altered, but a copy is generated and then altered.

Many object types can not be converted cleanly into a surface. Ellipses and ellipse parts must be converted into polylines, which invariably leads to a loss of quality. The accuracy of the conversion can be specified using the command Settings>Screen under the heading "Accuracy - Transform".

Related commands:

Clipping>Create, Outside

Clipping>Resolve

Clipping>Create, Outside (Trimming Menu)

This command can be used to create a clipping surface that leaves only its outside visible.

1. *Choose objects*
   The objects which are to be clipped are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Identify aerial object*
   Any fillable object can be identified as clipping surface.
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the objects themselves are not altered, but a copy is generated and then altered.

Many object types can not be converted cleanly into a surface. Ellipses and ellipse parts must be converted into polylines, which invariably leads to a loss of quality. The accuracy of the conversion can be specified using the command Settings>Screen under the heading "Accuracy - Transform".

Related commands:

Clipping>Create, Inside

Clipping>Resolve

Clipping>Resolve (Trimming Menu)

This command can be used to resolve a clipping surface into a surface and an instance.

1. *Identify clipping surface*
   Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

If the duplicate function was active during the selection process (e.g. because the CTRL key was held down), the object itself is not altered, but a copy is generated and then altered.

Related commands:

Clipping>Create, Inside

Clipping>Create, Outside

Text Menu

Text
- Standard
- Frame
- Reference
- Resolve

Dimension Line
- Straight
- Curved

Dimension
- Length, Points
- Length, Object
- Distance, Object - Point
- Distance, Object - Object
- Radius, Points
- Radius, Object
- Diameter, Points
- Diameter, Object
- Angle, Points
- Angle, Line - Line
- Angle, Circle Part
- Arc Length, Points
- Arc Length, Circle Part
- Coordinates

Calculate
- Area
- Perimeter / Length

Dimension Parameters

Edit Dimension Line
- Rotate
- Position

Edit Dimension
- Rotate
- Position
- Update

Fonts

Text>Standard (Text Menu)

This command can be used to generate a standard text. This type of text can consist of up to 8000 characters split into multiple lines.

1. *Enter text*
   After choosing the command, the Enter Standard Text dialog appears in which the text can be entered and its parameters can be specified. The text and parameters remain valid either until a new command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Enter position*
   The text position can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The text generated is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

Related commands:

Text>Frame

Text>Reference

Text>Frame (Text Menu)

This command can be used to generate a framed text. This type of text can consist of up to 8000 characters split into multiple lines. A rectangle, which can be rotated and scaled, is specified for the text to go into.

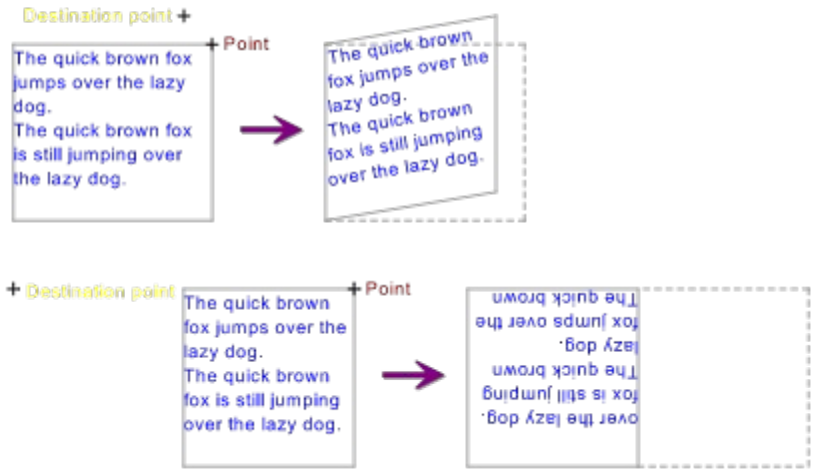
1. *Area: Enter corner point 1*
   The first corner point of the text area can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Area: Enter corner point 2*
   The second corner point of the text area can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter text*
   After specifying the text area, the Enter Frame Text dialog appears where the text can be entered and its parameters can be specified.

The text generated is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

The text area is defined internally by three points, which span a parallelogram. If one of the points is moved (e.g. using Move Single Point), then the text adapts itself accordingly.
The alteration only affects the way in which the individual lines of text run. The text size is not altered, and the characters are not distorted. The following diagram shows the effect of moving the definition point.



Related commands:

Text>Standard

Text>Reference

Text>Reference (Text Menu)

This command can be used to generate a reference text. This type of text can consist of up to 8000 characters split into multiple lines.

1. *Enter text*
   After choosing the command, the Enter Standard Text dialog appears in which the text can be entered and its parameters can be specified. The text and parameters remain valid either until a new command is chosen or the current command is restarted using the command Change Parameters (+ESC) or the key combination SHIFT+ESC.

2. *Enter reference point*
   The reference point, i.e. the end-point of the reference line, can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter destination point*
   The destination point, i.e. the position of the reference text filed, can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

The text generated is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

Related commands:

Text>Standard

Text>Frame

Text>Resolve (Text Menu)

This command can be used to resolve a text object into individual characters. The characters are converted into curves or surfaces. This is true for both TommySoftware® CAD/DRAW's own fonts and TrueType fonts.

1. *Choose texts*
   The text objects who should be resolved are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

If the duplicate function is active when the objects are chosen, a copy of the text objects is made, and the copy (not the originals) is resolved.

Characters converted to surfaces require significantly more memory than the original text. A single line of text in a complicated font can be over 100K in size when converted. Apart from the increase in required memory, the conversion results in screen redraws taking longer; the text can also no longer be edited. It is only necessary to convert text to surfaces if individual characters have to be manipulated.

If serious problems arise when sending TrueType fonts to a particular output device, then they can, at the same time, be converted to surfaces for screen display and/or printing (see the "TrueType always as surface" checkbox in the dialogs Screen Settings and Output Settings). The objects themselves are unchanged by these settings and can still be edited. In addition, the memory requirement does not increase.

Related command:

Group>Ungroup

Trim Object>Resolve Completely

Resolve Instances

Dimension Line>Straight (Text Menu)

This command can be used to draw a straight dimension line. A straight dimension line is basically a line with two end symbols, e.g. arrowheads or points.

1. *Enter start point*
   The dimension line's start point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter end point*
   The dimension line's end point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

When choosing the Change Parameters (+ESC) command, the Dimension Line dialog appears where this command's parameters can be specified.

Dimension lines are not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

Related commands:

Dimension Line>Curved

Line>Standard

Dimension Line>Curved (Text Menu)

This command can be used to draw a curved dimension line. A curved dimension line is basically a circular arc with two end symbols, e.g. arrowheads or points.

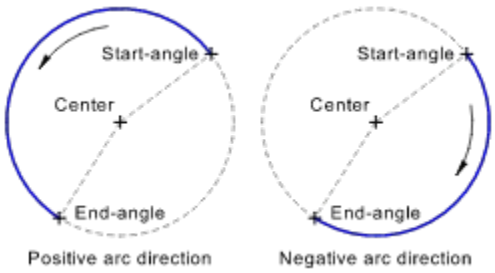
1. *Enter center point*
   The center point of the circular arc can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter radius*
   The circular arc radius can be entered with the mouse, by clicking anywhere in the drawing at a point through which the circle should run. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). It is also possible to enter the radius directly.

3. *Enter start angle*
   The circular arc's start angle can be entered using the mouse, by clicking anywhere in the drawing. The position of this point relative to the center point determines the start angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter the start angle directly.

4. *Enter end angle*
   The circular arc's end angle can be entered using the mouse, by clicking anywhere in the drawing. The position of this point relative to the center point determines the end angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). You can also enter the end angle directly. The angle can also be given relative to a previously entered start point.

   The arc's appearance is determined by the current arc direction. This is specified using the command Toggle Arc Direction(F2), with the F2 key or the button in the panel. If the arc direction is positive, the arc is drawn from the start angle in a mathematically positive sense (i.e. anticlockwise). If the arc direction is negative, the arc is drawn from the start angle in a mathematically negative sense (i.e. clockwise) (see graphic).



   The appearance of the circular arc is shown during input, and changes if the current arc direction is changed.

When choosing the Change Parameters (+ESC) command, the Dimension Line dialog appears where this command's parameters can be specified.

Dimension lines are not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

The arc direction saved in the curved dimension line remains unchanged if the current arc direction is

changed later. If a curved dimension line's arc direction is to be changed, use the command Transform Objects to>Invert.

Related commands:

Dimension Line>Straight

Circular Arc>Standard

Dimension>Length, Points (Text Menu)

This command can be used to measure a distance with specified end points, i.e. it can be used to measure distances which are not in the form of an edge or line.

1. *Enter start point*
   The dimension line's start point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter end point*
   The dimension line's end point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After identifying the distance to be measured, the position of the dimension line and number must be specified. To do this, several points have to be entered. How many, and which, points have to be entered, depends on the specified dimension parameters.

When choosing the Change Parameters (+ESC) command, the Dimension With Line dialog appears where this command's parameters can be specified.

If "Display Dimension Line" is set and the dimension direction is set to "Any":

3. *Enter dimension line direction*
   The dimension line direction is determined by a point which, together with the start point of the distance to be measured defines a straight line in whose direction the dimension should run. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension's direction.

If "Display Dimension Line" is enabled:

4. *Enter dimension line position*
   The position of the dimension line is determined by the position of a point through which it should run. If a particular distance has been entered, then the point determines the approximate position of the dimension line. This point can be entered using the mouse, by clicking anywhere in the drawing.

   If the dimension line mode is set to "Full Length, Random", then this point also determines the start point of the dimension line and therefore the direction of any extension line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, a length can be entered. This determines the distance between the dimension line and the length to be measured.

If "Display Dimension Line" is enabled and the dimension line mode is set to "Partial Length, Right Angled":

5. *Enter dimension line end point*
   The dimension line's end point can be entered using the mouse, by clicking anywhere in the drawing. A perpendicular is dropped from this point to the dimension line to determine the end point of the dimension line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Center Dimension Text" or "Close Dimension Text" is not set:

6. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the

drawing. The positioning of the dimension text can be restricted with the "Center Dimension Text" and "Close Dimension Text". Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

7. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

A dimension contains *two* property sets - one for the dimension line, and one for the dimension text. For example, this makes it possible to show the dimension text in a different color to the dimension line or assign the dimension text to a different layer from the dimension line.
If the command Edit Properties is used on a dimension, you can switch between the two property sets by clicking the corresponding tab. This allows you to view and alter both property sets separately.

Related commands:

 Dimension>Length, Points

 Dimension>Length, Object

 Dimension>Distance, Object - Point

 Dimension>Distance, Object - Object

 Dimension>Arc Length, Points

 Dimension>Arc Length, Circle Part

 Calculate>Perimeter / Length

Dimension>Length, Object (Text Menu)

This command can be used to measure an existing individual line or an edge in an object.

1. *Identify reference line*
   Any line in an existing object can be identified as the reference line. This can also be the edge of a rectangle or a leg of a circular segment.

   To identify it, the line is clicked on with the mouse. If more than one line lies close to the position clicked on, a small dialog appears at the bottom of the screen, with the help of which the 'right' line can be chosen.

   The position at which the reference line was identified (the ID point), affects the rest of the command. It determines which end of the distance is used as the start point. The end point nearer to the ID point is always used.

After identifying the distance to be measured, the position of the dimension line and dimension text has to be specified. To do this, several points have to be entered. How many, and which, points have to be entered, depends on the specified dimension parameters.

When choosing the Change Parameters (+ESC) command, the Dimension With Line dialog appears where this command's parameters can be specified.

If "Display Dimension Line" is set and the dimension direction is set to "Any":

2. *Enter dimension line direction*
   The dimension line direction is determined by a point which, together with the start point of the distance to be measured defines a straight line in whose direction the dimension should run. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension's direction.

If "Display Dimension Line" is enabled:

3. *Enter dimension line position*
   The position of the dimension line is determined by the position of a point through which it should run. If a particular distance has been entered, then the point determines the approximate position of the dimension line. This point can be entered using the mouse, by clicking anywhere in the drawing.

   If the dimension line mode is set to "Full Length, Random", then this point also determines the start point of the dimension line and therefore the direction of any extension line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, a length can be entered. This determines the distance between the dimension line and the length to be measured.

If "Display Dimension Line" is enabled and the dimension line mode is set to "Partial Length, Right Angled":

4. *Enter dimension line end point*
   The dimension line's end point can be entered using the mouse, by clicking anywhere in the drawing. A perpendicular is dropped from this point to the dimension line to determine the end point of the dimension line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Center Dimension Text" or "Close Dimension Text" is not set:

5. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. The positioning of the dimension text can be restricted with the "Center Dimension Text" and "Close Dimension Text". Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

6. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

A dimension contains *two* property sets - one for the dimension line, and one for the dimension text. For example, this makes it possible to show the dimension text in a different color to the dimension line or assign the dimension text to a different layer from the dimension line.
If the command Edit Properties is used on a dimension, you can switch between the two property sets by clicking the corresponding tab. This allows you to view and alter both property sets separately.

Related commands:


Dimension>Length, Points


Dimension>Distance, Object - Point


Dimension>Distance, Object - Object


Dimension>Arc Length, Points


Dimension>Arc Length, Circle Part

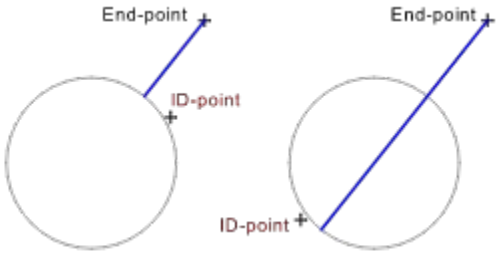

Calculate>Perimeter / Length

Dimension>Distance, Object - Point (Text Menu)

This command can be used to measure a distance defined as the distance between an object and a point.

1. *Identify reference object*
   Any part object which is part of an existing object can be identified as a reference object. This can be the edge of a rectangle, the arc of an ellipse segment or a circular arc within a surface.
   To identify a (part) object, click on it with the mouse. If more than one (part) object lies at the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the 'right' (part) object can be chosen.

   If a circle or ellipse part is identified the point at which the object was identified (the ID point) affects the rest of the command. If the distance from a point to a circle is being measured, then this distance can be calculated to either side of the circle. The point nearer the ID point is used as the end of the distance (see graphic).



2. *Enter end point*
   The end point of the distance can be entered using the mouse, by clicking anywhere in the drawing. A perpendicular is dropped from this point to the reference object in order to work out the distance. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After determining the distance to be measured, the position of the dimension line and dimension text must be specified. To do this, several points have to be entered. How many, and which, points have to be entered, depends on the specified dimension parameters.

When choosing the Change Parameters (+ESC) command, the Dimension With Line dialog appears where this command's parameters can be specified.

If "Display Dimension Line" is enabled:

3. *Enter dimension line position*
   The position of the dimension line is determined by the position of a point through which it should run. If a particular distance has been entered, then the point determines the approximate position of the dimension line. This point can be entered using the mouse, by clicking anywhere in the drawing.

   If the dimension line mode is set to "Full Length, Random", then this point also determines the start point of the dimension line and therefore the direction of any extension line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, a length can be entered. This determines the distance between the dimension line and the length to be measured.

If "Display Dimension Line" is enabled and the dimension line mode is set to "Partial Length, Right Angled":

4. *Enter dimension line end point*
   The dimension line's end point can be entered using the mouse, by clicking anywhere in the drawing. A perpendicular is dropped from this point to the dimension line to determine the end point of the dimension line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Center Dimension Text" or "Close Dimension Text" is not set:

5. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. The positioning of the dimension text can be restricted with the "Center Dimension Text" and "Close Dimension Text". Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

6. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

A dimension contains *two* property sets - one for the dimension line, and one for the dimension text. For example, this makes it possible to show the dimension text in a different color to the dimension line or assign the dimension text to a different layer from the dimension line.
If the command Edit Properties is used on a dimension, you can switch between the two property sets by clicking the corresponding tab. This allows you to view and alter both property sets separately.

Related commands:


Dimension>Length, Points


Dimension>Length, Object


Dimension>Distance, Object - Object


Dimension>Arc Length, Points


Dimension>Arc Length, Circle Part


Calculate>Perimeter / Length

Dimension>Distance, Object - Object (Text Menu)

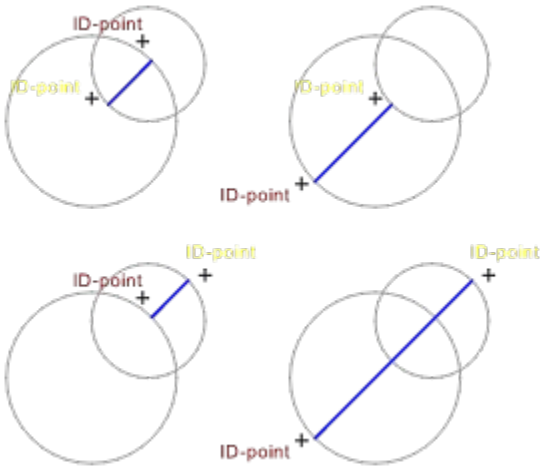
This command can be used to measure a distance defined as the space between two objects.

1. *Identify reference object 1*
   Any partial object in an existing object can be identified as the first reference object. This can be an edge of a rectangle or a circular arc within a surface. Ellipses and ellipse parts can not be identified. To identify a (part) object, click on it with the mouse. If more than one (part) object lies at the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the 'right' (part) object can be chosen.

   If a circle or circle part is identified, then the position at which it was identified (the ID point) affects the rest of the command. See the graphics below.

2. *Identify reference object 2*
   Any partial object in an existing object can be identified as the first reference object. This can be an edge of a rectangle or a circular arc within a surface. Ellipses and ellipse parts can not be identified. To identify a (part) object, click on it with the mouse. If more than one (part) object lies at the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the 'right' (part) object can be chosen.

   If a circle or circle part is identified, then the position at which it was identified (the ID point) affects the rest of the command. The two ID points determine which of the two possible distances will be used (see graphics).



After determining the distance to be measured, the position of the dimension line and dimension text must be specified. To do this, several points have to be entered. How many, and which, points have to be entered, depends on the specified dimension parameters.

When choosing the Change Parameters (+ESC) command, the Dimension With Line dialog appears where this command's parameters can be specified.

If "Display Dimension Line" is enabled:

3. *Enter dimension line position*
   The position of the dimension line is determined by the position of a point through which it should run. If a particular distance has been entered, then the point determines the approximate position of

the dimension line. This point can be entered using the mouse, by clicking anywhere in the drawing.

If the dimension line mode is set to "Full Length, Random", then this point also determines the start point of the dimension line and therefore the direction of any extension line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, a length can be entered. This determines the distance between the dimension line and the length to be measured.

If "Display Dimension Line" is enabled and the dimension line mode is set to "Partial Length, Right Angled":

4. *Enter dimension line end point*
   The dimension line's end point can be entered using the mouse, by clicking anywhere in the drawing. A perpendicular is dropped from this point to the dimension line to determine the end point of the dimension line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Center Dimension Text" or "Close Dimension Text" is not set:

5. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. The positioning of the dimension text can be restricted with the "Center Dimension Text" and "Close Dimension Text". Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

6. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

A dimension contains *two* property sets - one for the dimension line, and one for the dimension text. For example, this makes it possible to show the dimension text in a different color to the dimension line or assign the dimension text to a different layer from the dimension line.
If the command Edit Properties is used on a dimension, you can switch between the two property sets by clicking the corresponding tab. This allows you to view and alter both property sets separately.

Related commands:

Dimension>Length, Points


Dimension>Length, Object


Dimension>Distance, Object - Point


Dimension>Arc Length, Points


Dimension>Arc Length, Circle Part


Calculate>Perimeter / Length

Dimension>Radius, Points (Text Menu)

This command can be used to measure the radius of a circle whose center point and radius are directly entered. Radiuses whose reference circle is not present in the form of an object can also be measured.

1. *Enter center point*
   The center of the reference circle can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter radius*
   The radius of the reference circle can be entered with the mouse by clicking anywhere in the drawing at a point through which the circle should run. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). It is also possible to enter the radius directly. This determines the radius of the reference circle.

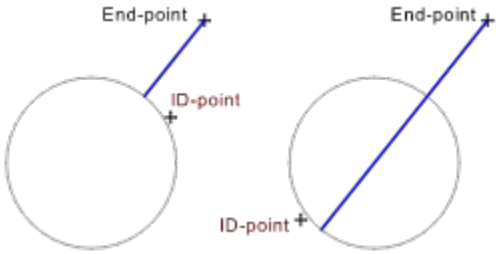
After specifying the reference circle to be measured, the position of the dimension line and the dimension text must be specified. To do this, several points have to be entered. How many, and which, points have to be entered, depends on the specified dimension parameters.

When choosing the Change Parameters (+ESC) command, the Dimension With Line dialog appears where this command's parameters can be specified.

If "Display Dimension Line" is set and the dimension direction is set to "Any":

3. *Enter dimension line direction*
   The direction of the dimension line is determined by a point which, together with the center point of the reference circle, defines a straight line in whose direction the dimension runs. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

   If the dimension line mode is set to "Partial Length, Right Angled", this point also determines which side of the reference circle the dimension line should appear on (see graphic).



If "Display Dimension Line" is enabled:

4. *Enter dimension line position*
   The position of the dimension line is determined by the position of a point through which it should run. If a particular distance has been entered, then the point determines the approximate position of the dimension line. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If the dimension line mode is not "Partial Length, Right Angled", it is possible to enter a length. This determines the distance between the dimension line and the length to be measured.

If the dimension line mode is set to "Partial Length, Right Angled", then this point determines the end point of the dimension line. A perpendicular is dropped from this point to the dimension line to work out the end point.

If the dimension line mode is set to "Full Length, Random", then this point also determines the start point of the dimension line and therefore the direction of any extension line.

If "Center Dimension Text" or "Close Dimension Text" is not set:

5. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. The positioning of the dimension text can be restricted with the "Center Dimension Text" and "Close Dimension Text". Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

6. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified. An *R* appears in front of the text.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

A dimension contains *two* property sets - one for the dimension line, and one for the dimension text. For example, this makes it possible to show the dimension text in a different color to the dimension line or assign the dimension text to a different layer from the dimension line.
If the command Edit Properties is used on a dimension, you can switch between the two property sets by clicking the corresponding tab. This allows you to view and alter both property sets separately.

Related commands:


Dimension>Radius, Object


Dimension>Diameter, Points


Dimension>Diameter, Object

Dimension>Radius, Object (Text Menu)

This command can be used to measure the radius of a circle which exists on its own or as part of an object's edge.

1. *Identify reference circle*
   Any circle which exists separately or as part of another object can be identified as the reference object. This can also be the arc of a circle segment or a circular arc within a surface.
   To identify a (part) object, click on it with the mouse. If more than one (part) object lies at the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the 'right' (part) object can be chosen.

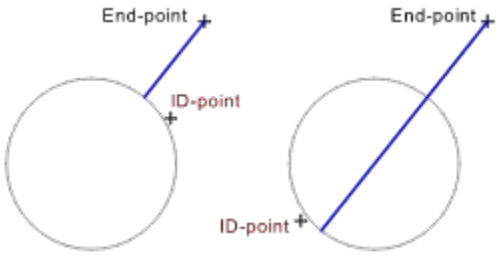
After selecting the reference circle to measure, the position of the dimension line and the dimension text must be specified. To do this, several points have to be entered. How many, and which, points have to be entered, depends on the specified dimension parameters.

When choosing the Change Parameters (+ESC) command, the Dimension With Line dialog appears where this command's parameters can be specified.

If "Display Dimension Line" is set and the dimension direction is set to "Any":

2. *Enter dimension line direction*
   The direction of the dimension line is determined by a point which, together with the center point of the reference circle, defines a straight line in whose direction the dimension runs. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension's direction.

   If the dimension line mode is set to "Partial Length, Right Angled", this point also determines which side of the reference circle the dimension line should appear on (see graphic).



If "Display Dimension Line" is enabled:

3. *Enter dimension line position*
   The position of the dimension line is determined by the position of a point through which it should run. If a particular distance has been entered, then the point determines the approximate position of the dimension line. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If the dimension line mode is not "Partial Length, Right Angled", it is possible to enter a length. This determines the distance between the dimension line and the length to be measured.

   If the dimension line mode is set to "Partial Length, Right Angled", then this point determines the end point of the dimension line. A perpendicular is dropped from this point to the dimension line to work

out the end point.

If the dimension line mode is set to "Full Length, Random", then this point also determines the start point of the dimension line and therefore the direction of any extension line.

If "Center Dimension Text" or "Close Dimension Text" is not set:

4. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. The positioning of the dimension text can be restricted with the "Center Dimension Text" and "Close Dimension Text". Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

5. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified. An *R* appears in front of the text.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

A dimension contains *two* property sets - one for the dimension line, and one for the dimension text. For example, this makes it possible to show the dimension text in a different color to the dimension line or assign the dimension text to a different layer from the dimension line.
If the command Edit Properties is used on a dimension, you can switch between the two property sets by clicking the corresponding tab. This allows you to view and alter both property sets separately.

Related commands:


Dimension>Radius, Points


Dimension>Diameter, Points


Dimension>Diameter, Object

Dimension>Diameter, Points (Text Menu)

This command can be used to measure the diameter of a circle whose center and radius are entered directly. This can also be used to measure a diameter whose reference circle is not present in the form of an object.

1. *Enter center point*
   The center of the reference circle can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter radius*
   The radius of the reference circle can be entered with the mouse by clicking anywhere in the drawing at a point through which the circle should run. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). It is also possible to enter the radius directly. This determines the radius of the reference circle.

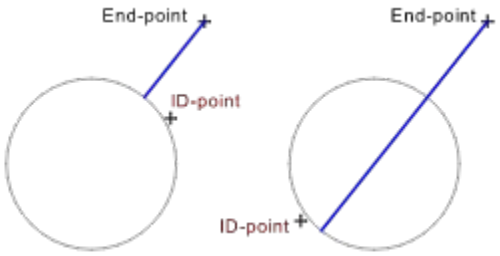
After specifying the reference circle to be measured, the position of the dimension line and the dimension text must be specified. To do this, several points have to be entered. How many, and which, points have to be entered, depends on the specified dimension parameters.

When choosing the Change Parameters (+ESC) command, the Dimension With Line dialog appears where this command's parameters can be specified.

If "Display Dimension Line" is set and the dimension direction is set to "Any":

3. *Enter dimension line direction*
   The direction of the dimension line is determined by a point which, together with the center point of the reference circle, defines a straight line in whose direction the dimension runs. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension's direction.

   If the dimension line mode is set to "Partial Length, Right Angled", this point also determines which side of the reference circle the dimension line should appear on (see graphic).



If "Display Dimension Line" is enabled:

4. *Enter dimension line position*
   The position of the dimension line is determined by the position of a point through which it should run. If a particular distance has been entered, then the point determines the approximate position of the dimension line. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If the dimension line mode is not "Partial Length, Right Angled", it is possible to enter a length. This determines the

distance between the dimension line and the length to be measured.

If the dimension line mode is set to "Partial Length, Right Angled", then this point determines the end point of the dimension line. A perpendicular is dropped from this point to the dimension line to work out the end point.

If "Center Dimension Text" or "Close Dimension Text" is not set:

5. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. The positioning of the dimension text can be restricted with the "Center Dimension Text" and "Close Dimension Text". Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

6. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified. An "Ø" symbol is placed in front of the dimension text.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

A dimension contains *two* property sets - one for the dimension line, and one for the dimension text. For example, this makes it possible to show the dimension text in a different color to the dimension line or assign the dimension text to a different layer from the dimension line.
If the command Edit Properties is used on a dimension, you can switch between the two property sets by clicking the corresponding tab. This allows you to view and alter both property sets separately.

Related commands:


Dimension>Radius, Points


Dimension>Radius, Object


Dimension>Diameter, Object

Dimension>Diameter, Object (Text Menu)

This command can be used to measure the diameter of a circle which exists independently or as part of an object's edge.

1. *Identify reference circle*
   Any circle which exists separately or as part of another object can be identified as the reference object. This can also be the arc of a circle segment or a circular arc within a surface.
   To identify a (part) object, click on it with the mouse. If more than one (part) object lies at the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the 'right' (part) object can be chosen.

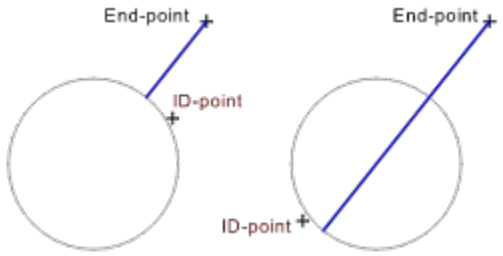
After selecting the reference circle to measure, the position of the dimension line and the dimension text must be specified. To do this, several points have to be entered. How many, and which, points have to be entered, depends on the specified dimension parameters.

When choosing the Change Parameters (+ESC) command, the Dimension With Line dialog appears where this command's parameters can be specified.

If "Display Dimension Line" is set and the dimension direction is set to "Any":

2. *Enter dimension line direction*
   The direction of the dimension line is determined by a point which, together with the center point of the reference circle, defines a straight line in whose direction the dimension runs. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension's direction.

   If the dimension line mode is set to "Partial Length, Right Angled", this point also determines which side of the reference circle the dimension line should appear on (see graphic).



If "Display Dimension Line" is enabled:

3. *Enter dimension line position*
   The position of the dimension line is determined by the position of a point through which it should run. If a particular distance has been entered, then the point determines the approximate position of the dimension line. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If the dimension line mode is not "Partial Length, Right Angled", it is possible to enter a length. This determines the distance between the dimension line and the length to be measured.

   If the dimension line mode is set to "Partial Length, Right Angled", then this point determines the end point of the dimension line. A perpendicular is dropped from this point to the dimension line to work

out the end point.

If "Center Dimension Text" or "Close Dimension Text" is not set:

4. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. The positioning of the dimension text can be restricted with the "Center Dimension Text" and "Close Dimension Text". Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

5. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified. An "Ø" symbol is placed in front of the dimension text.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

A dimension contains *two* property sets - one for the dimension line, and one for the dimension text. For example, this makes it possible to show the dimension text in a different color to the dimension line or assign the dimension text to a different layer from the dimension line.
If the command Edit Properties is used on a dimension, you can switch between the two property sets by clicking the corresponding tab. This allows you to view and alter both property sets separately.

Related commands:


Dimension>Radius, Points


Dimension>Radius, Object


Dimension>Diameter, Points

Dimension>Angle, Points (Text Menu)

This command can be used to measure an angle specified by entering the ends of two legs. This can also be used to measure angles which are not formed by existing object edges.

1. *Enter start point of leg 1*
   The dimension line's start point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter end point of leg 1*
   The dimension line's end point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter start point of leg 2*
   The dimension line's start point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

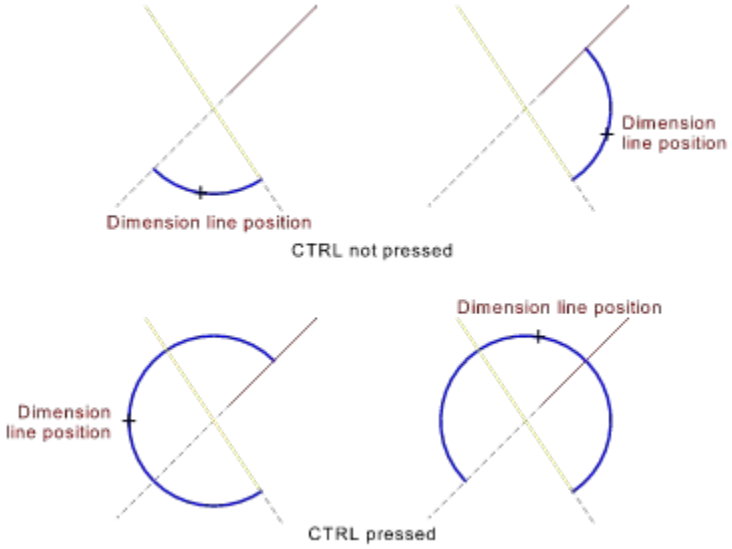
4. *Enter end point of leg 2*
   The dimension line's end point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After entering the legs of the angle to be measured, the position of the dimension line and the dimension text must be specified. To do this, several points have to be entered. How many, and which, points have to be entered, depends on the specified dimension parameters.

When choosing the Change Parameters (+ESC) command, the Dimension With Line dialog appears where this command's parameters can be specified.

5. *Enter dimension line position*
   The position of the dimension line is determined by the position of a point through which it should run. If a particular distance has been entered, then the point determines the approximate position of the dimension line. This point can be entered using the mouse, by clicking anywhere in the drawing. The dimension line position must always be entered, even if the dimension line display is disabled, since it determines the angle to be measured. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). It is also possible to enter the radius directly. This determines the radius of the dimension line.

   If during dimension line position entry the CTRL key is pressed, the "large" angle (above 180°) will be measured. Else, the "small" angle (below 180°) will be measured (see graphic).



If "Center Dimension Text" or "Close Dimension Text" is not set:

6. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. The positioning of the dimension text can be restricted with the "Center Dimension Text" and "Close Dimension Text". Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

7. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified. The "°" symbol is added after the dimension text.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

A dimension contains *two* property sets - one for the dimension line, and one for the dimension text. For example, this makes it possible to show the dimension text in a different color to the dimension line or assign the dimension text to a different layer from the dimension line.
If the command Edit Properties is used on a dimension, you can switch between the two property sets by clicking the corresponding tab. This allows you to view and alter both property sets separately.

Related commands:

Dimension>Angle, Line - Line

Dimension>Angle, Circle Part

Dimension>Angle, Line - Line (Text Menu)

This command can be used to measure an angle shown by the position of two legs. It can be used to measure angles formed by existing objects.

1. *Identify reference line 1*
   Any line in an existing object can be identified as the first reference line (first leg). This can also be the edge of a rectangle or a leg of a circular segment.

   To identify it, the line is clicked on with the mouse. If more than one line lies close to the position clicked on, a small dialog appears at the bottom of the screen, with the help of which the 'right' line can be chosen.

2. *Identify reference line 2*
   Any line in an existing object can be identified as the second reference line (second leg). This can also be the edge of a rectangle or a leg of a circular segment.

   To identify it, the line is clicked on with the mouse. If more than one line lies close to the position clicked on, a small dialog appears at the bottom of the screen, with the help of which the 'right' line can be chosen.

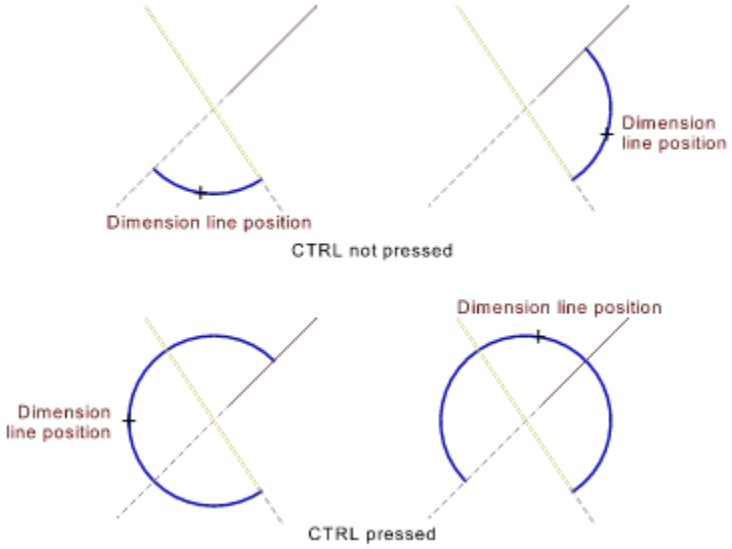
   The position of the ID point with reference to each leg determines which angle is measured.

After selecting the legs for the angle to be measured, the positions of the dimension line and the dimension text must be specified. To do this, several points have to be entered. How many, and which, points have to be entered, depends on the specified dimension parameters.

When choosing the Change Parameters (+ESC) command, the Dimension With Line dialog appears where this command's parameters can be specified.

3. *Enter dimension line position*
   The position of the dimension line is determined by the position of a point through which it should run. If a particular distance has been entered, then the point determines the approximate position of the dimension line. This point can be entered using the mouse, by clicking anywhere in the drawing. The dimension line position must always be entered, even if the dimension line display is disabled, since it determines the angle to be measured. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). It is also possible to enter the radius directly. This value determines the radius for the dimension line.

   If during dimension line position entry the CTRL key is pressed, the "large" angle (above 180°) will be measured. Else, the "small" angle (below 180°) will be measured (see graphic).



If "Center Dimension Text" or "Close Dimension Text" is not set:

4. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. The positioning of the dimension text can be restricted with the "Center Dimension Text" and "Close Dimension Text". Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

5. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified. The "°" symbol is added after the dimension text.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

A dimension contains *two* property sets - one for the dimension line, and one for the dimension text. For example, this makes it possible to show the dimension text in a different color to the dimension line or assign the dimension text to a different layer from the dimension line.
If the command Edit Properties is used on a dimension, you can switch between the two property sets by clicking the corresponding tab. This allows you to view and alter both property sets separately.

Related commands:

Dimension>Angle, Points

Dimension>Angle, Circle Part

Dimension>Angle, Circle Part (Text Menu)

This command can be used to measure the opening angle of an existing circle part.

1. *Identify reference circle part*
   Any circle part in an existing object (circular arc, sector or segment) can be identified as the reference circle part. This can also be a circular arc within a curve or a curved dimension line.

   To identify a circle part, click on it with the mouse. If more than one circle part lies at the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the 'right' circle part can be selected.

After selecting the circle part and therefore the angle to be measured, the position of the dimension line and the dimension text must be specified. To do this, several points have to be entered. How many, and which, points have to be entered, depends on the specified dimension parameters.

When choosing the Change Parameters (+ESC) command, the Dimension With Line dialog appears where this command's parameters can be specified.

2. *Enter dimension line position*
   The position of the dimension line is determined by the position of a point through which it should run. If a particular distance has been entered, then the point determines the approximate position of the dimension line. This point can be entered using the mouse, by clicking anywhere in the drawing. The dimension line position must always be entered, even if the dimension line display is disabled, since it determines the angle to be measured. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). It is also possible to enter the radius directly. This value determines the radius for the dimension line.

   If during dimension line position entry the CTRL key is pressed, the "large" angle (above 180°) will be measured. Else, the "small" angle (below 180°) will be measured.

If "Center Dimension Text" or "Close Dimension Text" is not set:

3. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. The positioning of the dimension text can be restricted with the "Center Dimension Text" and "Close Dimension Text". Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

4. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified. The "°" symbol is added after the dimension text.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults

is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

A dimension contains *two* property sets - one for the dimension line, and one for the dimension text. For example, this makes it possible to show the dimension text in a different color to the dimension line or assign the dimension text to a different layer from the dimension line.
If the command Edit Properties is used on a dimension, you can switch between the two property sets by clicking the corresponding tab. This allows you to view and alter both property sets separately.

Related commands:

Dimension>Angle, Points

Dimension>Angle, Line - Line

Dimension>Arc Length, Points (Text Menu)

This command can be used to measure the arc length of a circle part, shown by its center, radius, start and end points. It can also be used to measure the length of arcs which are not part of an existing circle part.

1. *Enter center point*
   The center of the reference circle part can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

2. *Enter radius*
   The reference circle's radius can be entered with the mouse by clicking anywhere in the drawing at a point through which the reference circle should run. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). It is also possible to enter the radius directly. This determines the radius of the reference circle.

3. *Enter start angle*
   The reference circle part's start angle can be entered using the mouse, by clicking anywhere in the drawing. The position of this point relative to the center point determines the start angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the reference circle part's start angle.

4. *Enter end angle*
   The reference circle part's end angle can be entered using the mouse, by clicking anywhere in the drawing. The position of this point relative to the center point determines the end angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the reference circle part's end angle. The angle can also be given relative to a previously entered start point.

   The reference circle part's appearance is determined by the current arc direction. This is specified using the command Toggle Arc Direction(F2), with the F2 key or the button in the panel. If the arc direction is positive, the circle part is drawn from the start angle to the end angle in a mathematically positive (i.e. anticlockwise) direction. If the arc direction is negative, the circle part is drawn from the start angle to the end angle in a mathematically negative (i.e. clockwise) direction (see graphic).

   The eventual appearance of the reference circle part is shown during entry, and changes if the arc direction is changed during entry.

After entering the circle part and the angle to be measured, the position of the dimension line and the dimension text must be specified. To do this, several points have to be entered. How many, and which, points have to be entered, depends on the specified dimension parameters.

When choosing the Change Parameters (+ESC) command, the Dimension With Line dialog appears where this command's parameters can be specified.

If "Display Dimension Line" is enabled:

5. *Enter dimension line position*
   The position of the dimension line is determined by the position of a point through which it should run. If a particular distance has been entered, then the point determines the approximate position of

the dimension line. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). It is also possible to enter the radius directly. This value determines the radius for the dimension line.

If "Center Dimension Text" or "Close Dimension Text" is not set:

6. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. The positioning of the dimension text can be restricted with the "Center Dimension Text" and "Close Dimension Text". Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

7. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

A dimension contains *two* property sets - one for the dimension line, and one for the dimension text. For example, this makes it possible to show the dimension text in a different color to the dimension line or assign the dimension text to a different layer from the dimension line.
If the command Edit Properties is used on a dimension, you can switch between the two property sets by clicking the corresponding tab. This allows you to view and alter both property sets separately.

Related commands:

Dimension>Arc Length, Circle Part

Calculate>Perimeter / Length

Dimension>Arc Length, Circle Part (Text Menu)

This command can be used to measure the arc length of an existing circle part.

1. *Identify reference circle part*
   Any circle part (arc, segment or sector) which exists can be identified. This can also be a circular arc within a curve or a curved dimension line.

   To identify a circle part, click on it with the mouse. If more than one circle part lies at the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the 'right' circle part can be selected.

After identifying the circle par and with it the angle to be measured, the position of the dimension line and the dimension text must be specified. To do this, several points have to be entered. How many, and which, points have to be entered, depends on the specified dimension parameters.

When choosing the Change Parameters (+ESC) command, the Dimension With Line dialog appears where this command's parameters can be specified.

If "Display Dimension Line" is enabled:

2. *Enter dimension line position*
   The position of the dimension line is determined by the position of a point through which it should run. If a particular distance has been entered, then the point determines the approximate position of the dimension line. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). It is also possible to enter the radius directly. This value determines the radius for the dimension line.

If "Center Dimension Text" or "Close Dimension Text" is not set:

3. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. The positioning of the dimension text can be restricted with the "Center Dimension Text" and "Close Dimension Text". Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

4. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

A dimension contains *two* property sets - one for the dimension line, and one for the dimension text. For example, this makes it possible to show the dimension text in a different color to the dimension line or assign the dimension text to a different layer from the dimension line.
If the command Edit Properties is used on a dimension, you can switch between the two property sets by clicking the corresponding tab. This allows you to view and alter both property sets separately.

Related commands:

Dimension>Arc Length, Points

Calculate>Perimeter / Length

Dimension>Coordinates (Text Menu)

This command can be used to measure the coordinates of a point.

1. *Enter reference point*
   The reference point whose dimensions are to be measured can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After entering the reference point, the position of the dimension text must be specified. To do this, one or two points (depending on the dimension parameters) must be entered.

When choosing the Change Parameters (+ESC) command, the Dimension Without Line dialog appears where this command's parameters can be specified.

2. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

3. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Coordinates and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified. A "(" appears in font of the text, a "/" in front of the mid text and a ")" after the text.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

Related command:

No related command

Calculate>Area (Text Menu) 

This command can be used to calculate the surface area of several selected objects and insert the value as a dimension. The surfaces are calculated once, and the total is then saved in a scale-independent form. The dimension is adapted to the current scale.

1. *Choose objects*
   The objects whose surfaces are to be calculated and added together have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing the objects, their surface areas are calculated. Complex objects with Bézier curves and ellipse parts must first be converted to polylines. The accuracy of the conversion can be set using the command Settings>Screen, "Accuracy - Transform" section.

The position of the dimension text must be specified. To do this, one or two points (depending on the dimension parameters) must be entered.

By choosing Change Parameters (+ESC) this command's parameters can be specified.

2. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

3. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified. The surface symbol appears in front of the text.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

Related command:

Calculate>Perimeter / Length

Calculate>Perimeter / Length (Text Menu)

This command can be used to measure the perimeters and side lengths of several selected objects and insert the value as a dimension. The perimeters and lengths are calculated once and the result is saved in a scale-independent form. The dimension displayed adapts to the scale in use.

1. *Choose objects*
   The objects whose perimeters and lengths are to be calculated and added together have to be chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing the objects their extent is calculated. Complex objects like Bézier curves and ellipses must first be converted to polylines. The accuracy of the conversion can be set using the command Settings>Screen, "Accuracy - Transform" section.

The position of the dimension text must be specified. To do this, one or two points (depending on the dimension parameters) must be entered.

By choosing Change Parameters (+ESC) this command's parameters can be specified.

2. *Enter dimension text position*
   The position of the dimension text can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If "Rotate Dimension Text" is set:

3. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

After entering all the parameters, the measurement is made. If the "Tolerances" check box was enabled in the dimension parameters, then the Dimension and Tolerances dialog appears, in which the tolerances and supplementary dimension text can be specified.

The dimension is not automatically assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The same goes for pens, which can be assigned using the command Pens>Defaults.

Related commands:

Dimension>Length, Object

Dimension>Arc Length, Circle Part

Calculate>Area

Dimension Parameters (Text Menu)

This command can be used to set the general dimension parameters. These parameters are valid for all types of measurement. They are changed from the Dimension Parameters dialog.

These parameters are applied to all new measurements. To alter the dimension properties of existing measurements, use the command Edit Properties.

Affected commands:

- Dimension>Length, Points
- Dimension>Length, Object
- Dimension>Distance, Object - Point
- Dimension>Distance, Object - Object
- Dimension>Radius, Points
- Dimension>Radius, Object
- Dimension>Diameter, Points
- Dimension>Diameter, Object
- Dimension>Angle, Points
- Dimension>Angle, Line Line
- Dimension>Angle, Circle Part
- Dimension>Arc Length, Points
- Dimension>Arc Length, Circle Part
- Dimension>Coordinates
- Calculate>Area
- Calculate>Perimeter / Length

Edit Dimension Line>Rotate (Text Menu)

This command can be used to draw alter the direction of an existing measurement. If there is one, the dimension line will be rotated accordingly.

1. *Identify measurement object*
   Any measurement object can be identified. Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

2. *Enter dimension line direction*
   The new dimension line direction is determined by a point which, together with the start point of the dimension defines a straight line in the direction of which the dimension should run. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension's direction.

If the duplicate function is active during the entry of the last point (e.g. because the CTRL key is held down), the dimension object is copied, and the copy (not the original) is altered.

After entering the new dimension line direction the measured object is adapted accordingly; the dimension line direction mode is set to "Any".

Related commands:


Edit Dimension Line>Position


Edit Dimension>Rotate


Edit Dimension>Position

Edit Dimension Line>Position (Text Menu)

This command can be used to change the position or length of an existing dimension line.

1. *Identify measurement object*
   Any measurement object can be identified. Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

2. *Enter dimension line position*
   The new dimension line position is determined by a point through which it should run. If the dimension line mode of the selected object is set to "Partial Length, Right Angled", then the point determines the end point of the dimension line. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). If the dimension line mode is not "Partial Length, Right Angled", it is possible to enter a length. This determines how much the dimension line should be moved by.

If the duplicate function is active during the entry of the last point (e.g. because the CTRL key is held down), a copy of the measurement object is made and the copy (not the original) is moved

After entering the new dimension line position, the object is adapted correspondingly; the dimension line distance mode is set to "Any".

Related commands:


Edit Dimension Line>Rotate


Edit Dimension>Rotate


Edit Dimension>Position

Edit Dimension>Rotate (Text Menu)

This command can be used to alter the rotation of an existing dimension text.

1. *Identify measurement object*
   Any measurement object can be identified. Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

4. *Enter dimension text direction*
   The dimension text direction can be entered using the mouse, by clicking anywhere in the drawing. The location of this position relative to the dimension text position determines the dimension text's rotation angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, it is possible to enter an angle. This determines the dimension text's rotation angle.

If the duplicate function is active during the entry of the last point (e.g. because the CTRL key is held down), a copy of the measurement object is made and the copy (not the original) is moved.

After entering the new dimension text position, the measurement object is rotated accordingly. The dimension text mode is set to "Rotated".

Related commands:

 Edit Dimension>Position

 Edit Dimension>Update

 Edit Dimension Line>Rotate

 Edit Dimension Line>Position

Edit Dimension>Position (Text Menu)

This command can be used to draw alter the position of an existing piece of dimension text.

1. *Identify measurement object*
   Any measurement object can be identified. Objects are identified by clicking on them. If more than one object lies at the point clicked on, then the correct object can be selected with the help of a small dialog which appears at the bottom of the screen.

2. *Enter dimension text position*
   The dimension text's new position is determined by entering a point which places the mid point of the text base line. This point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

If the duplicate function is active during the entry of the last point (e.g. because the CTRL key is held down), the object itself is not altered, but a copy is made and the copy is altered.

After entering the new dimension text position, the measurement object is adapted correspondingly and, if set, the "Close" and "Centered" dimension text modes are cleared.

Related commands:


Edit Dimension>Rotate


Edit Dimension>Update


Edit Dimension Line>Rotate


Edit Dimension Line>Position

Edit Dimension>Update (Text Menu)

This command can be used to recalculate selected measurements, altering the dimension text, for example to adapt them to a new scale or coordinate system.

1. *Choose dimension texts*
   The dimension objects whose dimensions should be recalculated are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing the objects to be measured, their dimensions are recalculated and inserted. This happens whether the "Adapt" dimension text mode is active or not.

The "Adapt" dimension text mode is not altered by this command. If dimension texts should be recalculated automatically from now on, the command Edit Properties should be used to enable the "Adapt" dimension text mode.

Related commands:


Edit Dimension>Rotate


Edit Dimension>Position


Edit Dimension Line>Rotate


Edit Dimension Line>Position

Fonts (Text Menu)

This command can be used to set the font parameters. In addition, TommySoftware® CAD/DRAW's own fonts can be loaded into or removed from memory. To do this, the Fonts dialog appears.

The actions chosen in this dialog are carried out when the dialog is closed by clicking on OK. Afterwards, the drawing windows are redrawn.

The list of loaded fonts is saved permanently. All fonts which are loaded when the program is exited, are automatically loaded when the program is next started.

Related command:

Libraries

Library Menu

Block
- Insert...
- Create (Insertion Point)
- Create (Frame)
- Edit

Position Number
- Assign
- Delete
- Reorder

Generate Parts List
Clean Up Internal Blocks
Convert External Blocks
Resolve Instances
Replace Block Name
Replace Library Name
Libraries

Block>Insert (Library Menu)

This command can be used to insert an instance of a block into a drawing. Instances of blocks which are not currently in memory can be inserted. These will (as long as automatic library loading is enabled) be loaded automatically.

1. *Choose block*
   After choosing this command the Block>Insert dialog appears, where the block to insert can be chosen. The choice of block and parameters remains active until either a new command is chosen or the current command is canceled or restarted using Change Parameters (+ESC) or the key combination SHIFT +ESC.

After closing the dialog by clicking on "OK", a rectangle with a cross through it appears at the cursor. The rectangle represents the block to be inserted. It is already rotated and sized according to the chosen settings.

2. *Enter position*
   The point at which the block is inserted is chosen by clicking the mouse at any chosen point within the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After specifying the insertion point, an instance of the chosen block is created. If the "Edit" check box in the "Local Attributes" section has been enabled, the Define Attributes dialog appears in which the local attributes of the instance can be edited. If this dialog is closed by clicking on the "Cancel" button, the instance is still created. The alterations to the attributes will however be ignored.

The instance which is generated will not automatically be assigned to the current layer. Instead, the command Layers>Defaults is used to decide whether they should be assigned to the current layer, or another specified layer. The instance is drawn using the current pen and the appearance of the instanced block is not changed.

An instance of a block can only be displayed if the relevant block is loaded to memory. If, at a later time, this is not the case (e.g. because the relevant library is not available), a text in the form "(Library Name)Block Name" is shown at the insertion point. The instance itself can however be manipulated as usual. Local attributes can be edited and commands like copying, rotating and scaling work as normal. As soon as the block is available again, the instance is displayed again.
If a drawing is going to be given to someone who does not have the library, all external blocks should be converted to internal blocks (see Convert External Blocks).

Related commands:

Block>Create (Insertion Point)

Block>Create (Frame)

Block>Create (Insertion Point) (Library Menu)

This command can be used to create a new block. After selecting the objects and instances which are to make up the block, an insertion point is entered.

1. *Choose objects*
   The objects and instances which are to make up the new block are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The block's insertion point can be entered using the mouse, by clicking anywhere in the drawing.

   The insertion point should be placed at a distinct place in the block, e.g. at the end point of a line or the corner point of a rectangle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

After entering the insertion point, the Generate Block dialog appears, in which the name of the block to be generated can be entered. In addition, some parameters can be specified. After closing the dialog by clicking on "OK", the block is generated.

If the "Edit Attributes" check box is enabled in the parameters dialog, then the Define Attributes dialog appears, in which the local and global attributes of the block can be edited. If this dialog is closed with the "Cancel" button, the block is still generated! The alterations to the attributes will however be ignored.

If the "Edit Properties" check box in the parameters dialog is enabled, then the Edit Properties dialog appears, in which the properties of the block can be edited. If this dialog is closed with the "Cancel" button, the block is still generated! The alterations to the properties will however be ignored.

Blocks may themselves contain instances of other blocks and groups, i.e. nested instances are possible. This makes it possible, for example, to create components made up of components from several blocks from different libraries. There are however two restrictions, which should help avoid problems.:

Internal blocks and groups cannot be used in external blocks! This would make the block, and the whole library dependent on the drawing which contains the internal block or group.

The maximum number of levels which can be nested is 10. Deeper nestings are possible, but they will not be displayed. If a circular reference is accidentally created (block X contains an instance of Y, block Y contains an instance of X), this is the same as an unending nesting and the blocks will not be displayed.
In general, the use of nesting within libraries should be avoided if possible. They make the library more difficult to maintain and less clear.

Related commands:

Block>Insert

Block>Create (Frame)

Group>Create Group

Block>Create (Frame) (Library Menu)

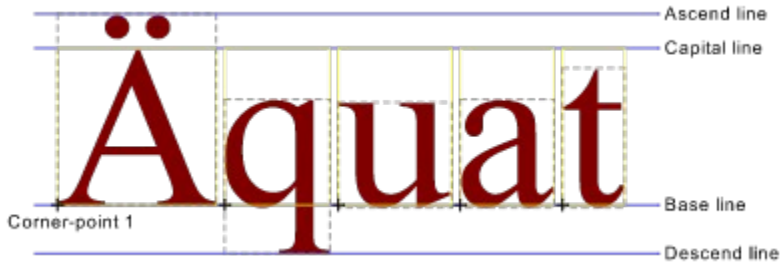
This command can be used to create a new block. After selecting the objects and instances which should make up the block, an area which specifies the extent of the block is entered

It is only necessary to generate blocks using a frame if the block is to be used as a character in a TommySoftware® CAD/DRAW font. For more information on generating fonts, refer to the description of font libraries in the help file `TVG40.HLP`.

1. *Choose objects*
   The objects and instances which are to make up the new block are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

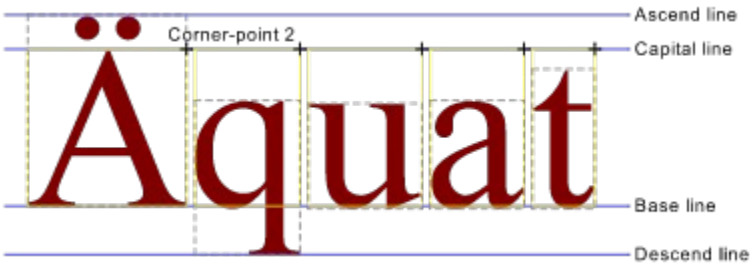
2. *Area: Enter corner point 1*
   The first corner point of the area can be entered using the mouse, by clicking anywhere in the drawing. This corner point also shows the block's insertion point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

   This must be the left limit on the baseline of a character (see graphic). This point does not have to be on the frame surrounding the block.



3. *Area: Enter corner point 2*
   The area's second corner point can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

   For a font character, this point should be the right limit of the character at the height of a capital A (ascend line, see graphic). This point does not have to be on the frame surrounding the block.



After entering the area, the Generate Block dialog appears, in which the name of the block to be generated can be entered. In addition, some parameters can be specified. After closing the dialog by clicking on "OK", the block is generated.

If the "Edit Attributes" check box is enabled in the parameters dialog, then the Define Attributes, in which the local and global attributes of the block can be edited. If this dialog is closed with the "Cancel" button, the block is still generated! The alterations to the attributes will however be ignored.

If the "Edit Properties" check box in the parameters dialog is enabled, then the Edit Properties dialog appears, in which the properties of the block can be edited. If this dialog is closed with the "Cancel" button, the block is still generated! The alterations to the properties will however be ignored.


Blocks may themselves contain instances of other blocks and groups, i.e. nested instances are possible. This makes it possible, for example, to create components made up of components from several blocks from different libraries. There are however two restrictions, which should help avoid problems.:


Internal blocks and groups cannot be used in external blocks! This would make the block, and the whole library dependent on the drawing which contains the internal block or group.


The maximum number of levels which can be nested is 10. Deeper nestings are possible, but they will not be displayed. If a circular reference is accidentally created (block X contains an instance of Y, block Y contains an instance of X), this is the same as an unending nesting and the blocks will not be displayed.
In general, the use of nesting within libraries should be avoided if possible. They make the library more difficult to maintain and less clear.


Related commands:


Block>Insert


Block>Create (Insertion Point)


Group>Create Group

Block>Edit (Library Menu)

This command can be used to edit blocks and folders. They can be deleted, copied and renamed and in addition the attributes and properties of individual blocks can be edited. This is done from the Edit Block dialog.

The operations carried out in this dialog take immediate effect on the blocks in memory and can not be undone (there is no "Cancel" button). Be very careful when using this command!
Blocks deleted with this command can not be recovered using Undo. If blocks are accidentally deleted, the libraries which are affected should be updated using the Libraries command, without previously saving the changes.

Related commands:

Convert External Blocks

Replace Block Name

Replace Library Name

Position Number>Assign (Library Menu) 

This command can be used to assign a position number to chosen objects. This is done by creating a group containing the chosen objects and a reference text. In addition, this group contains an attribute, the actual position number.

1. *Choose objects*
   The objects and instances to which a position number shall be assigned are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

2. *Enter reference point*
   The reference point, i.e. the end-point of the reference line, can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

3. *Enter destination point*
   The destination point, i.e. the position of the reference text filed, can be entered using the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

By choosing Change Parameters (+ESC) this command's parameters can be specified.

After destination point entry, a dialog window appears where you can enter the desired position number. It is initialized with the smallest position number that has not been used yet.

When the position number has been entered, the chosen objects will be combined to a block and associated with the position number.

Related commands:

 Position Number>Delete

 Position Number>Reorder

Position Number>Delete (Library Menu) 

This command can be used to delete position numbers. In order to do so, the previously created groups containing objects and the position number are chosen. Those groups will then be resolved, and reference texts displaying the position number are deleted.

1. *Choose instances*
   The instances of those position number groups to be resolved are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing the instances, all position number found in the chosen instances will be resolved, i.e. the position numbers themselves will be erased, and the blocks containing the associated objects will be resolved.

Related commands:

 Position Number>Assign

 Position Number>Reorder

Position Number>Reorder (Library Menu) 

This command can be used to reorder all existing position numbers. Reordering means that the numerical values of the position numbers will be updated so that no "gaps" exist.

After choosing this command a dialog appears, prompting for whether to keep equal position numbers equal or not. If you choose "Yes", all position numbers that had equal values before will still have equal values afterwards - but not necessarily the same values as before!

Related commands:


Position Number>Assign


Position Number>Delete

Generate Parts List (Library Menu) 

This command can be used to generate a parts list. A parts list is a tabular list of local, global, and implicit attributes defined in blocks and instances in the current drawing as well as the currently loaded libraries. To do this, the dialog Generate Parts List appears, in which the parameters for the parts list can be specified.

A parts list can also be generated if neither global nor local attributes have been defined. Each instance has its own implicit attributes (block name, library name, position etc.), which are always available, and these can be used to generate a parts list.

Related command:

No related command

Clean Up Internal Blocks (Library Menu)

This command clears up the internal blocks in the current drawing. This means that all internal blocks which are currently defined but not in use are deleted.

Before this command is carried out, you are asked for confirmation, to avoid the accidental deletion of internal blocks.

Groups are not deleted by this command, even though they are a special type of internal block. Groups are cleared out when drawings are loaded and saved.

Internal Blocks deleted with this command can not be recovered using Undo. If internal blocks are accidentally removed they can usually be restored from the last saved version of the drawing using the command Import>TVG 4.0 (*.T4G).

Related command:

No related command

Convert External Blocks (Library Menu)

This command converts external blocks currently used in the drawing to internal blocks. The Convert External Blocks dialog appears, in which the external blocks to be converted can be specified and where you also can specify whether the instances of these blocks are to be adapted.

If a drawing which makes use of external blocks from libraries is to be given to someone who does not have those libraries, then there can be problems. Instances which refer to those external blocks cannot be created and the drawing is useless.

If however the external blocks are converted to internal blocks, the drawing can still be used. The advantages of instances (low memory use and changeability) are retained.

The original external blocks are unchanged, new blocks are however created. Conversion of external blocks to internal blocks can not be undone using Undo. The adaptation of the instances can however be undone.

Related commands:


Block>Edit


Replace Block Name


Replace Library Name

Resolve Instances (Library Menu)

This command can resolve instances of blocks. This means that the objects from which the blocks are made up are inserted into the drawing instead of the instance.

All the settings in the instance (position, rotation angle, scaling, transmission of properties) are kept, so that the newly created objects may be different from the original objects which made up the block.

This command cannot be used to resolve groups. This must be done using the Group>Ungroup command.

1. *Choose instances*
   Those instances to be resolved are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

After choosing, the instances are resolved. Only one layer is resolved at a time. If the blocks which the selected instances refer to contain other instances, then they are not resolved. If neccessary this must be done in a second step.

If the duplicate function is active while the objects are chosen, the chosen instances are copied and the copy is resolved.

If there are no instances among the selected objects, then a warning appears. If all the instances in the object are to be resolved, this can be done by pressing the F10 Key to select all objects, choosing the command and repeating the process until the warning appears.

Some objects cannot be distorted. These are, above all, circles and circle parts, but also curves and surfaces which contain circular arcs. If an instance which is to be resolved contains such an onject in distorted form, then the circles and circle parts are converted to ellipses and ellipse parts during resolution.
Circular arcs within curves and surfaces can not be transformed. This leads, unavoidably, to problems in the display.

Related commands:


Group>Ungroup


Trim Objects>Resolve Completely


Text>Resolve

Replace Block Name (Library Menu)

This command replaces instances of particular blocks with other instances. This is done by altering the reference contained in the instance. The display parameters (position, rotation angle, scaling) and the local attributes of the instance remain unchanged.

The Replace Block Name dialog appears, in which you can specify which block is to replaced by which other block.

As only instances are affected by this command, the substitution can be undone using the Undo command

Related commands:

- Convert External Blocks
- Replace Library Name
- Edit Properties

Replace Library Name (Library Menu)

This command replaces instances of blocks from one library with instances of identically named blocks from another library. This is done by altering the reference contained in the instance. The display parameters (position, rotation angle, scaling) and the local attributes of the instance remain unchanged.

The Replace Library Name dialog appears where you can specify which library is to be replaced by which other library.

This replacement can only be successful if both libraries contain identically named blocks. If this is not the case, the references in the instances are changed and may point to a non-existent block. The program does not check this beforehand!

As only instances are affected by this command, the substitution can be undone using the Undo command

Related commands:

- Convert External Blocks
- Replace Block Name
- Edit Properties

Libraries (Library Menu)

This command can be used to set the library parameters. In addition, libraries can be loaded, saved, updated or removed from memory. This is done from the Libraries dialog.

The actions specified in this dialog are carried out as soon as the "OK" button is clicked on. After this, all the drawing windows are redrawn.

The list of loaded libraries will be permanently saved. All libraries which are loaded when the program is closed will be automatically loaded when the program is next started. External blocks from libraries which use the cache are not initially loaded.

Related command:

Fonts

Extra Menu

- Identify All (F10)
- Previous Identification (F11)
- Use Permanent Selection (F12)
- Permanent Selection
  - Set
  - Clear
  - Invert
- Toggle Area Mode (+F6)
- Coordinate Entry (F8)
- Previous Point (ESC)
- Change Parameters (+ESC)
- Duplicate / Multiline (F7)
- Orthogonal Mode (F5)
- Orthogonal Mode Parameters (+F5)
- Snap Modes
  - Snap Active (F6)
  - Center
  - Quadrant
  - Edge
  - Corner / End-Point
  - Intersection
  - Geometry
  - Marking
  - Other Point
  - Relative
  - Snap Radius
- Control Keys
  - Window Refresh (SPACE)
  - Left Mouse Button
  - Middle Mouse Button
  - Right Mouse Button
  - Menu Keys (ALT)
  - Step Left (ARROWLEFT)
  - Step Right (ARROWRIGHT)
  - Step Up (ARROWUP)
  - Step Down (ARROWDOWN)
  - Page Left (HOME)
  - Page Right (END)
  - Page Up (PAGEUP)
  - Page Down (PAGEDOWN)
  - Edit Step

Identify All (F10) (Extra Menu)

This command can be used to select *all* objects and instances in a drawing. If a Selection Filter is in use, then only objects fulfilling the specified conditions are selected.

Objects in hidden or frozen layers are not selected.

Related commands:

Previous Identification (F11)

Use Permanent Selection (F12)

Previous Identification (F11) (Extra Menu)

This command can be used to select all objects that were previously identified.

Some commands restrict the information about previously selected objects to certain groups of objects. For example, if the command Paste was used to insert objects from the clipboard, then the relevant information is available for these (and *only* these) objects. If the pasted objects are to be moved immediately after pasting, choose the Move Objects>Standard command and press the F11 key.

Related commands:

Identify All (F10)

Use Permanent Selection (F12)

Use Permanent Selection (F12) (Extra Menu)

This command can be used to identify all at the moment permanently selected objects.

In this way a permanent selection can be used as a container for a frequently required group of objects. If you stop working on a group for a moment you can mark the group by permanently selecting its objects to continue working on it later. For that choose the command Permanent Selection>Set and press the F11 Key. This permanently selects all previously identified objects. You can now identify other objects and work with them without affecting the permanent selection

If you now want to work with the permanently selected objects again, press the F12 key to identify them. If the permanent selection is no longer required, it can be deleted using the command Permanent Selection>Clear.

Related commands:

Identify All (F10)

Previous Identification (F11)

Permanent Selection>Set (Extra Menu)

This command can be used to permanently select the specified objects.

Permanent selection has two uses: Firstly, a group of objects can be marked by the permanent selection. It can then be used later as the basis for normal object selection. This can be done with the F12 Key.

Secondly the selected group of objects can be used as an additional object selection alongside the normal, directly selected objects. This is the case with the command Multiple Copy>Markings.

1. *Choose objects*
   The objects to be permanently selected are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

The chosen objects are then included in the permanent selection. The permanent selection remains active until it is explicitly changed. However, permanent selections are not saved with drawings and are therefore not available when a drawing is opened.

Related commands:

Permanent Selection>Clear

Permanent Selection>Invert

Permanent Selection>Clear (Extra Menu)

This command can be used to clear the permanent selection of the specified objects.

Permanent selection has two uses: Firstly, a group of objects can be marked by the permanent selection. It can then be used later as the basis for normal object selection. This can be done with the F12 Key.

Secondly the selected group of objects can be used as an additional object selection alongside the normal, directly selected objects. This is the case with the command Multiple Copy>Markings.

1. *Choose objects*
   The objects to be deselected are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

Next, the permanent selection of the chosen objects is cleared.

Related commands:


Permanent Selection>Set


Permanent Selection>Invert

Permanent Selection>Invert (Extra Menu)

This command can be used to invert the permanent selection of the specified objects. Objects which were previously permanently selected are deselected and vice versa.

Permanent selection has two uses: Firstly, a group of objects can be marked by the permanent selection. It can then be used later as the basis for normal object selection. This can be done with the F12 Key.

Secondly the selected group of objects can be used as an additional object selection alongside the normal, directly selected objects. This is the case with the command Multiple Copy>Markings.

1. *Choose objects*
   The objects of which the permanent selection is to be inverted are chosen. Click on objects to choose them. For further information on object selection, see the chapter Object Selection.

Next, the permanent selection of the chosen objects is inverted.

Related commands:

Permanent Selection>Set

Permanent Selection>Clear

Toggle Area Mode (+F6) (Extra Menu)

This command can be used to toggle the current area mode. The area mode determines whether, when selecting objects using areas (pressing the SHIFT key), the objects to be selected must be completely within the specified area or only have to overlap the area partly.

The area mode can also be changed by left-clicking the button in the panel shown to the left. The button also shows the current area mode.

Related Command:

No related commands

Coordinate Entry (F8) (Extra Menu)

Using this command, specific numeric coordinates can be entered, instead of using the mouse.

The coordinates are entered in the Status Line. Direct coordinate entry can also be called by clicking the left mouse button in the status line, by pressing the F8 key, or (if using the standard keyboard assignments) by pressing ENTER.

During entry, the input focus can be moved using the TAB key. Entry is confirmed by pressing RETURN or interrupted by pressing ESC. The value or value pair that is active when confirming the input will be used to calculate the final coordinates.

If the status line is not already displayed when this command is chosen, it appears and then disappears when the coordinates have been entered (see also Windows>Display Status Line).

Standard Examples

These standard examples illustrate the most important input possibilities. Not all the possibilities shown are always available. Relative angles and lengths can only be entered if there is an obvious reference. If need be, attention is drawn to this in the description of the relevant command in the help system.

Standard Coordinate Entry

Input: x= `150` y= `100`
Result: A point is inserted at the coordinates (150;100) with reference to the current coordinate system.

Input: x= `100+30/4` y= `7*18`
Result: A point is inserted at the coordinates (107.5;126) with reference to the current coordinate system.

Input: x= `x+150` y= `y+100`
Result: A point is inserted 150 Units to the right of the reference point and 100 units above the reference point.

Input: dx= `0` dy= `100`
Result: A point is inserted 100 Units to the right of the reference point.

Input: a= `45` l= `2*50`
Result: A point is inserted 100 units at an angle of 45° from the reference point.

Input: x= `15cm` y=`10cm`
Result: A point with the coordinates (15 cm;10 cm) with reference to the current coordinate system is inserted - independent of the current unit.

Radius Entry

Input: r= `15`
Result: The object to be entered is given a radius of 15 units.

Input: r= `r+10`
Result: The object to be entered is given a radius 10 units larger than the reference object.

Input: x= 150 y= 100
Result: A point is inserted at coordinates (150;100) with reference to the current coordinate system.

Input: dx= 150 dy= 100
Result: A point is inserted 150 Units to the right of the reference point and 100 units above the reference point. The reference point is the center point of the previously entered object.

Length Entry

An arrow is shown over the icon in the status window. This arrow shows the current direction for length measurements. Positive lengths run in the direction of the arrow, negative lengths in the opposite direction.

Input: l= 30
Result: The object to be entered is given a length of 30 units. The length is entered in the direction which the arrow in the status window shows.

Input: l= -50mm
Result: The object to be entered is given a length of 50 mm - regardless of the current unit. The length is entered in the opposite direction to that in which the arrow in the status window shows.

Angle and Length Entry

Input: a= 30 l=120
Result: The object has an angle of 30° and a length of 120 units.

Full Description

Entry takes the form of references to particular variables. According to which variables the value refers to, they are interpreted differently.

Variables and Terms

There are six variables to which values can be assigned.

x	Absolute X-coordinate of a point
y	Absolute Y-coordinate of a point
dx	Relative X-coordinate of a point (the same as x=x+)
dy	Relative Y-coordinate of a point (the same as y=y+)
l	Length, amount of a polar coordinate
r	Radius (handled like l)
a	Angle
w	Angle (handled like a)

Each variable is given a value in the form of a term. This is done in a similar way to writing mathematical

equations with an equals sign. This means that terms can be used which contain numeric values, brackets, operators and functions. Possible operators are:

`+`	Addition	Example: `8+2`
`-`	Subtraction	Example: `10-2`
`*`	Multiplication	Example: `3*4`
`/`	Division	Example: `15/3`
`^`	Power	Example: `2^8`

Arguments of functions must be in brackets. The available functions are:

`sqr`	Square
`sqrt`	Square Root
`ln`	Natural Logarithm
`sin`	Sine
`cos`	Cosine
`tan`	Tangent
`arcsin`	Arcsine
`arccos`	Arccosin
`arctan`	Arctan
`ceil`	smallest whole number larger than or equal to the argument
`floor`	largest whole number smaller than or equal to the argument

Constants

According to the type of input being carried out, certain constants are available which can be used in the terms. They allow, for example, relative values to be entered. The following constants are available.

`x` and `y`	current X- and Y-coordinates of the reference point
`l`	current length of the reference object or previously entered line section
`r`	current radius of the reference object
`a` and `w`	current angle of the reference object or previously entered line section
`_fl`	"**F**rame **L**eft"
`_fr`	"**F**rame **R**ight"
`_ft`	"**F**rame **T**op"
`_fb`	"**F**rame **B**ottom"
	Rectangular coordinates of the frame which surrounds all currently selected objects.
`_pl`	"**P**age **L**eft"
`_pr`	"**P**age **R**ight"
`_pt`	"**P**age **T**op"
`_pb`	"**P**age **B**ottom"
	Corner coordinates of the current page.
`e`	base number for natural logarithms (2.7182...)
`pi`	pi (π) (3.1415...)

Units

All values entered without a specific unit relate to the current length or angle unit of the active window's

coordinate system.

To specify a unit for a value, the appropriate abbreviation should be used. These are the possible units and their abbreviations.

µm	Micrometer	1/1000 mm
mm	Millimeter	1 mm
cm	Centimeter	10 mm
dm	Decimeter	100 mm
m	Meter	1000 mm
km	Kilometer	1000000 mm
mil	Mil	1/1000 in
in	Inch	1 in = 25,4 mm
ft	Foot	12 in
yd	Yard	3 ft
ml	Mile	1760 yd
dp	Decipoint	1/720 in
pt	Point	1/72 in
bp	Didot	1/2660 m
cic	Cicero	12/2660 m

Possible angle units and abbreviations

deg	Degree	360 per rotation
gra	Gon	400 per rotation
rad	Radian	2pi per rotation
rel	Relative	1 per rotation

Units can be mixed within a term. These are examples using `mm` as the current length unit and `deg` as the current angle unit.

`100+1in`	is the same as	100 mm + 1 in = 125,4 mm
`(100+1)in`	is the same as	101 in = 2565,4 mm
`2yd+1ft+9,5in`	is the same as	93,5 in = 2374,9 mm
`sin(pi rad)`	is the same as	sin(180°) = 0.0

Examples

Start Point at Line>Standard:

x= `150`	y=`200`	These entries both describe the point with an X-coordinate of 150 and a Y-coordinate of 200.
a= `45`	l= `60`	This entry describes the location of a point 60 units at a direction of 45° from the origin. The polar coordinates are calculated relative to the origin because there is no reference point.

End Point at Line>Standard:

x= `150`	y= `200`	These entries both describe the point with an X-coordinate of 150 and a Y-coordinate of 200.

x= x+50	y= y+100	
dx= 50	dy= 100	These entries describe a point whose position is 50 units to the right of and 100 units above the lines start point. The relative coordinates are calculated relative to the start point of the line.
a= 45	l= 60	This entry describes the location of a point 60 units at a direction of 45° from the start point of the line. The polar coordinates are calculated relative to the start point of the line.

End Point at a Polyline:

At least the start point and end point have already been entered.

x= 150	y= 200	These entries describe a point with an X-coordinate of 150 and a Y-coordinate of 200.
x= x+50	y= y+100	
dx= 50	dy= 100	These entries describe the position of a point 50 units to the right of the current part line's start point and 100 units above it. The relative coordinates are calculated relative to the current line's start point.
a= 45	l= 60	This entry describes the location of a point 60 units at a direction of 45° from the start point of the part line. The polar coordinates are calculated relative to the start point of the part line.
a= a+45	l= 1	This entry describes the end point of a part line with an angle of 45° relative to the previous part line and whose length is the same as the preceding part line. Angles and lengths are calculated relative to the preceding part line.

Length entered with Trim Object>Length to Point:

A line is selected as the object to be trimmed. An arrow is shown over the icon in the status window. This arrow shows the current direction for length measurements. Positive lengths run in the direction of the arrow, negative lengths in the opposite direction.

l= 80		The selected end of the line is moved so that the line has a length of 80 units. The length is measured from the other end of the line in the direction indicated by the arrow in the status window.
l= l+20		The selected end of the line is moved so that the line is lengthened by 20 units.
l= 2*l		The selected end of the line is moved so that the length of the line doubles.
x= 150	y= 200	The line ends at the foot of a perpendicular dropped from a point with the X-coordinate 150 and the Y-coordinate 200 on to the reference straight line.
x= x+50	y= y+100	
dx= 50	dy= 100	The line ends at the foot of a perpendicular dropped onto the reference straight line from a point 50 units to the right and 100 units above its start point. The

relative coordinates are relative to the start point of the line.

Radius entered with Circle>Concentric:

A circle is selected as a reference object.

r= 50		The circle is given a radius of 50 units.
r= r+20		The circle is given a radius, 20 units larger than the reference circle.
r= 2*r		The circle is given a radius twice as large as that of the reference circle.
x= 150	y= 200	The circle runs through the point at the X-coordinate 150 and the Y-coordinate 200.
x= x+50	y= y+100	
dx= 50	dy= 100	The circle runs through a point 50 units to the right and 100 units above the center of the reference circle.

Angle entered with Trim Object>Angle to Point:

A circular arc is selected as the object to be trimmed.

a= 45		The selected end of the circular arc is lengthened or shortened to an angle of 45°.
a= a+10		The selected end of the circular arc is moved by 10° anticlockwise.
a= a-15		The selected end of the circular arc is moved by 15° clockwise.
x= 150	y=200	The selected end of the circular arc has its position determined by the X-coordinate 150 and the Y-coordinate 200.
x= x+50	y= y+100	
dx= 50	dy= 100	The selected end of the circular arc has its position determined by a point 50 units to the right of and 100 units above the center of the reference circle.

Combined Angle and Length entry with Line>Perpendicular:

A circle is selected as the reference object.

a= 30	l= 100	The perpendicular runs 100 units from the circle at an angle of 30°.
x= 150	y= 200	The perpendicular ends at the point with the X-coordinate 150 and the Y-coordinate 200.
x= x+50	y= y+100	
dx= 50	dy= 100	The perpendicular ends at a point 50 units to the right of and 100 units above the center of the reference circle.

Related command:

No related commands

Previous Point (ESC) (Extra Menu)

This command can be used to undo the last point entry. This only works during execution of a command

All previous entries can and must be redone. This may include parameter settings and object selection.

Related command:

No related commands

Change Parameters (+ESC) (Extra Menu)

This command can be used to edit the current command's parameters. If available, a dialog windows will appear in which the parameters can be viewed and edited. See Change Parameters of Current Command for details.

Usually, a command will have to be restarted after its parameters have been altered. If, e.g., a dimension parameter has been altered, the currently active dimension entry has to be started from the beginning.

This command can also be activated by left-clicking the button in the panel shown to the left. The button always shows the current command's icon.

Related command:

No related commands

Duplicate / Multiline (F7) (Extra Menu)

This command can be used to turn the duplicate / multiline function on or off. Depending on the currently active command this function will either operate as the "duplicate function" or as the "multiline function".


When modifying existing objects (e.g. scaling or rotating), the "duplicate function" is used. It determines whether the objects are modified themselves or whether a duplicate is generated and modified instead. If the "duplicate function" is on, a small plus sign + is shown above and to the right of the crosshair intersection.


When creating new objects (e.g. drawing a line or circle), the "multiline function" is used. It determines whether several parallel or concentric objects are created at once (depending on the multiline parameters) or not. If the "multiline function" is on, three small parallel lines /// are shown above and to the right of the crosshair intersection. To edit the multiline parameters, use the Change Parameters (+ESC) command during a command's execution.


The duplicate / multiline function can be turned on or off by left-clicking the button in the panel shown to the left. The button shows the current status of the duplicate / multiline function.

During point entry the duplicate / multiline function can be briefly (de)activated by holding down the CTRL key.

Related command:

No related commands

Orthogonal Mode (F5) (Extra Menu)

This command can be used to turn the orthogonal mode on or off.

 The orthogonal mode can be turned on or off by left-clicking the button in the panel shown to the left. The button shows the current status of the orthogonal mode.

Related command:

 No related commands

Orthogonal Mode Parameters (+F5) (Extra Menu)

This command can be used to edit the orthogonal mode parameters.

When choosing this command, the Orthogonal Mode Parameters dialog appears.

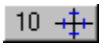
 The orthogonal mode parameters can be edited by right-clicking the button in the panel shown to the left. The button shows the current status of the orthogonal mode.

Related command:

 No related commands

Snap Mode>Snap Active (F6) (Extra Menu)

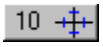
This command can be used to turn the snapping function on and off. It can only be turned on if at least one snap mode is active. If the snapping function is active, points which are entered 'jump', according to the snap mode, to a defined point in the drawing or a point is inserted at a position calculated according to object identification.



Snapping can also be turned on or off by left-clicking the button in the panel shown to the left. The button shows the snapping function's current status.

During point entry, the snapping function can be briefly (de)activated by holding down the SHIFT key.

If snapping is on, four small lines are shown on the crosshair. Their distance from the crosshair's intersection shows the current Snap Radius an. This determines how closely a point must be clicked to in order to use it as a snapping reference.



The snap radius can also be set by right-clicking the button in the panel shown to the left. The value shown in the button is the snap radius.

Related commands:

- Snap Modes>Center
- Snap Modes>Quadrant
- Snap Modes>Edge
- Snap Modes>Corner / End-Point
- Snap Modes>Intersection
- Snap Modes>Geometry
- Snap Modes>Marking
- Snap Modes>Other Point
- Snap Modes>Relative
- Snap Modes>Snap Radius

Snap Modes>Center (Extra Menu)

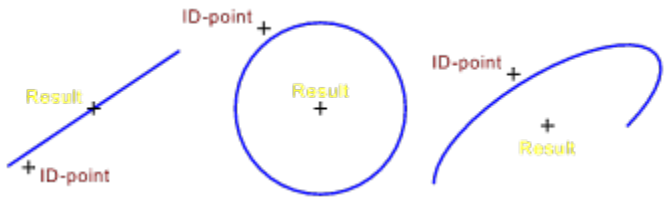
The snap modes determine which type of point should be used as a snapping reference. There are various snap modes, some of which can be used together.

The snap modes take effect when snapping is active. Snapping can be turned on or off using the F6 Key or the first button in the Panel. It can be temporarily (de)activated during point entry using the SHIFT key.

This snap mode uses center points as snapping references. These can be the midpoints of lines or the center points of circles, circle parts, ellipses and ellipse parts. If snapping and this snap modes are active, instead of entering a point a (part) object is identified whose center point is then worked out.

1. *Identify reference object*
   Any part object in an existing object can be identified as a reference object. This can be the side of a rectangle, the arc of an ellipse segment or a circular arc within a surface. A geometry line can not be identified as it has no "middle".
   The (part) object is identified by clicking with the mouse on its outline. If more than one (part) object lies at the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the 'right' object can be chosen.

After selecting the (part) object, its center or mid point is calculated. This point is then used like a point entered by hand (see graphic).



The "Midpoint" snap mode is an exclusive mode. If it is active, then all other snap modes are deactivated.

 Optionally, this snap mode can be turned on or off by left-clicking the button in the panel shown to the left. In addition the button always shows the snap mode's current status.

Related commands:

- Snap Modes>Snap Active
- Snap Modes>Quadrant
- Snap Modes>Edge
- Snap Modes>Corner / End-Point
- Snap Modes>Intersection
- Snap Modes>Geometry
- Snap Modes>Marking

Snap Modes>Other Point
Snap Modes>Relative
Snap Modes>Snap Radius

Snap Modes>Quadrant (Extra Menu)

The snap modes determine which type of point should be used as a snapping reference. There are various snap modes, some of which can be used together.

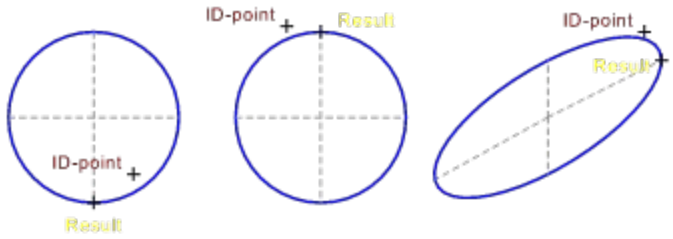
The snap modes take effect when snapping is active. Snapping can be turned on or off using the F6 Key or the first button in the Panel. It can be temporarily (de)activated during point entry using the SHIFT key.

This snap mode enables quadrant points to be used as snapping references. With circles and circle parts these are the points on the circle which lie at angles of 0°, 90°, 180° and 270° relative to the circle's center. With ellipses, they lie at the four ends of the positive and negative half axes. If snapping is on and this snap mode is active, instead of entering a point, an object is identified whose quadrant points are then worked out.

1. *Identify reference object*
   Any part object in an existing object can be identified as a reference object. This can be the arc of an ellipse or a circular arc within a circle. A line can not be chosen.
   The (part) object is identified by clicking with the mouse on its outline. If more than one (part) object lies at the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the 'right' object can be chosen.

   The position at which the (part) object was identified (the "ID point") affects the rest of the command. It determines which of the four possible quadrant points is selected. The one nearest to the ID point is always chosen (see below).

After choosing the (part) object all quadrant points are worked out and the one nearest to the ID point is selected. This point is then used like a point entered by hand (see graphic).



The "Quadrant" snap mode is an exclusive snap mode. If it is active, then all other snap modes are deactivated.

 Optionally, this snap mode can be turned on or off by left-clicking the button in the panel shown to the left. In addition the button always shows the snap mode's current status.


Related commands:

 Snap Modes>Snap Active

 Snap Modes>Center

 Snap Modes>Edge

Snap Modes>Edge (Extra Menu)

The snap modes determine which type of point should be used as a snapping reference. There are various snap modes, some of which can be used together.

The snap modes take effect when snapping is active. Snapping can be turned on or off using the F6 Key or the first button in the Panel. It can be temporarily (de)activated during point entry using the SHIFT key.

With this snap mode points can be placed exactly on the edge of an object. If snapping and this snap mode are active, instead of entering a point an object is identified on whose edge a point is placed.

1. *Identify reference object*
   Any part object in an existing object can be identified as a reference object. This can be the side of a rectangle, the arc of an ellipse segment or a circular arc within a surface.
   The (part) object is identified by clicking with the mouse on its outline. If more than one (part) object lies at the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the 'right' object can be chosen.

   The position at which the (part) object was identified (the "ID point") affects the rest of the command. It determines the position from which the perpendicular should be dropped onto the object (see below).

After choosing the (part) object, a perpendicular is dropped from the ID point to the selected object. A snapping reference point is inserted where the perpendicular touches the object (see graphic).

The "Edge" snap mode is an exclusive snap mode. If it is active, then all other snap modes are deactivated.

 Optionally, this snap mode can be turned on or off by left-clicking the button in the panel shown to the left. In addition the button always shows the snap mode's current status.

Related commands:

Snap Modes>Snap Active

Snap Modes>Center

Snap Modes>Quadrant

Snap Modes>Corner / End-Point

Snap Modes>Intersection

Snap Modes>Corner / End-Point (Extra Menu)

The snap modes determine which type of point should be used as a snapping reference. There are various snap modes, some of which can be used together.

The snap modes take effect when snapping is active. Snapping can be turned on or off using the F6 Key or the first button in the Panel. It can be temporarily (de)activated during point entry using the SHIFT key.

This snap mode can be used to insert snapping reference points at the corners and ends of objects. These are the end points of lines, curves, circular and elliptical arcs, dimension lines and hatching lines, as well as corner points within curves, surfaces, circular sectors, circular segments, elliptic sectors and elliptic segments. To snap to such a point, simply click near it.

The "Corner / End-Point" snap mode can be combined with the Intersection, Geometry, Marking and Other Point snap modes.

 Optionally, this snap mode can be turned on or off by left-clicking the button in the panel shown to the left. In addition the button always shows the snap mode's current status.

Related commands:

- Snap Modes>Snap Active
- Snap Modes>Center
- Snap Modes>Quadrant
- Snap Modes>Edge
- Snap Modes>Intersection
- Snap Modes>Geometry
- Snap Modes>Marking
- Snap Modes>Other Point
- Snap Modes>Relative
- Snap Modes>Snap Radius

Snap Modes>Intersection (Extra Menu)

The snap modes determine which type of point should be used as a snapping reference. There are various snap modes, some of which can be used together.

The snap modes take effect when snapping is active. Snapping can be turned on or off using the F6 Key or the first button in the Panel. It can be temporarily (de)activated during point entry using the SHIFT key.

This snap mode can be used to place snapping reference points at intersections within objects. If the "Geometry" snap mode is active at the same time, snapping reference points can also be placed at the intersections of objects with the geometry and (if present) reference objects. To place a snapping reference point on an intersection, simply click near it.

The "Intersection" snap mode can be combined with the Corner / End-Point, Geometry, Marking and Other Point snap modes.

 Optionally, this snap mode can be turned on or off by left-clicking the button in the panel shown to the left. In addition the button always shows the snap mode's current status.

Related commands:

- Snap Modes>Snap Active
- Snap Modes>Center
- Snap Modes>Quadrant
- Snap Modes>Edge
- Snap Modes>Corner / End-Point
- Snap Modes>Geometry
- Snap Modes>Marking
- Snap Modes>Other Point
- Snap Modes>Relative
- Snap Modes>Snap Radius

Snap Modes>Geometry (Extra Menu)

The snap modes determine which type of point should be used as a snapping reference. There are various snap modes, some of which can be used together.

The snap modes take effect when snapping is active. Snapping can be turned on or off using the F6 Key or the first button in the Panel. It can be temporarily (de)activated during point entry using the SHIFT key.

This snap mode can be used to insert snapping reference points at intersections of geometry objects and intersections of reference objects with geometry objects. If "Intersection" snap mode is also active, snapping reference points can also be inserted at the intersection of objects with the geometry. To insert a snapping reference point at an intersection, simply click near it.

The "Geometry" snap mode can be combined with the Corner / End-Point, Intersection, Marking and Other Point snap modes.

 Optionally, this snap mode can be turned on or off by left-clicking the button in the panel shown to the left. In addition the button always shows the snap mode's current status.

Related commands:

- Snap Modes>Snap Active
- Snap Modes>Center
- Snap Modes>Quadrant
- Snap Modes>Edge
- Snap Modes>Corner / End-Point
- Snap Modes>Intersection
- Snap Modes>Marking
- Snap Modes>Other Point
- Snap Modes>Relative
- Snap Modes>Snap Radius

Snap Modes>Marking (Extra Menu)

The snap modes determine which type of point should be used as a snapping reference. There are various snap modes, some of which can be used together.

The snap modes take effect when snapping is active. Snapping can be turned on or off using the F6 Key or the first button in the Panel. It can be temporarily (de)activated during point entry using the SHIFT key.

Snapping reference points can be placed on top of markings using this command. To place a snapping reference on a marking, simply click near it.

The "Marking" snap mode can be combined with the Corner / End-Point, Intersection, Geometry and Other Point snap modes.

 Optionally, this snap mode can be turned on or off by left-clicking the button in the panel shown to the left. In addition the button always shows the snap mode's current status.

Related commands:

- Snap Modes>Snap Active
- Snap Modes>Center
- Snap Modes>Quadrant
- Snap Modes>Edge
- Snap Modes>Corner / End-Point
- Snap Modes>Intersection
- Snap Modes>Geometry
- Snap Modes>Other Point
- Snap Modes>Relative
- Snap Modes>Snap Radius

Snap Modes>Other Point (Extra Menu)

The snap modes determine which type of point should be used as a snapping reference. There are various snap modes, some of which can be used together.

The snap modes take effect when snapping is active. Snapping can be turned on or off using the F6 Key or the first button in the Panel. It can be temporarily (de)activated during point entry using the SHIFT key.

Snapping reference points can be placed at other points using this command. "Other Points" are all definition points which cannot have snapping reference points placed on them with other snap modes, for example the angel points of Bézier curves, end points of ellipse's half axes, position points of dimensions, the corners of multiple line text frames, etc. To place a snapping reference point on one of these points, simply click near it.

The "Other Point" snap mode can be combined with the Corner / End-Point, Intersection, Geometry and Marking snap modes.

 Optionally, this snap mode can be turned on or off by left-clicking the button in the panel shown to the left. In addition the button always shows the snap mode's current status.

Related commands:

- Snap Modes>Snap Active
- Snap Modes>Center
- Snap Modes>Quadrant
- Snap Modes>Edge
- Snap Modes>Corner / End-Point
- Snap Modes>Intersection
- Snap Modes>Geometry
- Snap Modes>Marking
- Snap Modes>Relative
- Snap Modes>Snap Radius

Snap Modes>Relative (Extra Menu)

The snap modes determine which type of point should be used as a snapping reference. There are various snap modes, some of which can be used together.

The snap modes take effect when snapping is active. Snapping can be turned on or off using the F6 Key or the first button in the Panel. It can be temporarily (de)activated during point entry using the SHIFT key.

Snapping reference points can be placed at a specific offset to a given point. After entering the reference point (either directly or using another snapping mode), entry fields appear in the status line, allowing to enter the offset.

The "Relative" snap mode can be combined with any other snapping mode(s), or it can be activated alone.

 Optionally, this snap mode can be turned on or off by left-clicking the button in the panel shown to the left. In addition the button always shows the snap mode's current status.

Related commands:

 Snap Modes>Snap Active

 Snap Modes>Center

 Snap Modes>Quadrant

 Snap Modes>Edge

 Snap Modes>Corner / End-Point

 Snap Modes>Intersection

 Snap Modes>Geometry

 Snap Modes>Marking

 Snap Modes>Other Point

 Snap Modes>Snap Radius

Snap Modes>Snap Radius (Extra Menu)

This command can be used to specify the parameters for placing snapping reference points and the identification of objects. This is done from a dialog in which the snap radius and the tolerance can be specified.

The snap radius is given in screen pixels and determines how far from a point or object the crosshair can be to be effective when placing a snapping reference point or identifying an object.
The snap radius can have values from 5 to 50. The current snap radius is displayed in the Panel's top button.

The tolerance specifies how precisely the distance should be calculated when placing a snapping reference point or identifying an object. It is given in screen pixels.
For example, if the tolerance is *two* pixels, all objects less than two pixels from the ID point are chosen. A small dialog then appears at the bottom of the screen with the help of which the 'right' object can be chosen.
The tolerance can have values between 1 and 10.

 Optionally, the right mouse button can be clicked on the button in the panel shown to the left, calling the dialog described above.


Related commands:

 Snap Modes>Snap Active

 Snap Modes>Center

 Snap Modes>Quadrant

 Snap Modes>Edge

 Snap Modes>Corner / End-Point

 Snap Modes>Intersection

 Snap Modes>Geometry

 Snap Modes>Marking

 Snap Modes>Other Point

Control Keys>Window Refresh (SPACEBAR) (Extra Menu)

This command can be used to force a redraw of the drawing window. The window over which the crosshair is located is redrawn. If the mouse is outside the drawing window, then the current window is redrawn.

If the SPACEBAR is held down, the zoom can be changed using the mouse. If the left mouse button is clicked in a drawing window the zoom factor is doubled in that window. If the right mouse button is clicked then the zoom factor in the window clicked in is halved.

In principle this is the same as using the commands Zoom>Double Zoom and Zoom>Halve Zoom.

Related command:

No related commands

Control Keys>Left Mouse Button (Extra Menu)

This command can be used to simulate the clicking of the left mouse button. The program reacts as if the left mouse button had been clicked and then immediately released at the current mouse position.

Related commands:

Control Keys>Middle Mouse Button

Control Keys>Right Mouse Button

Control Keys>Middle Mouse Button (Extra Menu)

This command can be used to simulate the clicking of the middle mouse button. The program reacts as if the middle mouse button had been clicked and then immediately released at the current mouse position. This also functions if the mouse in use does not have a middle button.

The middle mouse button usage can be specified using Settings>Mouse Button Usage.

Related commands:


Settings>Mouse Button


Control Keys>Left Mouse Button


Control Keys>Right Mouse Button

Control Keys>Right Mouse Button (Extra Menu)

This command can be used to simulate the clicking of the right mouse button. The program reacts as if the right mouse button had been clicked and then immediately released at the current mouse position.

The right mouse button can be configured using Settings>Mouse Button Usage.

Related commands:

Settings>Mouse Button

Control Keys>Left Mouse Button

Control Keys>Middle Mouse Button

Control Keys>Menu Button (ALT) (Extra Menu)

With this command the pressing of the Windows menu key (i.e. ALT) can be simulated. The program reacts as if the ALT key had been pressed and immediately released.

This activates the menu. If the menu bar is visible, the **File** menu drops down. If the menu bar is not visible the menu appears at the current mouse position with the **File** menu selected.

Related command:

No related commands

Control Keys>Step Left (LEFTARROW) (Extra Menu)

This command can be used to simulate pressing the cursor key with a left pointing arrow. The program reacts as if the LEFTARROW had been pressed and immediately released.

This moves the window by a specified step to the left, that is, the window content moves to the right. The size of the step by which the section is moved can be specified using the command Control Keys>Edit Step.

The window in which the crosshair is located is changed. If the mouse pointer is outside the drawing area, the current (active) window is changed.

If the drawing window's scroll bars are visible (this can be set using the command Settings>Screen), the corresponding scroll bar arrow can be clicked on. The specified step size will be used.

Related commands:

 Control Keys>Step Right (RIGHT ARROW)

 Control Keys>Step Up (UP ARROW)

 Control Keys>Step Down (DOWN ARROW)

 Zoom>Previous View

Control Keys>Step Right (RIGHTARROW) (Extra Menu)

This command can be used to simulate pressing the cursor key with the right pointing arrow. The program reacts as if the RIGHTARROW key had been pressed and immediately released.

This moves the window by a specified step to the right, that is, the window content moves to the left. The size of the step by which the section is moved can be specified using the command Control Keys>Edit Step.

The window in which the crosshair is located is changed. If the mouse pointer is outside the drawing area, the current (active) window is changed.

If the drawing window's scroll bars are visible (this can be set using the command Settings>Screen), the corresponding scroll bar arrow can be clicked on. The specified step size will be used.

Related commands:

 Control Keys>Step Left (LEFTARROW)

 Control Keys>Step Up (UP ARROW)

 Control Keys>Step Down (DOWN ARROW)

 Zoom>Previous View

Control Keys>Step Up (UP ARROW) (Extra Menu)

This command can be used to simulate pressing the cursor key with an upward pointing arrow. The program reacts as if the UP ARROW had been pressed and immediately released.

This moves the window up by a specified step, that is, the window content moves down. The size of the step by which the section is moved can be specified using the command Control Keys>Edit Step.

The window in which the crosshair is located is changed. If the mouse pointer is outside the drawing area, the current (active) window is changed.

If the drawing window's scroll bars are visible (this can be set using the command Settings>Screen), the corresponding scroll bar arrow can be clicked on. The specified step size will be used.

Related commands:


Control Keys>Step Left (LEFTARROW)


Control Keys>Step Right (RIGHT ARROW)


Control Keys>Step Down (DOWN ARROW)


Zoom>Previous View

Control Keys>Step Down (DOWN ARROW) (Extra Menu)

This command can be used to simulate pressing the cursor key with the downward pointing arrow. The program reacts as if the DOWNARROW key had been pressed and immediately released.

This moves the window down by a specified step, that is, the window content moves up. The size of the step by which the section is moved can be specified using the command Control Keys>Edit Step.

The window in which the crosshair is located is changed. If the mouse pointer is outside the drawing area, the current (active) window is changed.

If the drawing window's scroll bars are visible (this can be set using the command Settings>Screen), the corresponding scroll bar arrow can be clicked on. The specified step size will be used.

Related commands:


Control Keys>Step Left (LEFTARROW)


Control Keys>Step Right (RIGHT ARROW)


Control Keys>Step Up (DOWN ARROW)


Zoom>Previous View

Control Keys>Page Left (HOME) (Extra Menu)

This command simulates pressing the HOME key. The program reacts as if the HOME key had been pressed and immediately released.

The window on the drawing is moved by 90% to the left, that is, the window content moves to the right.

The window in which the crosshair is located is changed. If the mouse pointer is outside the drawing area, the current (active) window is changed.

If the drawing window's scroll bars are visible (this can be set using the command Settings>Screen), the corresponding scroll bar can be clicked on.

Related commands:


Control Keys>Page Right (END)


Control Keys>Page Up (PAGEUP)


Control Keys>Page Down (PAGEDOWN)


Zoom>Previous View

Control Keys>Page Right (END) (Extra Menu)

This command can be used to simulate pressing the END key. The program reacts as if the END key had been pressed and immediately released.

The window on the drawing is moved to the right by 90%, that is, the window content moves to the left.

The window in which the crosshair is located is changed. If the mouse pointer is outside the drawing area, the current (active) window is changed.

If the drawing window's scroll bars are visible (this can be set using the command Settings>Screen), the corresponding scroll bar can be clicked on.

Related commands:


Control Keys>Page Left (HOME)


Control Keys>Page Up (PAGEUP)


Control Keys>Page Down (PAGEDOWN)


Zoom>Previous View

Control Keys>Page Up (PAGEUP) (Extra Menu)

This command can be used to simulate the pressing of the PAGEUP key. The program reacts as if the PAGEUP had been pressed and immediately released.

The window on the drawing is moved up by 90%, that is the window content moves down.

The window in which the crosshair is located is changed. If the mouse pointer is outside the drawing area, the current (active) window is changed.

If the drawing window's scroll bars are visible (this can be set using the command Settings>Screen), the corresponding scroll bar can be clicked on.

Related commands:


Control Keys>Page Left (HOME)


Control Keys>Page Right (END)


Control Keys>Page Down (PAGEDOWN)


Zoom>Previous View

Control Keys>Page Down (PAGEDOWN) (Extra Menu)

This command can be used to simulate pressing the PAGEDOWN key. The program reacts as if the PAGEDOWN key had been pressed and immediately released.

This moves the window on the drawing down by 90%, that is, the window content moves to the up.

The window in which the crosshair is located is changed. If the mouse pointer is outside the drawing area, the current (active) window is changed.

If the drawing window's scroll bars are visible (this can be set using the command Settings>Screen), the corresponding scroll bar can be clicked on.

Related commands:


Control Keys>Page Left (HOME)


Control Keys>Page Right (END)


Control Keys>Page Up (PAGEUP)


Zoom>Previous View

Control Keys>Edit Step (Extra Menu)

This command can be used to alter the step size for moving the viewable part of the drawing with the help of the buttons in the panel or the cursor keys. This is done via a dialog in which the step size, in percent, can be entered. Values between 5 and 100 percent are allowed.

The step size orientates itself to the smaller of the drawing window's height and width.

Affected commands

Control Keys>Step Left (LEFT ARROW)

Control Keys>Step Right (RIGHT ARROW)

Control Keys>Step Up (UP ARROW)

Control Keys>Step Down (DOWN ARROW)

Help Menu

Contents of Reference
Help on Active Command
Help Cursor
How to Use Help
Registration
Questions & Answers
TommySoftware® Direct
Start Puzzle
About TommySoftware® CAD/DRAW

Contents of Reference (Help Menu)

This command calls the TommySoftware® CAD/DRAW Reference.

If you want specific information about a particular TommySoftware® CAD/DRAW command, there are various ways of finding it directly from the program.

1. Go to the reference's index. In the index you will find a list of menus. Click on the desired menu. A list of all commands in this menu appears. Click on the command which you want information about.

2. Open the relevant menu from the TommySoftware® CAD/DRAW menu bar. Use the cursor keys to move the highlight bar until it is over the command which you want information about. If there is a submenu, you can open it with the right arrow key, and close it again with the left arrow key. As soon as the highlight bar is over the command which you want information about, press the F1 key.

3. Activate the popup menu. Click on the menu icon for the desired menu, and if neccessary, on the submenu icon of the desired submenu. Move the mouse pointer over the icon of the command which you want information about. The command name should appear in the title bar of the popup menu. Then press the F1 key.

4. Display the tools window. Move the mouse pointer over the icon of the command which you want information about. Press the F1 key.

5. Choose the command which you want information about as usual (from the menus, the popup menu or with a keyboard shortcut). As soon as the command's icon appears in the status window, press F1. This displays the help topic of the current command.
   This works only with commands which remain active after use (e.g. **Draw>Line>Standard**), not with commands which are carried out immediately, (e.g. **File>Open Drawing**).

If you want information on a particular dialog box, simply press the F1 key while that dialog box is open.

Related commands:


Help on Active Command


Help Cursor


How to Use Help

Help on Active Command (Help Menu)

This command calls the help topic for the currently active command.

Related commands:

Contents of Reference

Help Cursor

How to Use Help

Help Cursor (Help Menu)

This command activates the help cursor. The help cursor shows a mouse pointer combined with a question mark.

By clicking with this cursor onto a screen element, the corresponding help topic will be displayed. The corresponding help topic will also be displayed if you choose a command.

To cancel the help cursor, press the ESC key.

Related commands:


Contents of Reference


Help on Active Command


How to Use Help

How to Use Help (Help Menu)

This command calls a description of the Windows Help System. This explains the use of the help system beginning with the basics.

This description is part of Windows and therefore varies according to your Windows version.

Related commands:

Contents of Reference

Help on Active Command

Help Cursor

Registration (Help Menu)

This command calls the registration dialog which allows you to choose from several options:

You can order the registration code via e-mail. For that you have to fill in a form (address, credit card information, etc.) which is copied to the clipboard afterwards. You then paste this order text (your credit card number is encrypted to prevent abuse) into your e-mail editor and send the e-mail to "sales@tommysoftware.com". The registration code is based on your user name and will also be sent to you via e-mail by TommySoftware®. After receiving your registration code you can unlock TommySoftware® CAD/DRAW.

Alternatively you can order TommySoftware® CAD/DRAW on CD via e-mail. And it is also possible to order the CD version, libraries, and add-ons by fax/letter. For that you have to print out and fill in a registration form.

If you purchased the CD version of TommySoftware® CAD/DRAW the registration will be carried out automatically by the installation program. However this command is always available.

Related command:

No related commands

Questions & Answers (Help Menu)

This command calls a text that contains some frequently asked questions concerning TommySoftware® CAD/DRAW and the answers.

Related command:

No related commands

TommySoftware® Direct (Help Menu)

This command calls "TommySoftware® Direct", the customer newspaper of TommySoftware®. It will inform you about TommySoftware® CAD/DRAW, libraries, add-ons, and other useful products of TommySoftware®.

Related command:

No related commands

Start Puzzle (Help Menu)

This command calls the TommySoftware® CAD/DRAW Puzzle. The Puzzle is a standalone program with the filename `PUZZLE.EXE` . It can also be started from the File Manager or the Program Manager.

The rules for the puzzle are well-known, but in case you are not familiar with them, here is a summary.


The aim is to get the fifteen pieces into the correct order again by moving them around. You move a piece by clicking on it. The pieces can only be moved into the empty space.

To make it easier, the fifteen pieces all have a small number. The puzzle is solved when the hole is at the top left and all the pieces are correctly sorted.

The Puzzle can *always* be solved. If you sometimes think that you are making no progress, then you are doing something wrong. It just takes a little patience, that's all.

Have fun!


Related command:


No related commands

About TommySoftware® CAD/DRAW (Help Menu)

This command calls the TommySoftware® CAD/DRAW information dialog. As well as the usual copyright information, this dialog contains various important information.

The first line of text shows the exact version number plus the creation data in the form year-month-day. The letter at the end of the version number shows the language version (›d‹ for German, ›e‹ for English, ›f‹ for French etc.).

Between the two horizontal lines the registration information is shown. If this version has been registered it displays the user name and registration code here. If not a short message is displayed.

Please do *never* state the registration code in any public mail (e.g. in a CompuServe forum)! Anyone knowing both your user name and the registration code can profit from this knowledge by illegally registering his copy to your name!

If you wish to register the application press the "Register" button. This will call the Registration command.

If you play around within this dialog (keep CTRL+SHIFT pressed and try double clicking with both mouse buttons) you may see something unexpected...

Related Command:

Program Status

Other Commands

These commands are *not* contained in the TommySoftware® CAD/DRAW menu system, but are called directly from different locations in the application.

Insert Block from Window
Define Block in Window

Open Drawing from List

Select Coordinate System
Edit Coordinate System
Select Hatching Type
Edit Hatching Type
Select Layer
Edit Layer
Select Pen
Edit Pen

Zoom Section
Zoom Overview
Zoom Page
Zoom Original Size
Zoom Factor
Zoom Previous View

Define Digitizer Command Field
Delete Digitizer Command Field

Insert Block from Window (Other Commands)

This command inserts a block assigned to a block list button into the drawing.

You can find further information on the way in which the block list works can be found in the Block List topic.

Related Command:

Define Block in Window

Define Block in Window (Other Commands)

This command is used to allocate a block to a block list button. Therefore, the dialog window Insert Block appears, in which the desired block can be chosen.

You can find further information on the way in which the block list works can be found in the Block List topic.

Related Command:

Insert Block from Window

Open Drawing from List (Other Commands)

This command opens one of the drawings whose name is in the drawing list.

For further information on this, refer to the section on the Drawing List.

Related commands:

Open Drawing

Drawing List

Select Coordinate System (Other Commands)

This command selects one of the currently defined coordinate systems.

Related commands:


Edit Coordinate System


Select Hatching Type


Edit Hatching Type


Select Layer


Edit Layer


Select Pen


Edit Pen

Edit Coordinate System (Other Commands)

This command allows to edit one of the currently defined coordinate systems.

Related commands:

 Select Coordinate System

 Select Hatching Type

 Edit Hatching Type

 Select Layer

 Edit Layer

 Select Pen

 Edit Pen

Select Hatching Type (Other Commands)

This command selects one of the currently defined hatching types.

Related commands:


Select Coordinate System


Edit Coordinate System


Edit Hatching Type


Select Layer


Edit Layer


Select Pen


Edit Pen

Edit Hatching Type (Other Commands)

This command allows to edit one of the currently defined hatching types.

Related commands:

 Select Coordinate System

 Edit Coordinate System

 Select Hatching Type

 Select Layer

 Edit Layer

 Select Pen

 Edit Pen

Select Layer (Other Commands)

This command selects one of the currently defined layers.

Related commands:

 Select Coordinate System

 Edit Coordinate System

 Select Hatching Type

 Edit Hatching Type

 Edit Layer

 Select Pen

 Edit Pen

Edit Layer (Other Commands)

This command allows to edit one of the currently defined layer.

Related commands:

 Select Coordinate System

 Edit Coordinate System

 Select Hatching Type

 Edit Hatching Type

 Select Layer

 Select Pen

 Edit Pen

Select Pen (Other Commands)

This command selects one of the currently defined pens.

Related commands:

 Select Coordinate System

 Edit Coordinate System

 Select Hatching Type

 Edit Hatching Type

 Select Layer

 Edit Layer

 Edit Pen

Edit Pen (Other Commands)

This command allows to edit one of the currently defined pens.

Related commands:

 Select Coordinate System

 Edit Coordinate System

 Select Hatching Type

 Edit Hatching Type

 Select Layer

 Edit Layer

 Select Pen

Zoom Section (Other Commands)

This command changes the area displayed in the related drawing window so that a given rectangular area is displayed at the maximum possible size.

For further information see Zoom>Section.

Related commands:

Zoom Overview

Zoom Page

Zoom Original Size

Zoom Factor

Zoom Previous View

Zoom Overview (Other Commands)

This command alters the area displayed in the current drawing window so that you can see all of the drawing's objects and instances.

For further information see Zoom>Overview.

Related commands:


Zoom Section


Zoom Page


Zoom Original Size


Zoom Factor


Zoom Previous View

Zoom Page (Other Commands)

You can use this command so that you can see the whole page as large as possible in the current drawing window.

For further information see Zoom>Page.

Related commands:


Zoom Section


Zoom Overview


Zoom Original Size


Zoom Factor


Zoom Previous View

Zoom Original Size (Other Commands)

This command changes the zoom factor used for the current drawing window so that all objects and instances appear at their original size.

For further information see Zoom>Original Size.

Related commands:


Zoom Section


Zoom Overview


Zoom Page


Zoom Factor


Zoom Previous View

Zoom Factor (Other Commands)

This command can be used to specify a zoom factor for the current drawing window. It calls a small dialog box, in which the factor to be used (between 0.00000000001 and 10000000000) is entered.

For further information see Zoom>Factor.

Related commands:


Zoom Section


Zoom Overview


Zoom Page


Zoom Original Size


Zoom Previous View

Zoom Previous View (Other Commands)

This command restores the previous view of the current drawing window. The last 20 changes to the view (changes to the zoom factor, scrolling, etc.) are saved separately for each visible window.

For further information see Zoom>Previous Zoom.

Related commands:


Zoom Section


Zoom Overview


Zoom Page


Zoom Original Size


Zoom Factor

Button and Tracking Options (Other Commands)

This command can be used to set up the key assignments for a connected digitizer's input device.

The Digitizer Button and Tracking Options dialog appears, in which the button key assignment of the digitizer and some values for the coordinate tracking can be edited.

If possible, an input device button which does not function as a mouse button should be defined as tracking button. In the other case, it would only be possible to enter coordinates in areas outside the area visible on screen. However, this may be a good idea if using a very large digitizer.

This command is only included in the TommySoftware® CAD/DRAW menu system if a digitizer with the correct driver is connected. For general tips on using digitizers, see Using Digitizers.

Related commands:

- Tracking Calibration
- Set Tracking Origin
- Load Assignment
- Save Assignment as
- Information
- Options
- Define Digitizer Command Field
- Delete Digitizer Command Field

Tracking Calibration (Other Commands)

This command can be used to calibrate a connected digitizer. This helps match the sizes of objects entered to the program.

1. *Enter start point*
   The start point for the reference distance must be entered with the *Digitize Input Device* using the button which is set for direct coordinate entry (by default, the second button of the input device).

2. *Enter end point*
   The end point for the reference distance must also be entered with the *Digitize Input Device* using the button which is set for direct coordinate entry (by default, the input device's second button).

After entering the points, there distance from each other (dependent on the scale in use) is calculated and a dialog showing this distance is displayed. Enter, instead of this, the distance which the points should be apart according to your template.

All the following, directly chosen coordinates are recalculated so that they fit in with the scale specified in the program.

This command is only included in the TommySoftware® CAD/DRAW menu system if a digitizer with the correct driver is connected. For general tips on using digitizers, see Using Digitizers.

Related commands:

- Button and Tracking Options
- Set Tracking Origin
- Load Assignment
- Save Assignment as
- Information
- Options
- Define Digitizer Command Field
- Delete Digitizer Command Field

Set Tracking Origin (Other Commands)

The origin for a connected digitizer can be set using this command. This helps fit the positions of objects entered with the digitizer with the program.

1. *Enter position*
   The origin must be entered with the *Digitize Input Device* using the button which is set for direct coordinate entry (by default, the input device's second button).

The position is then saved as the new origin.

This command is only included in the TommySoftware® CAD/DRAW menu system if a digitizer with the correct driver is connected. For general tips on using digitizers, see Using Digitizers.

Related commands:

- Button and Tracking Options
- Tracking Calibration
- Load Assignment
- Save Assignment as
- Information
- Options
- Define Digitizer Command Field
- Delete Digitizer Command Field

Load Assignment (Other Commands)

This command can be used to open an existing assignments file, i.e. load it into memory and activate it. This is done from a File Selection Dialog, from which the file to be used can be chosen.

This command is only included in the TommySoftware® CAD/DRAW menu system if a digitizer with the correct driver is connected. For general tips on using digitizers, see Using Digitizers.

Related commands:

- Button and Tracking Options
- Tracking Calibration
- Set Tracking Origin
- Save Assignment as
- Information
- Options
- Define Digitizer Command Field
- Delete Digitizer Command Field

Save Assignment as (Other Commands)

This command can be used to save the current digitizer settings including the calibration data and key assignments in an assignments file which can be loaded again at a later point.

A File Selection Dialog appears in which a name for the file is entered.

This command is only included in the TommySoftware® CAD/DRAW menu system if a digitizer with the correct driver is connected. For general tips on using digitizers, see Using Digitizers.

Related commands:

- Button and Tracking Options
- Tracking Calibration
- Set Tracking Origin
- Load Assignment
- Information
- Options
- Define Digitizer Command Field
- Delete Digitizer Command Field

Information (Other Commands)

This command display some information about the currently installed digitizer and its driver.

This command is only included in the TommySoftware® CAD/DRAW menu system if a digitizer with the correct driver is connected. For general tips on using digitizers, see Using Digitizers.

Related commands:

- Button and Tracking Options
- Tracking Calibration
- Set Tracking Origin
- Load Assignment
- Save Assignment as
- Options
- Define Digitizer Command Field
- Delete Digitizer Command Field

Options (Other Commands) 

This command displays a dialog in which you can edit settings of the current digitizer's driver. Content and function of that dialog depends on the driver used.

This command is only included in the TommySoftware® CAD/DRAW menu system if a digitizer with the correct driver is connected. For general tips on using digitizers, see Using Digitizers.

Related commands:

 Button and Tracking Options

 Tracking Calibration

 Set Tracking Origin

 Load Assignment

 Save Assignment as

 Information

 Define Digitizer Command Field

 Delete Digitizer Command Field

Define Digitizer Command Field (Other Commands)

A command field for the connected digitizer can be defined using this command. In the name of the command, the identifier for the command is given after the words "command field". It consists of three parts:

The first two numbers show the horizontal position of the command on the graphics tablet. This can be between 1 and 50 (inclusive).

The following letter shows the field's vertical position on the graphics tablet. It can be between 'A' and 'Z' and between 'a' and 'z' (inclusive).

The last number is the control layer number. It can be between 1 and 4 (inclusive).

1. *Choose command*
   The command which is to be assigned must be chosen. The command can be chosen by any valid method, i.e. from the menu bar, the Popup Menu, the toolbox or a shortcut key.

After choosing the command it is assigned to the relevant digitizer command field and is immediately available. Any existing command field assignment for this command field will be overwritten.

This command is not included in the TommySoftware® CAD/DRAW menu system. It is only available if a digitizer with the correct driver is connected. If these conditions are fulfilled, it can be called by clicking the button for the relevant command layer (by default the left or right "mouse button" on the input device) and at the same time holding down the SHIFT key). For general tips on using digitizers, see Using Digitizers.

Related commands:

- Button and Tracking Options
- Tracking Calibration
- Set Tracking Origin
- Load Assignment
- Save Assignment as
- Information
- Options
- Delete Digitizer Command Field

Delete Digitizer Command Field (Other Commands)

This command can be used to delete a command field of an attached digitizer. In the name of the command, the identifier for the command is given after the words "command field". It consists of three parts.:

The first two numbers show the horizontal position of the command on the graphics tablet. This can be between 1 and 50 (inclusive).

The following letter shows the field's vertical position on the graphics tablet. It can be between 'A' and 'Z' and between 'a' and 'z' (inclusive).

The last number is the control layer number. It can be between 1 and 4 (inclusive).

After clicking on the digitizer command field to be deleted, a dialog field appears which shows the current field assignment and allows the delete command to be canceled.

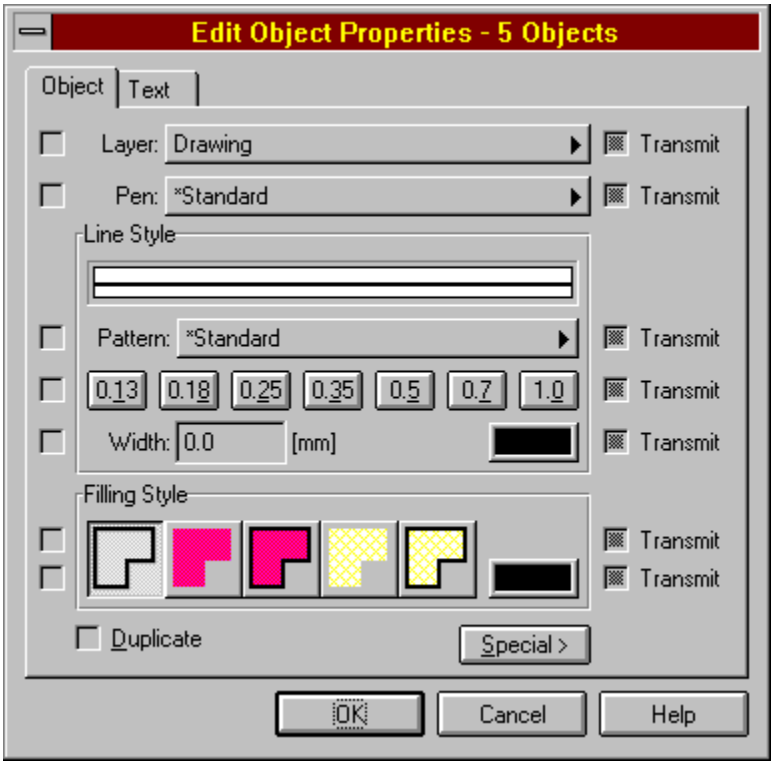
This command is not included in the TommySoftware® CAD/DRAW menu system. It is only available if a digitizer with the correct driver is connected. If these conditions are fulfilled, it can be called by clicking the button for the relevant command layer (by default the left or right "mouse button" on the input device) and at the same time holding down the SHIFT key). For general tips on using digitizers, see Using Digitizers.

Related commands:

- Button and Tracking Options
- Tracking Calibration
- Set Tracking Origin
- Load Assignment
- Save Assignment as
- Information
- Options
- Define Digitizer Command Field

"Edit Properties" Dialog

How can I access information on this dialog?



If several objects are selected the number of selected objects is displayed in the dialog caption.

The check boxes at the left edge of the dialog are only available when the properties of several objects are to be changed at the same time. In this case, the dialog will initially show the properties of one of the chosen objects. By activating one of the check boxes at the left edge the property belonging to that check box is copied to all selected objects.

The check boxes at the right edge of the dialog are used to change the transmission status. If such a check box is marked the transmission of the corresponding property is activated in all selected objects. By clearing a check box the transmission of the relevant property is deactivated in all selected objects. If the check box is grayed then no change is made.

If only one object is selected, the check boxes at the left edge are not visible and in addition the right check boxes can not be set to gray. The dialog's opening settings are then the settings for the chosen object which can now be changed directly. In this case the object type is displayed in the dialog caption. If the chosen object is not an instance, the check box on the right hand side will be titled "Fix" instead of "Transmit". The function in principle is the same.

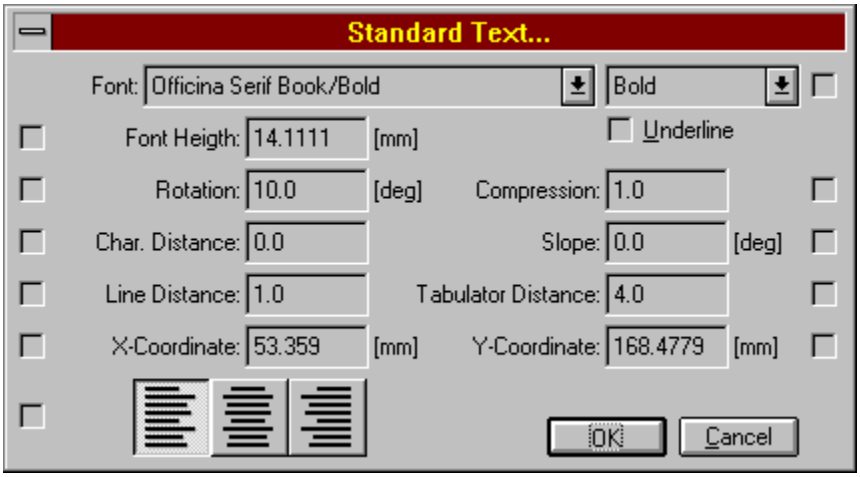
For further details please refer to Layers, Pens, Properties, and Transmission.

Associated command:

- Edit Properties

"Standard Text" Dialog

How can I access information on this dialog?



The check boxes at the left and right of the dialog are only shown when altering the properties of more than one object at once. In this case the properties shown in the dialog to start with are the properties of the first single line of text underneath the selected objects.

If only one object is selected, the check boxes are not visible. The properties shown in the dialog to start with are the properties of the selected object, which can then be altered directly.

Associated command:

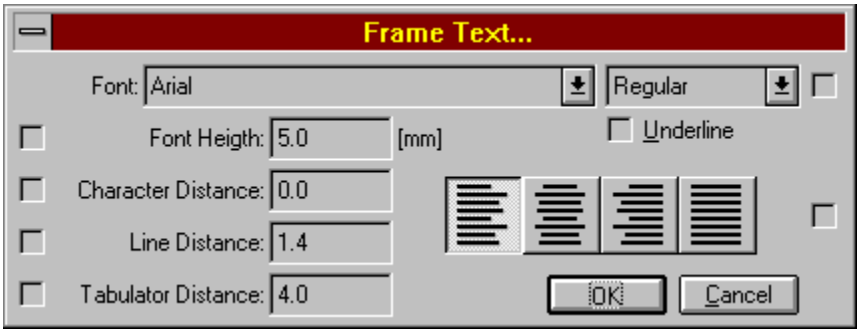
- Edit Properties

Possible preceding dialog:

- Edit Properties

"Frame Text" Dialog

How can I access information on this dialog?



The check boxes at the left and right of the dialog are only shown when altering the properties of more than one object at once. In this case the properties shown in the dialog to start with are the properties of the first multiple text lines underneath the selected objects.

If only one object is selected, the check boxes are not visible. The properties shown in the dialog to start with are the properties of the selected object, which can then be altered directly.

Associated command:

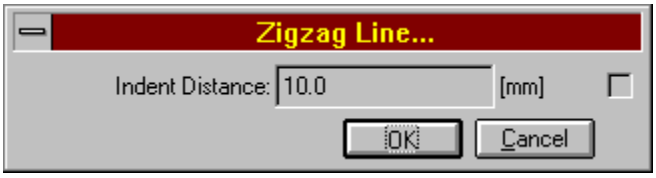
- Edit Properties

Possible preceding dialog:

- Edit Properties

"Zigzag Line" Dialog

How can I access information on this dialog?



The check boxes at the left and right of the dialog are only shown when altering the properties of more than one object at once. In this case the properties shown in the dialog to start with are the properties of the first zigzag line underneath the selected objects.

If only one object is selected, the check boxes are not visible. The properties shown in the dialog to start with are the properties of the selected object, which can then be altered directly.

Associated command:

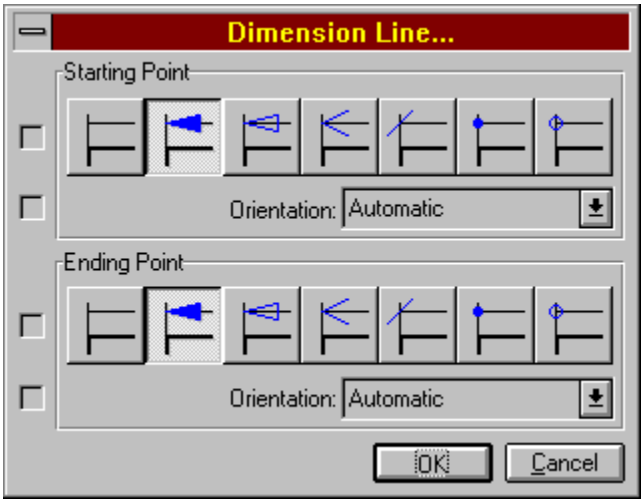
Edit Properties

Possible preceding dialog:

Edit Properties

"Dimension Line" Dialog

How can I access information on this dialog?



The check boxes at the left and right of the dialog are only shown when altering the properties of more than one object at once. In this case the properties shown in the dialog to start with are the properties of the first dimension line underneath the selected objects.

If only one object is selected, the check boxes are not visible. The properties shown in the dialog to start with are the properties of the selected object, which can then be altered directly.

Associated commands:

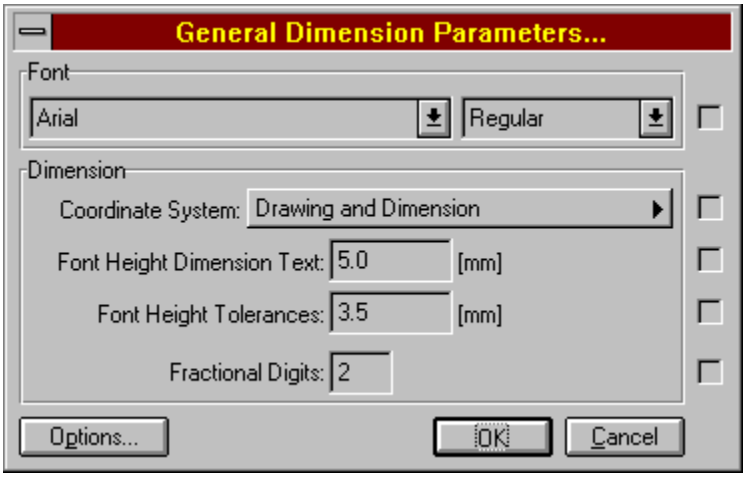
- Edit Properties
- Dimension Line>Straight
- Dimension Line>Curved

Possible preceding dialog:

- Edit Properties

"General Dimension Parameters" Dialog

How can I access information on this dialog?



The check boxes at the left and right of the dialog are only shown when altering the properties of more than one object at once. In this case the properties shown in the dialog to start with are the properties of the first dimension underneath the selected objects.

If only one object is selected, the check boxes are not visible. The properties shown in the dialog to start with are the properties of the selected object, which can then be altered directly.

If the dialog is used for setting a command's parameters, then the check boxes are not displayed. The properties shown in the dialog to start with are the last-set parameters for that command which can be altered directly.

The "Options" button is not displayed if this dialog is called while carrying out the Edit Properties command.

Associated commands:

- Edit Properties
- Dimension Line>Straight
- Dimension Line>Curved
- Dimension>Length, Points
- Dimension>Length, Objects
- Dimension>Length, Object - Point
- Dimension>Length, Object - Object
- Dimension>Radius, Points

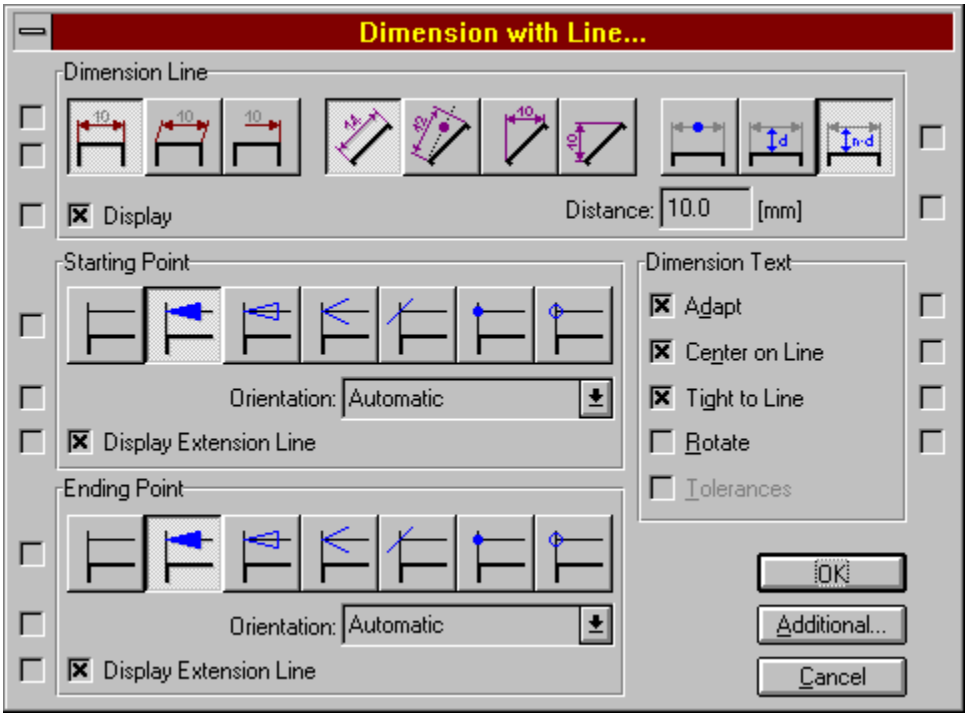
- Dimension>Radius, Object
- Dimension>Diameter, Points
- Dimension>Diameter, Object
- Dimension>Angle, Points
- Dimension>Angle, Line - Line
- Dimension>Angle, Circle Part
- Dimension>Arc Length, Points
- Dimension>Arc Length, Circle Part
- Dimension>Coordinates
- Calculate>Area
- Calculate>Perimeter / Length
- Dimension Parameters

Possible preceding dialogs:

- Edit Properties
- Dimension with Dimension Line
- Dimension without Dimension Line

"Dimension With Line" Dialog

How can I access information on this dialog?



The check boxes at the left and right of the dialog are only available if the properties of multiple objects are to be altered at the same time. In this case the dialog initially shows the settings of the first dimension with a dimension line under the chosen object.

If only one object is selected, the check boxes are not visible. The dialog then initially shows the properties of the selected object, which can then be altered *directly*.

If the dialog is used for setting a command's parameters, then the check boxes are not displayed. The properties shown in the dialog to start with are the last-set parameters for that command which can be altered directly. The "Tolerances" check box is only available in this case.

Associated commands:

- Edit Properties
- Dimension>Length, Points
- Dimension>Length, Objects
- Dimension>Length, Object - Point
- Dimension>Length, Object - Object
- Dimension>Radius, Points


Dimension>Radius, Object


Dimension>Diameter, Points


Dimension>Diameter, Object


Dimension>Angle, Points


Dimension>Angle, Line - Line


Dimension>Angle, Circle Part


Dimension>Arc Length, Points

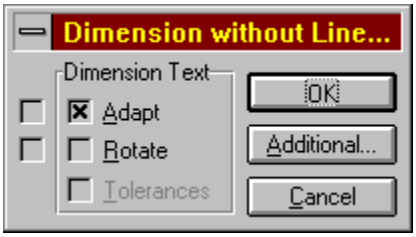

Dimension>Arc Length, Circle Part

Possible preceding dialog:


Edit Properties

"Dimension Without Line" Dialog

How can I access information on this dialog?



The check boxes at the left and right of the dialog are only available if the properties of multiple objects are to be altered at the same time. In this case the dialog initially shows the settings of the first dimension without a dimension line under the chosen object.

If only one object is selected, the check boxes are not visible. The dialog then initially shows the properties of the selected object, which can then be altered *directly*.

If the dialog is used for setting a command's parameters, then the check boxes are not displayed. The properties shown in the dialog to start with are the last-set parameters for that command which can be altered directly. The "Tolerances" check box is only available in this case.

Associated commands:

Edit Properties

Dimension>Coordinates

Calculate>Area

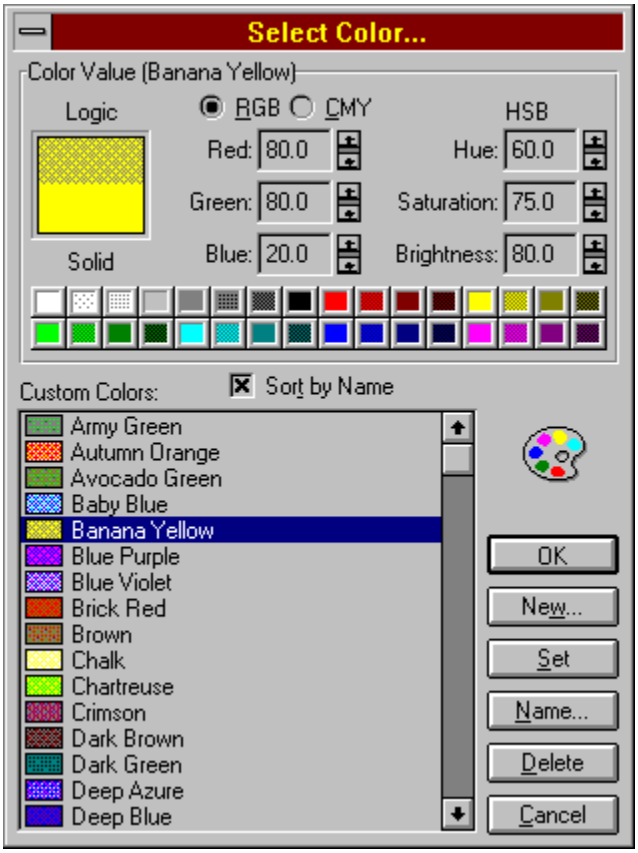
Calculate>Perimeter / Length

Possible preceding dialog:

Edit Properties

"Color Selection" Dialog

How can I access information on this dialog?



The colors defined in this dialog are not saved in drawing files, but in settings files; that is independently from the drawing (see Settings>Save As).

Associated commands:

No command-specific usage

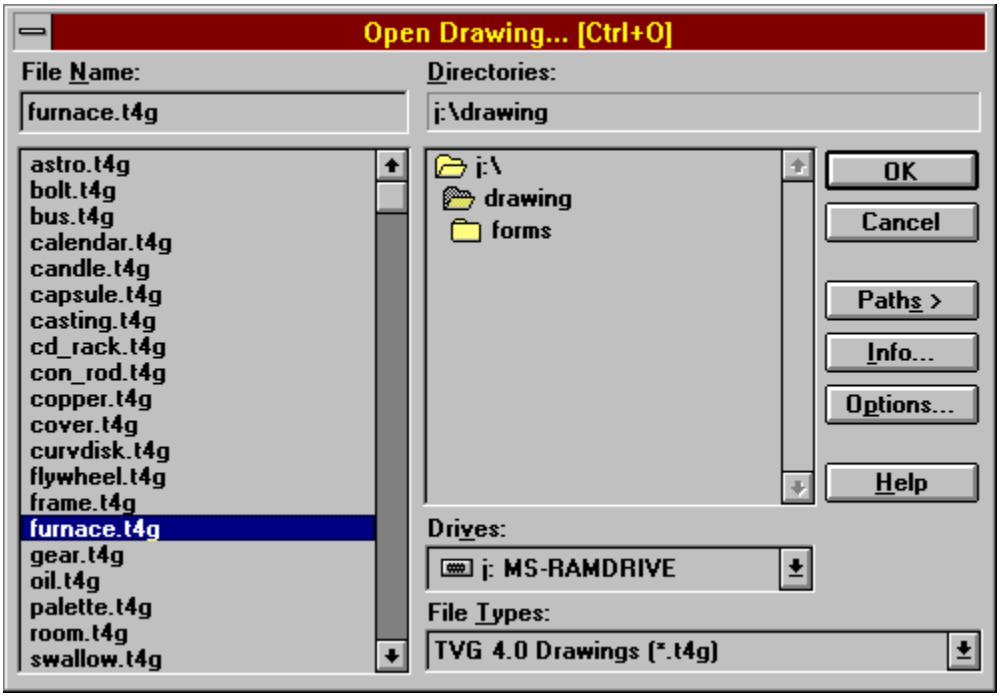
Possible preceding dialogs:

Edit Properties

Properties

"File Selection" Dialog

How can I access information on this dialog?

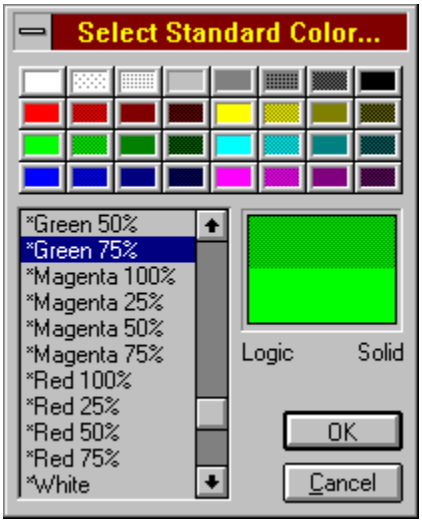


Associated command:

No command-specific usage

"Standard Color Selection" Dialog

How can I access information on this dialog?



Associated command:

No command-specific usage

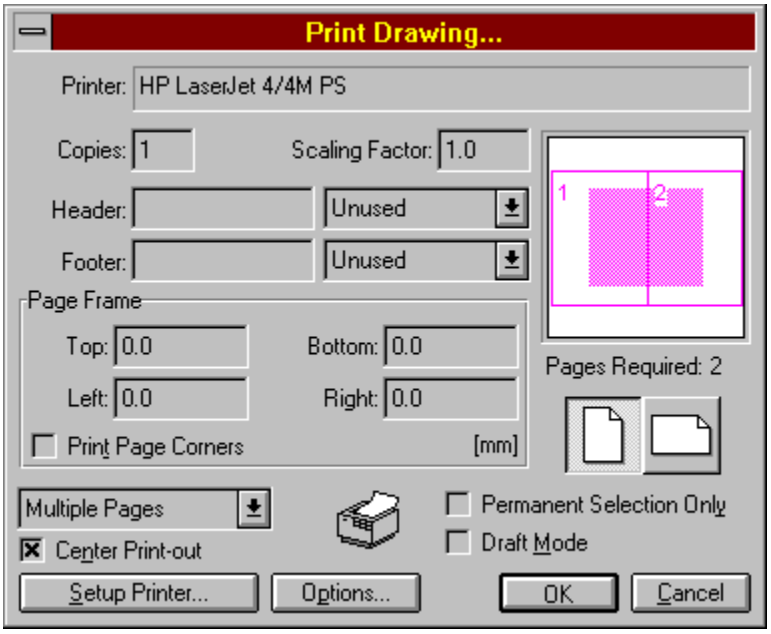
Possible preceding dialogs:

Comment Parameters

Color Settings

"Print Drawing" Dialog

How can I access information on this dialog?



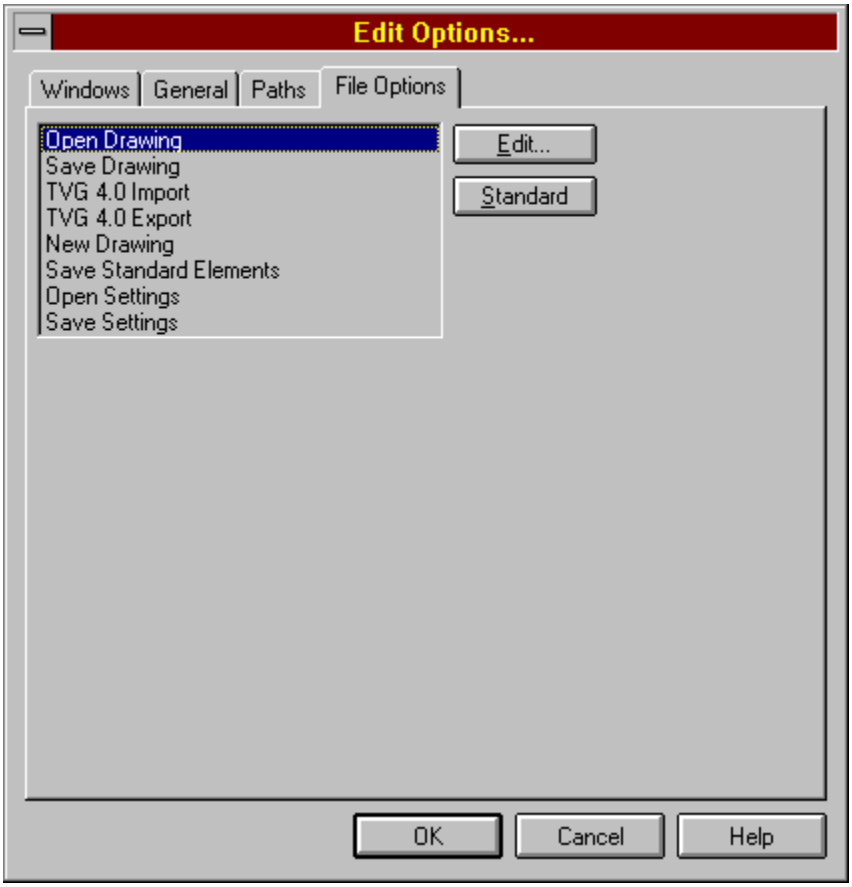
Associated commands:

Print Drawing

Print Section

"File Options" Dialog

How can I access information on this dialog?



Associated commands:

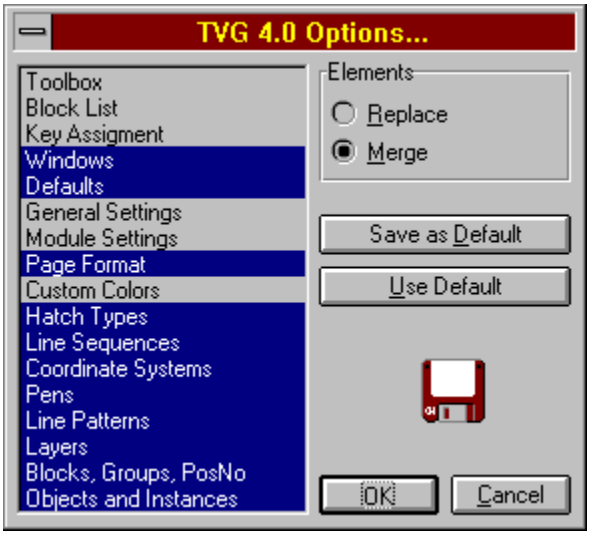
- New Drawing
- Open Drawing
- Save Drawing
- Save Drawing as
- Import>TVG 4.0 (*.T4G)
- Export>TVG 4.0 (*.T4G)
- Save Elements as Default
- Settings>Open
- Settings>Save As

Save Settings as Default

Save Settings on Exit

"Choose T4G Elements" Dialog

How can I access information on this dialog?



Associated command:

No command-specific usage

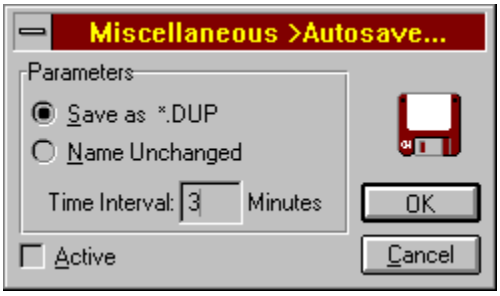
Possible preceding dialogs:

File Selection

File Options

"Autosave" Dialog

How can I access information on this dialog?

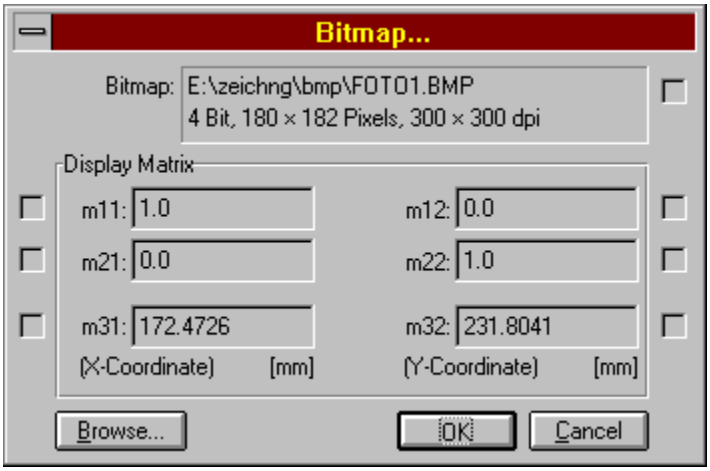


Associated command:

Miscellaneous>Autosave

"Bitmap" Dialog

How can I access information on this dialog?



The check boxes at the left and right of the dialog are only available if the properties of several objects are changed at the same time. In this case the initial setup of this window is the same as the setup of the first instance among the selected objects.

If only one object is selected, the check boxes are not visible. If only one object is chosen, this check box is not visible. In this case the initial setup of this window is the same as the setup of the selected object.

Associated command:

Edit Properties

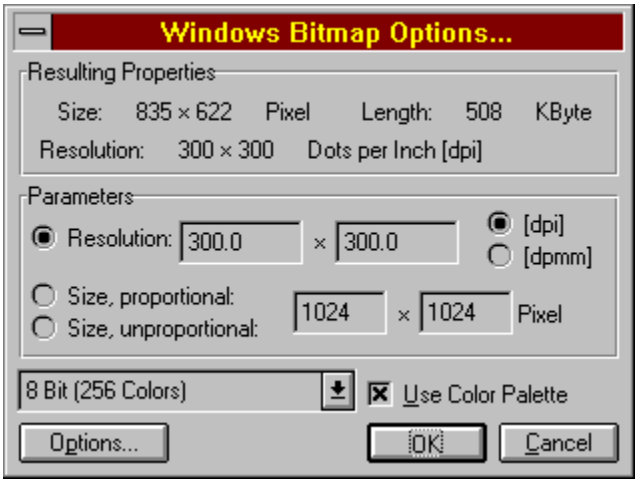
Import>Windows Bitmap (*.BMP)

Possible preceding dialogs:

Edit Properties

"Export Bitmap" Dialog

How can I access information on this dialog?



Associated commands:

Export>Windows Bitmap (*.BMP)

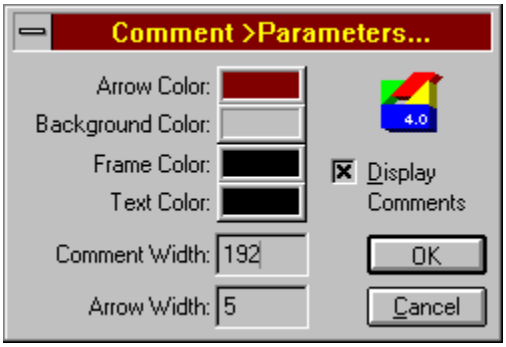
Export>Windows Bitmap, Section (*.BMP)

Possible preceding dialogs:

File Selection

"Comment Parameters" Dialog

How can I access information on this dialog?

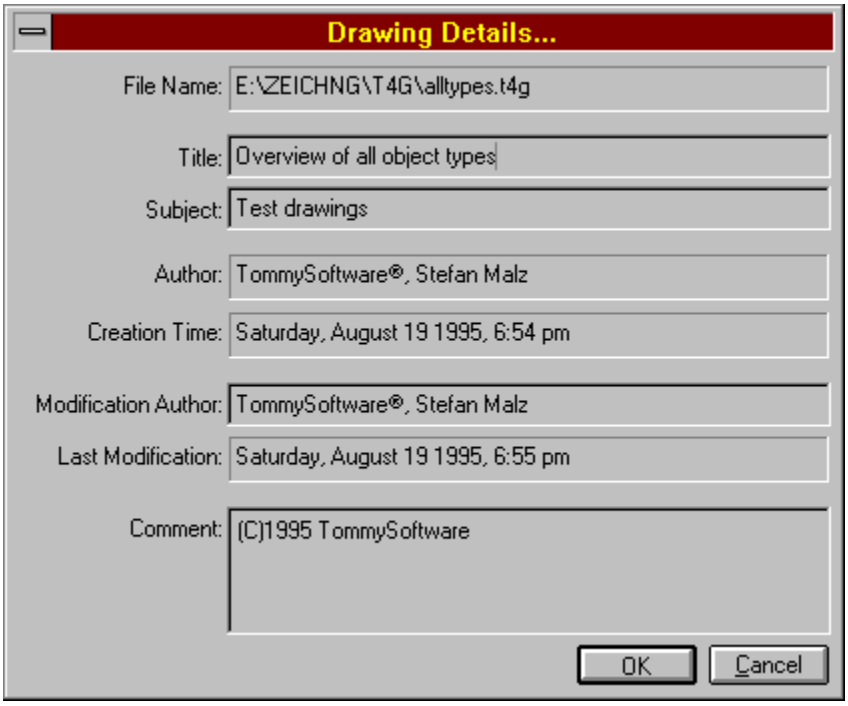


Associated command:

Comment Parameters

"Edit Details" Dialog

How can I access information on this dialog?



Associated commands:

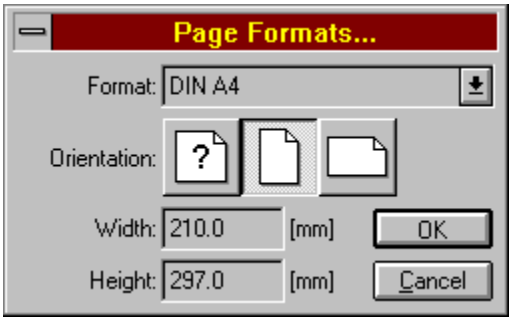
- Drawing Details
- Settings>Details
- Libraries

Possible preceding dialog:

- Libraries

"Page Format" Dialog

How can I access information on this dialog?

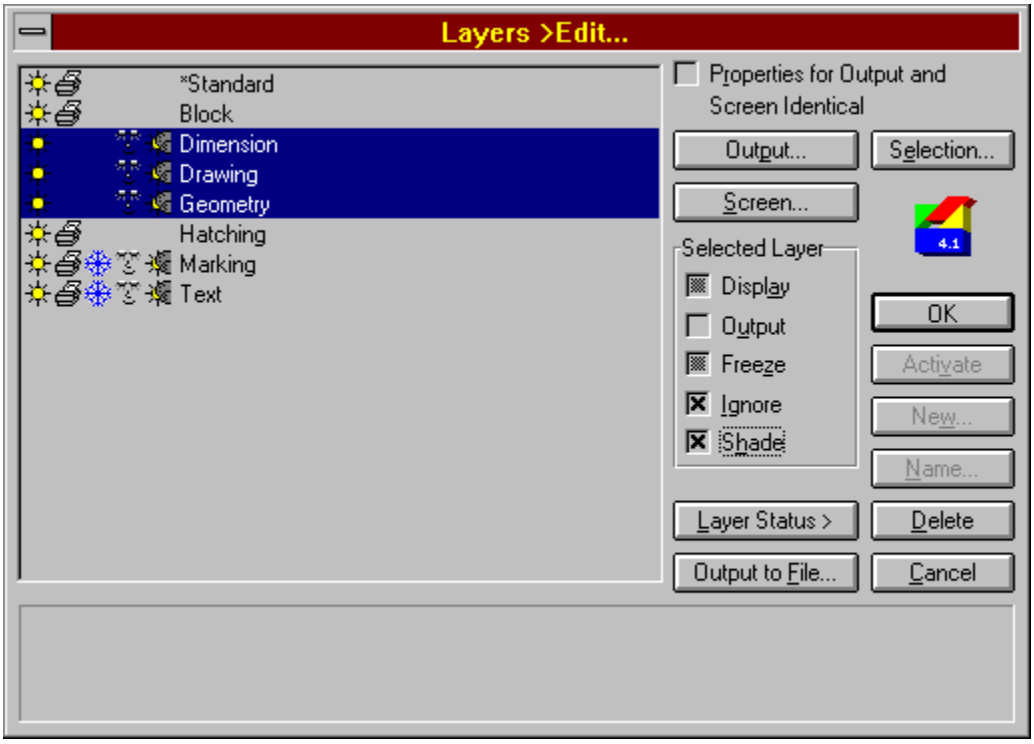


Associated command:

Page Formats

"Define Layers" Dialog

How can I access information on this dialog?

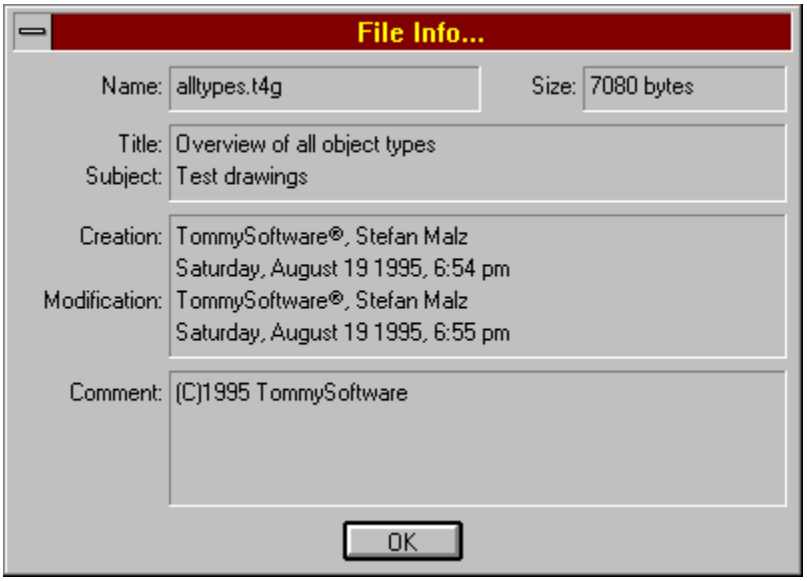


Associated command:

Layers>Edit

"View Details" Dialog

How can I access information on this dialog?



Associated command:

Fonts

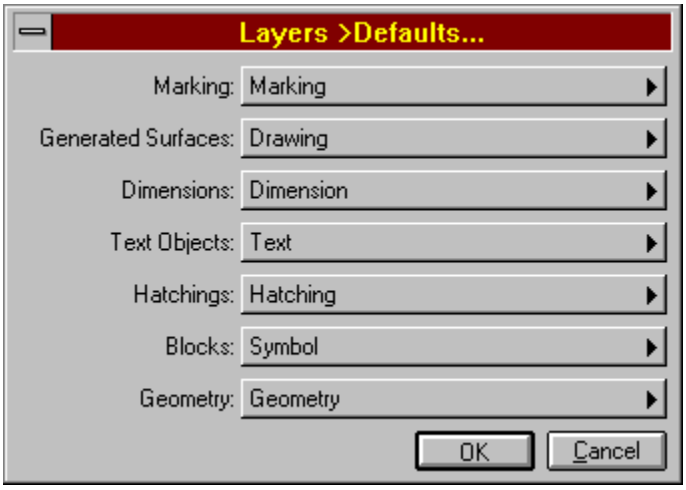
Possible preceding dialogs:

File Selection

Fonts

"Layer Defaults" Dialog

How can I access information on this dialog?

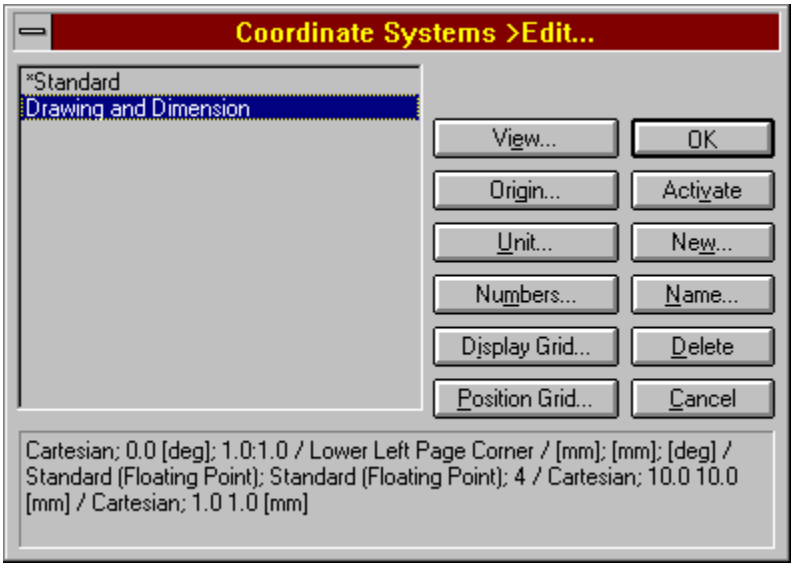


Associated command:

Layers>Defaults

"Define Coordinate Systems" Dialog

How can I access information on this dialog?

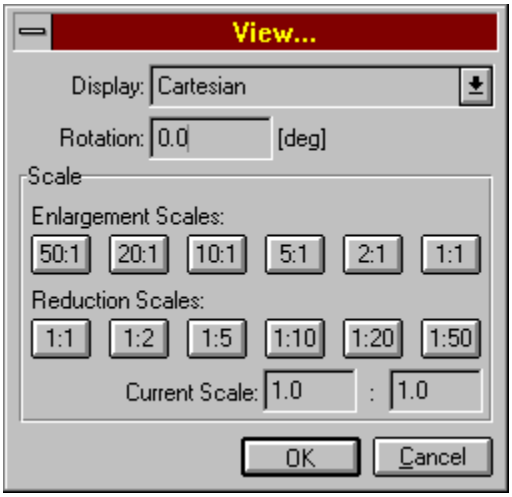


Associated command:

Coordinate Systems>Edit

"View Coordinate Systems" Dialog

How can I access information on this dialog?



Associated command:

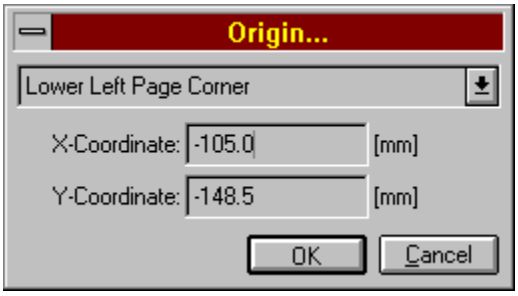
Coordinate Systems>Edit

Possible preceding dialog:

Define Coordinate Systems

"Coordinate System Origin" Dialog

How can I access information on this dialog?



Associated command:

Coordinate Systems>Edit

Possible preceding dialog:

Define Coordinate Systems

"Coordinate System Units" Dialog

How can I access information on this dialog?

Associated command:

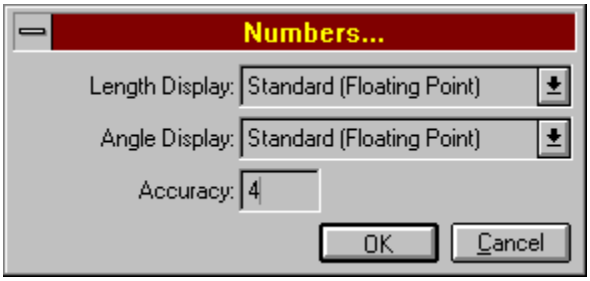
Coordinate Systems>Edit

Possible preceding dialog:

Define Coordinate Systems

"Coordinate System Numbers" Dialog

How can I access information on this dialog?



Associated command:

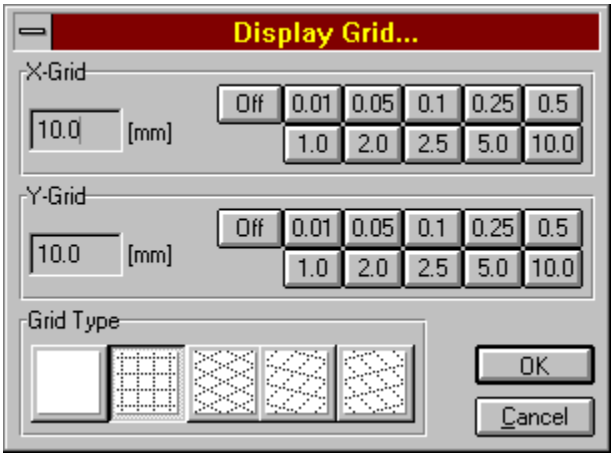
Coordinate Systems>Edit

Possible preceding dialog:

Define Coordinate Systems

"Coordinate System Display Grid" Dialog

How can I access information on this dialog?



Associated commands:

Coordinate Systems>Edit

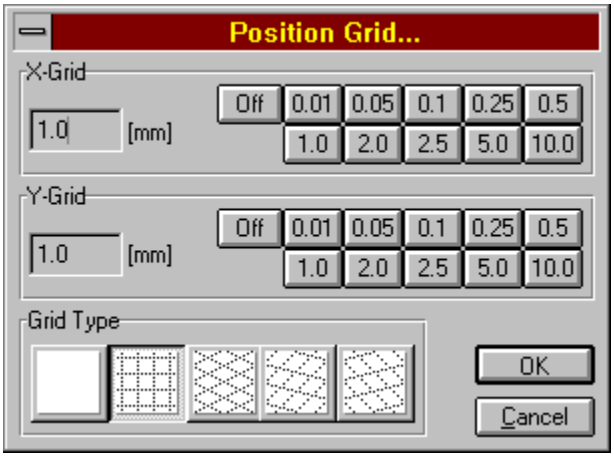
Coordinate Systems>Edit Display Grid

Possible preceding dialog:

Define Coordinate Systems

"Coordinate Systems Position Grid" Dialog

How can I access information on this dialog?



Associated command:

Coordinate Systems>Edit

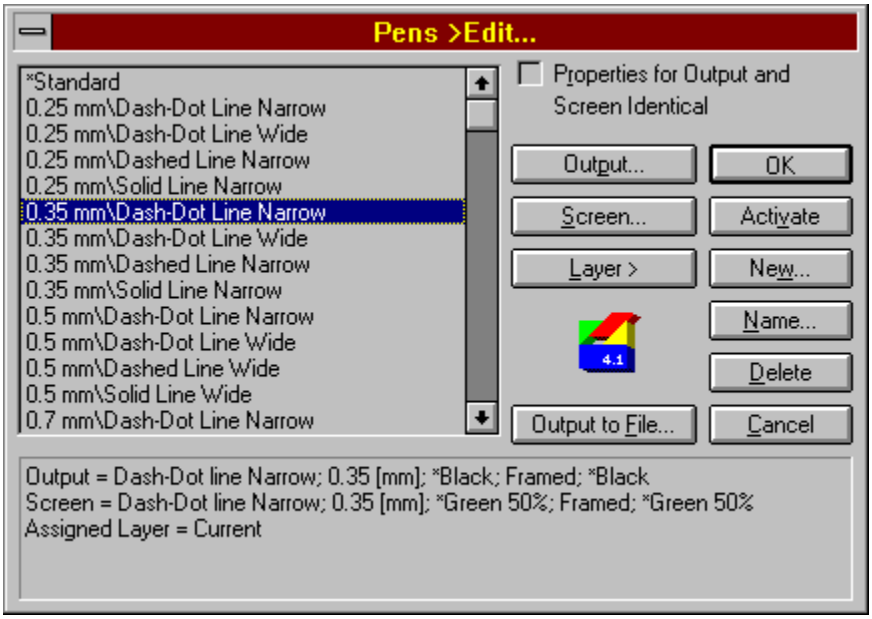
Coordinate Systems>Edit Position Grid

Possible preceding dialog:

Define Coordinate Systems

"Define Pens" Dialog

How can I access information on this dialog?

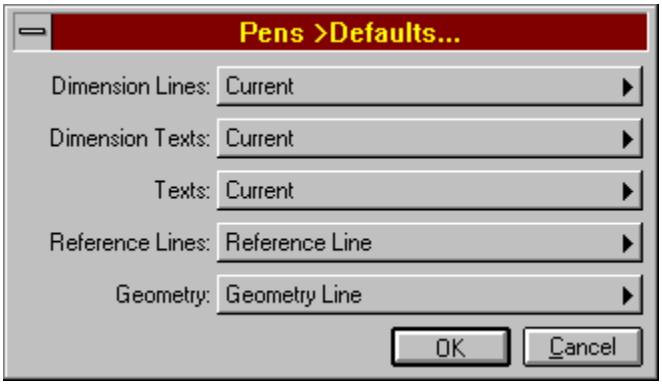


Associated command:

Pens>Edit

"Pen Defaults" Dialog

How can I access information on this dialog?

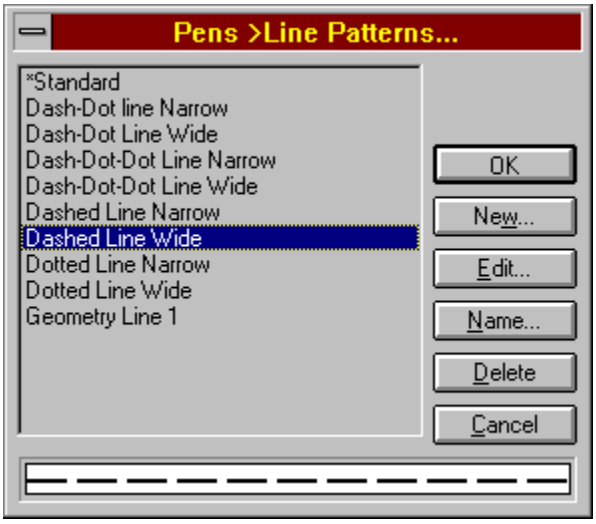


Associated command:

Pens>Defaults

"Define Line Pattern" Dialog

How can I access information on this dialog?

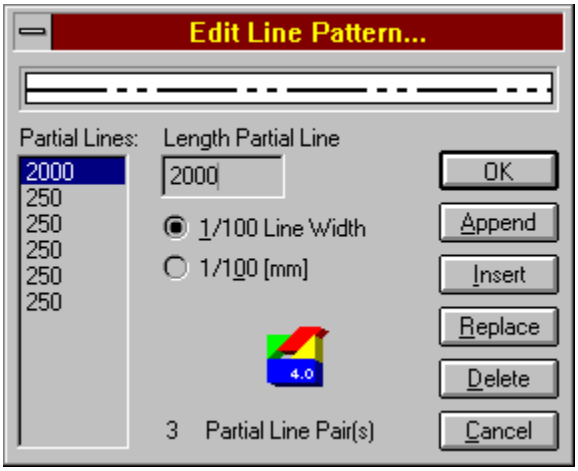


Associated command:

Pens>Line Pattern

"Edit Line Pattern" Dialog

How can I access information on this dialog?



Associated command:

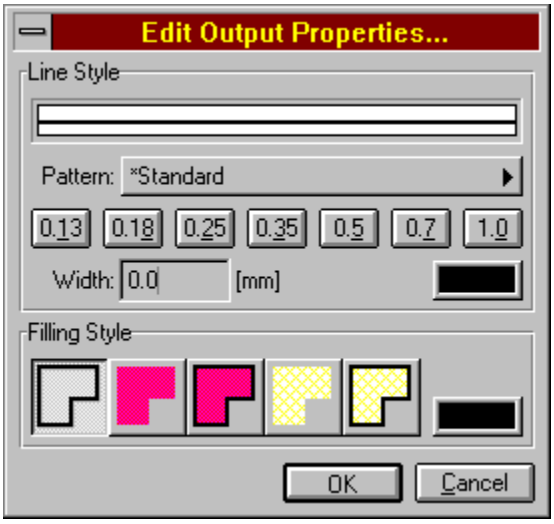
Pens>Line Pattern

Possible preceding dialog:

Edit Line pattern

"Properties" Dialog

How can I access information on this dialog?

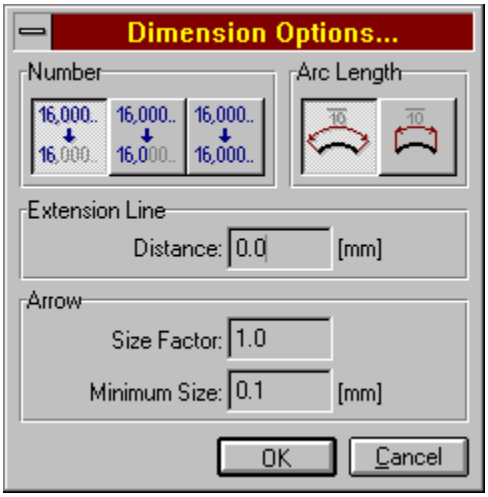


Associated command:

No command-specific usage

"Dimension Parameter Options" Dialog

How can I access information on this dialog?



Associated commands:

- Dimension Line>Straight
- Dimension Line>Curved
- Dimension>Length, Points
- Dimension>Length, Objects
- Dimension>Length, Object - Point
- Dimension>Length, Object - Object
- Dimension>Radius, Points
- Dimension>Radius, Object
- Dimension>Diameter, Points
- Dimension>Diameter, Object
- Dimension>Angle, Points
- Dimension>Angle, Line - Line
- Dimension>Angle, Circle Part
- Dimension>Arc Length, Points
- Dimension>Arc Length, Circle Part


Dimension>Coordinates


Calculate>Area


Calculate>Perimeter / Length

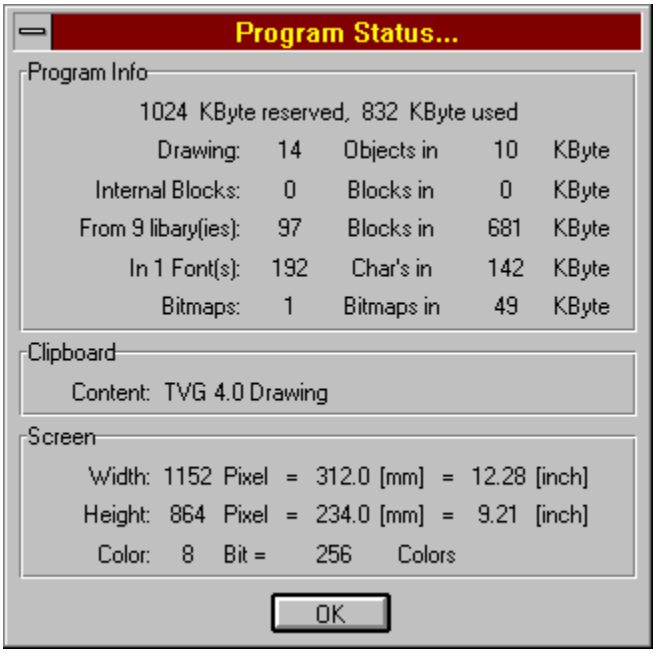

Dimension Parameters

Possible preceding dialog:


General Dimension Parameters

"Program Status" Dialog

How can I access information on this dialog?

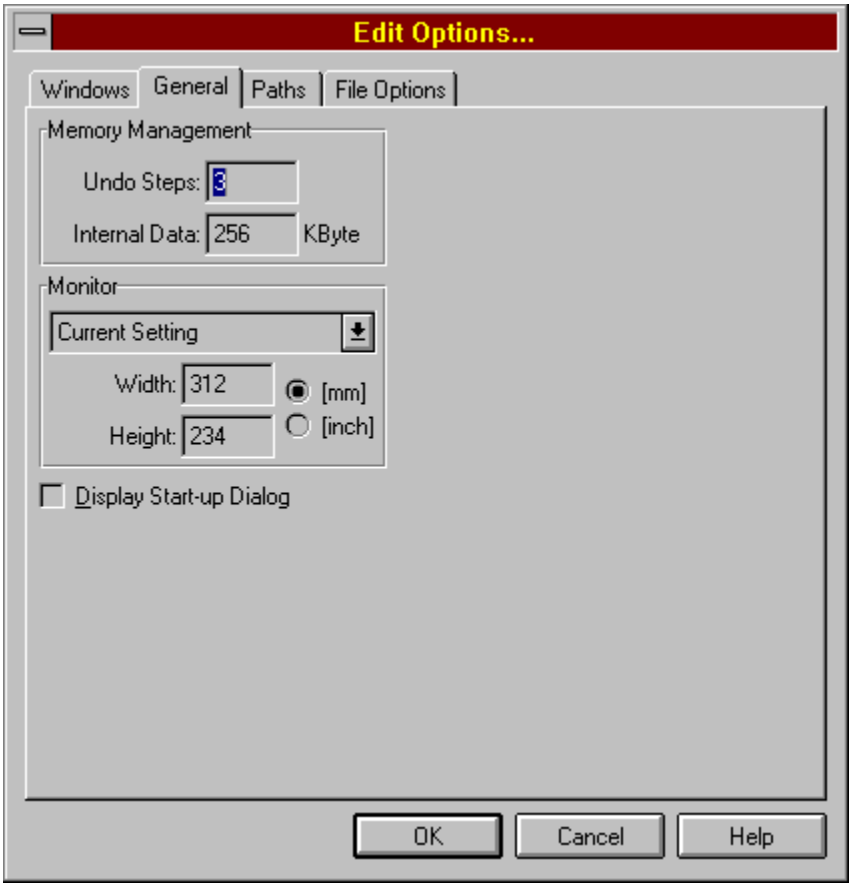


Associated command:

Program Status

"Memory and Monitor" Dialog

How can I access information on this dialog?

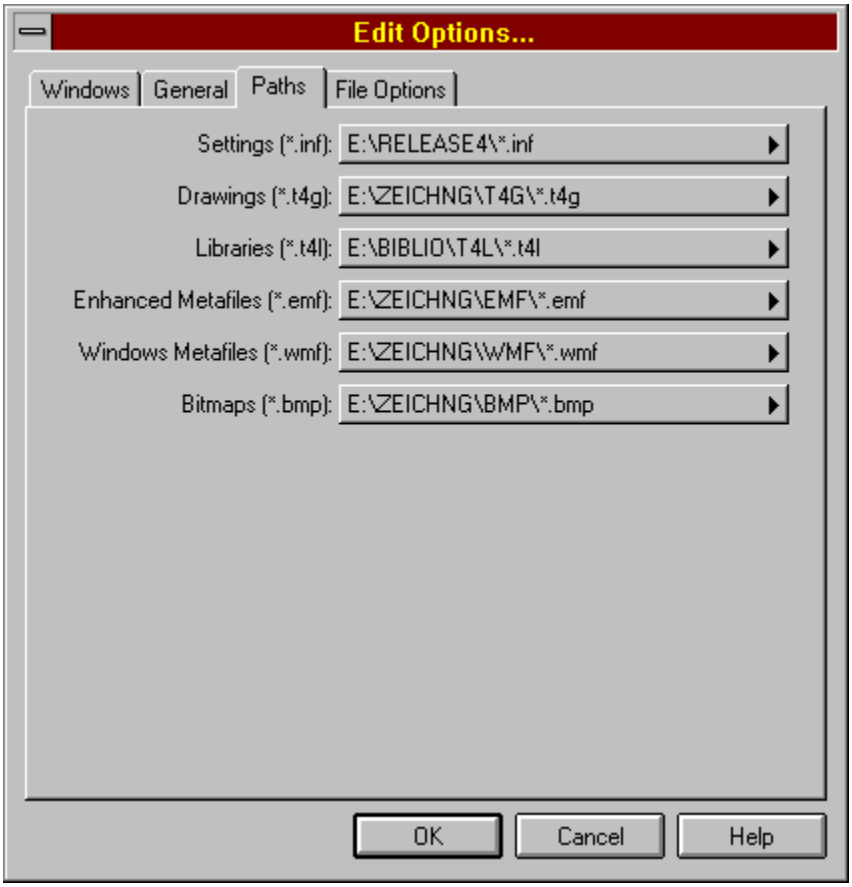


Associated command:

Options>General

"Standard Paths" Dialog

How can I access information on this dialog?

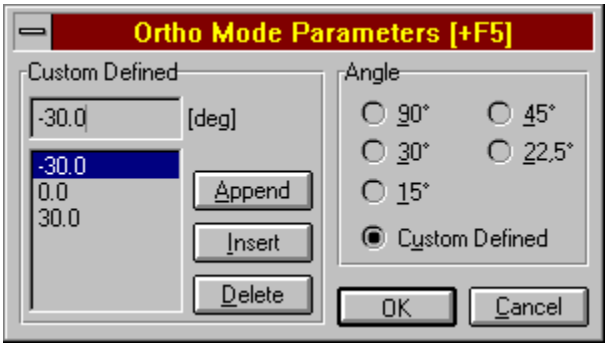


Associated command:

Options>Paths

"Orthogonal Mode Parameters" Dialog

How can I access information on this dialog?

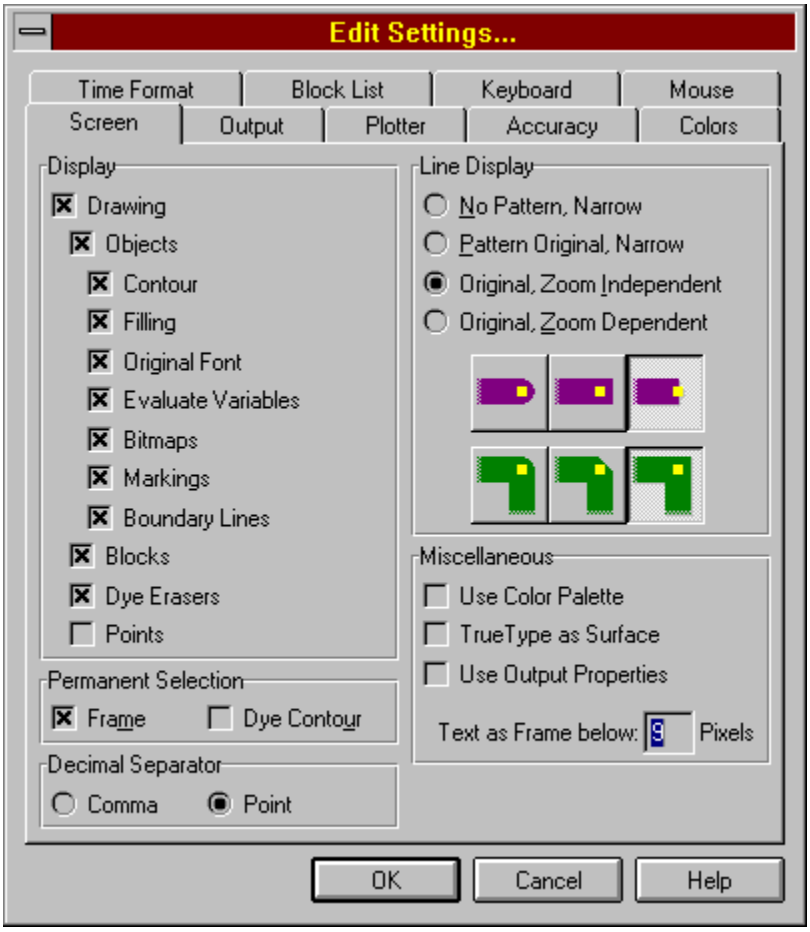


Associated command:

No command-specific usage

"Screen Settings" Dialog

How can I access information on this dialog?

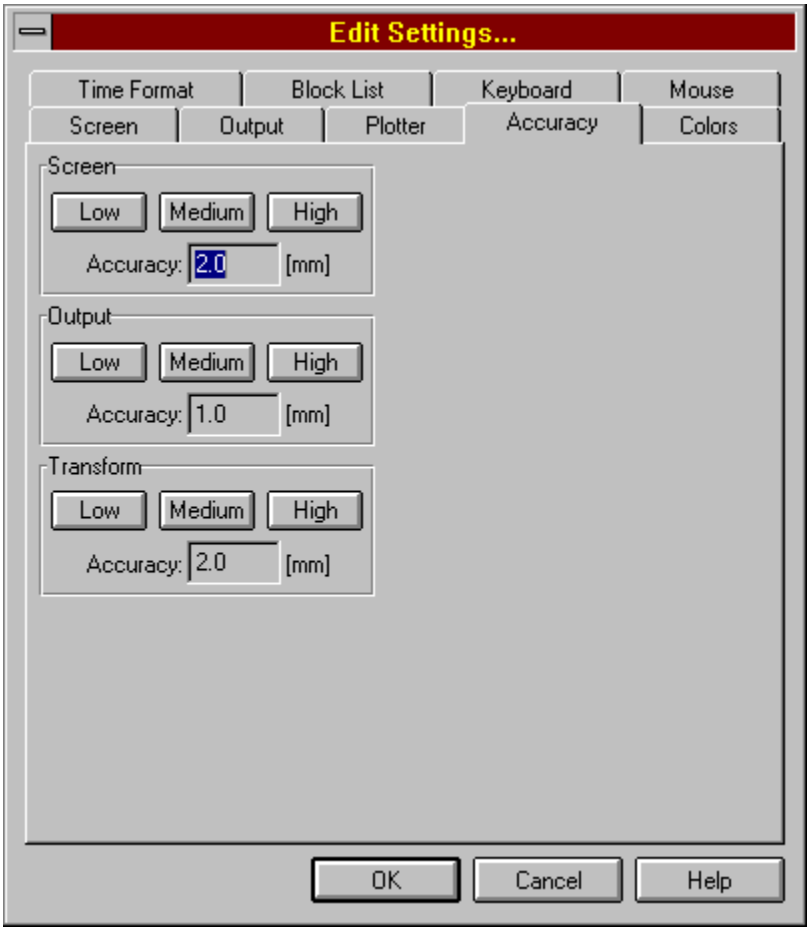


Associated command:

Settings>Screen

"Accuracy Settings" Dialog

How can I access information on this dialog?

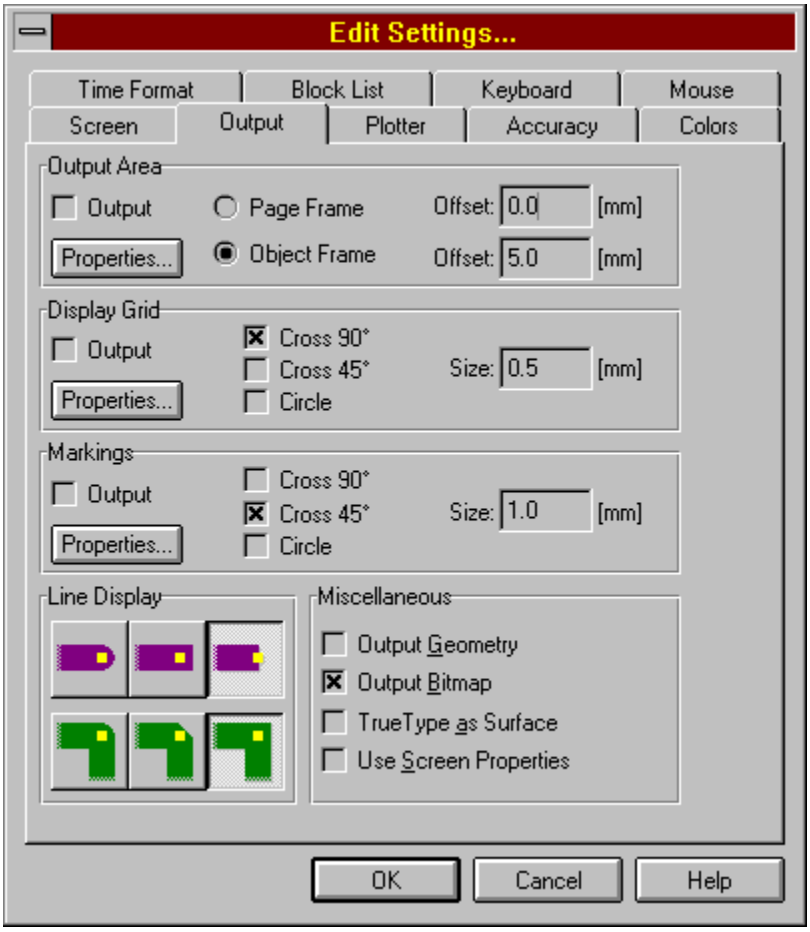


Associated commands:

- Settings>Accuracy
- Print Drawing
- Print Section
- Export>Enhanced Metafile (*.EMF)
- Export>Windows Metafile (*.WMF)
- Export>Windows Bitmap (*.BMP)
- Export>Windows Bitmap, Section (*.BMP)

"Output Settings" Dialog

How can I access information on this dialog?



Associated commands:

- Settings>Output
- Save Drawing
- Save Drawing as
- Export>TVG 4.0 (*.T4G)
- Export>Enhanced Metafile (*.EMF)
- Export>Windows Metafile (*.WMF)
- Export>Windows Bitmap (*.BMP)
- Export>Windows Bitmap, Section (*.BMP)


Print Drawing


Print Section


Cut


Copy

Possible preceding dialogs:

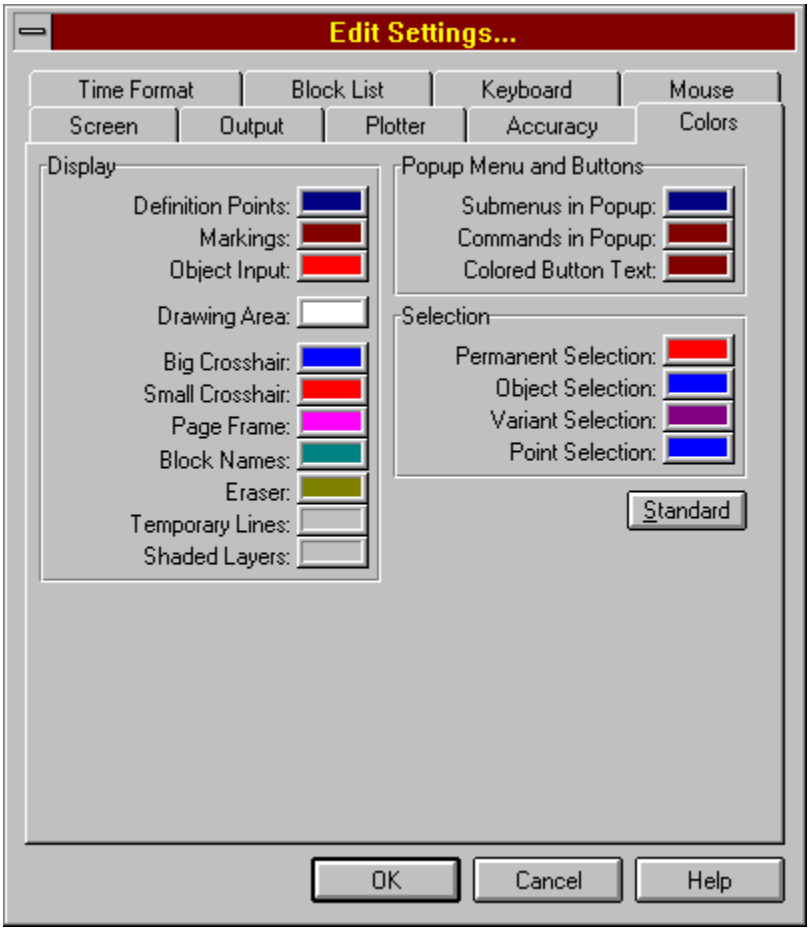

File Selection


Print Drawing


Export Bitmap

"Color Settings" Dialog

How can I access information on this dialog?

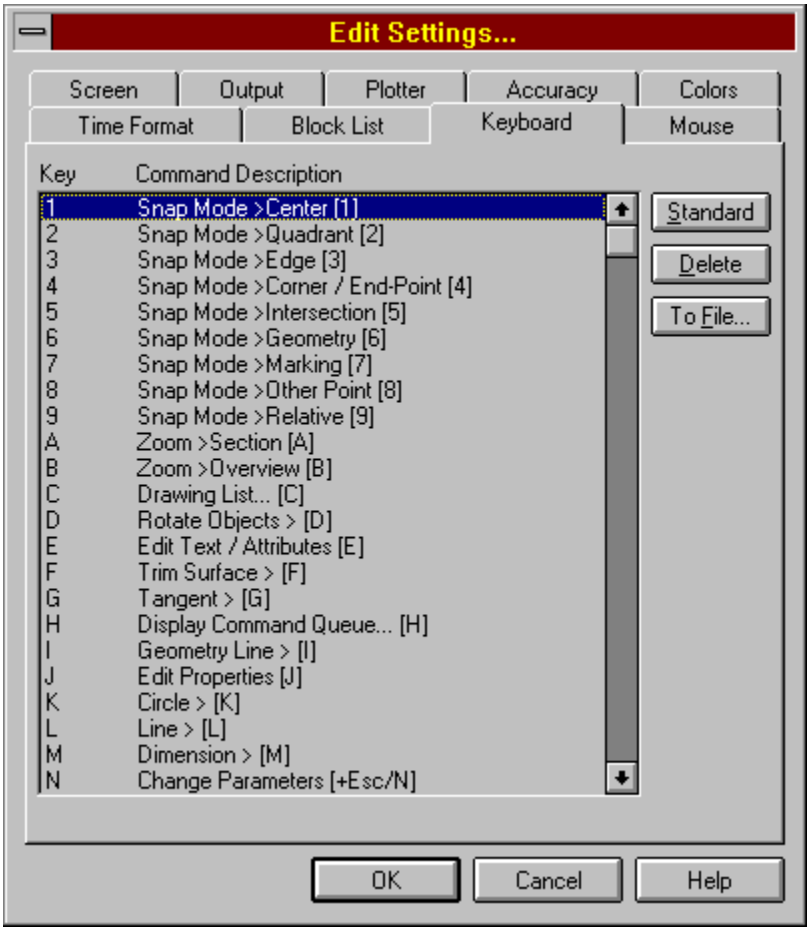


Associated command:

Settings>Colors

"Display Key Assignments" Dialog

How can I access information on this dialog?

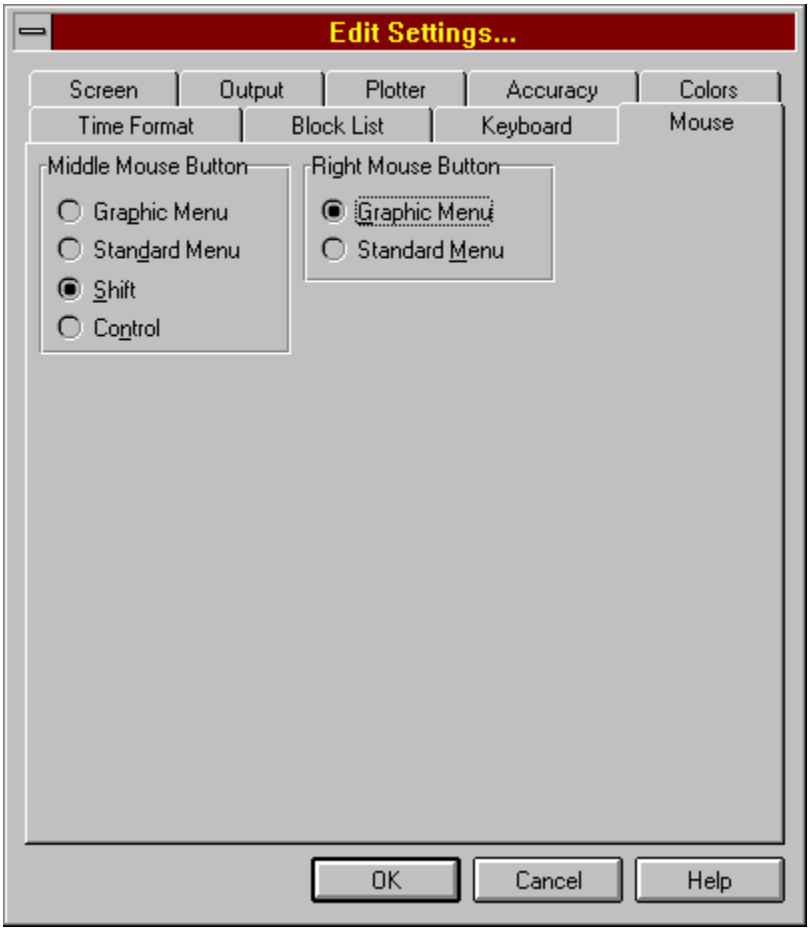


Associated command:

Settings>Display Key Assignments

"Mouse Button Usage Settings" Dialog

How can I access information on this dialog?



Associated command:

Settings>Mouse Button Usage

"Change Key Assignment" Dialog

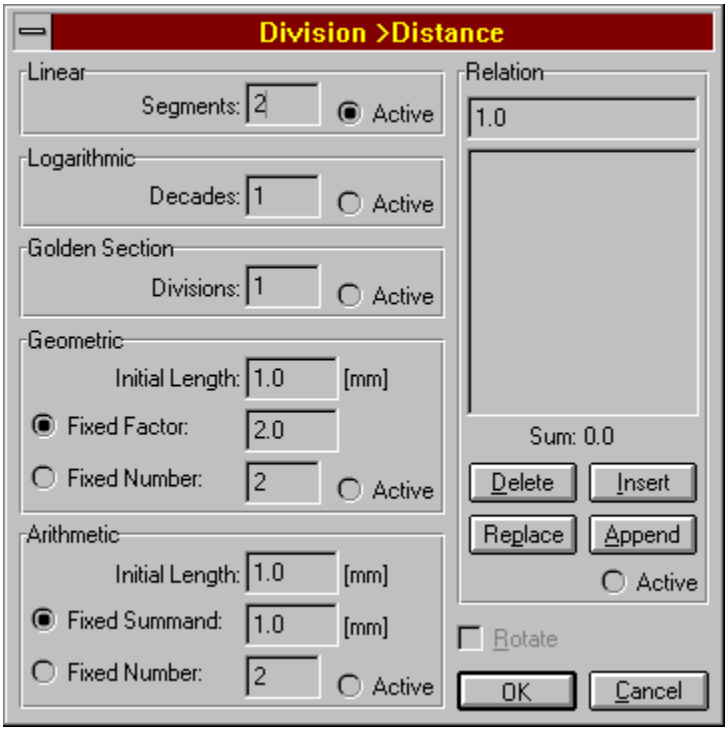
How can I access information on this dialog?

Associated command:

Settings>Change Key Assignment

"Division" Dialog

How can I access information on this dialog?

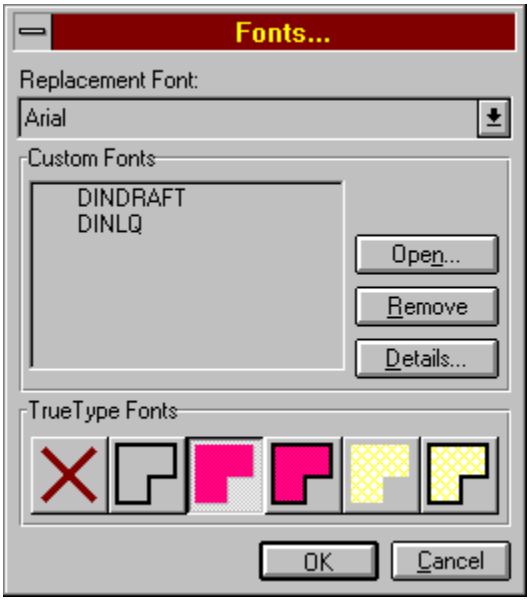


Associated commands:

- Division>Distance
- Division>Object
- Division>Array
- Multiple Copy>Distance
- Multiple Copy>Object
- Multiple Copy>Array

"Fonts" Dialog

How can I access information on this dialog?

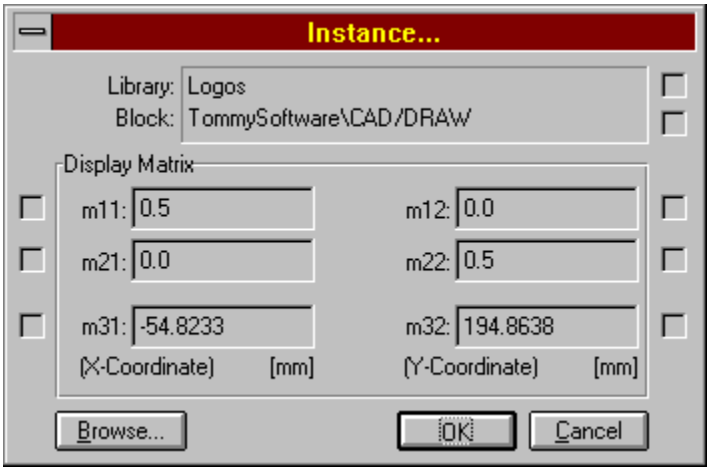


Associated command:

Fonts

"Instance" Dialog

How can I access information on this dialog?



The check boxes at the left and right of the dialog are only available if the properties of several objects are changed at the same time. In this case the initial setup of this window is the same as the setup of the first instance among the selected objects.

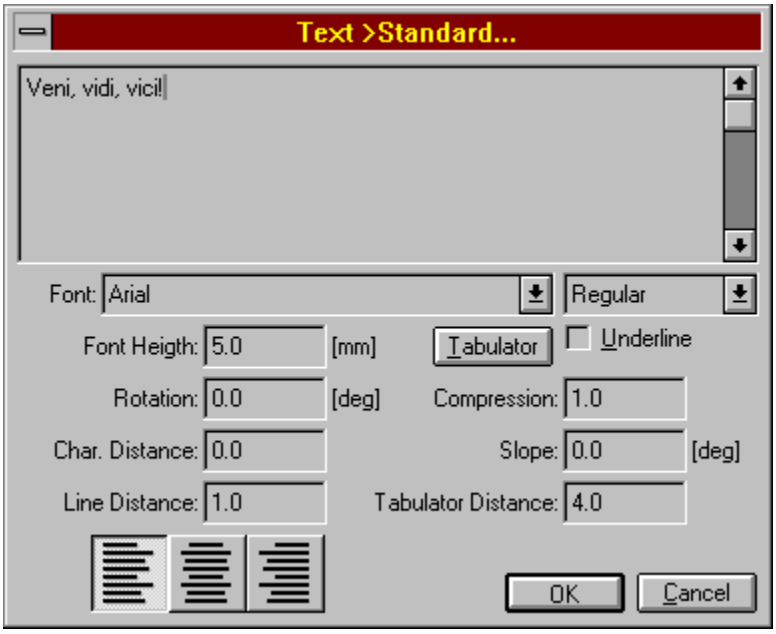
If only one object is selected, the check boxes are not visible. If only one object is chosen, this check box is not visible. In this case the initial setup of this window is the same as the setup of the selected object.

Associated command:

Edit Properties

"Enter Standard Text" Dialog

How can I access information on this dialog?



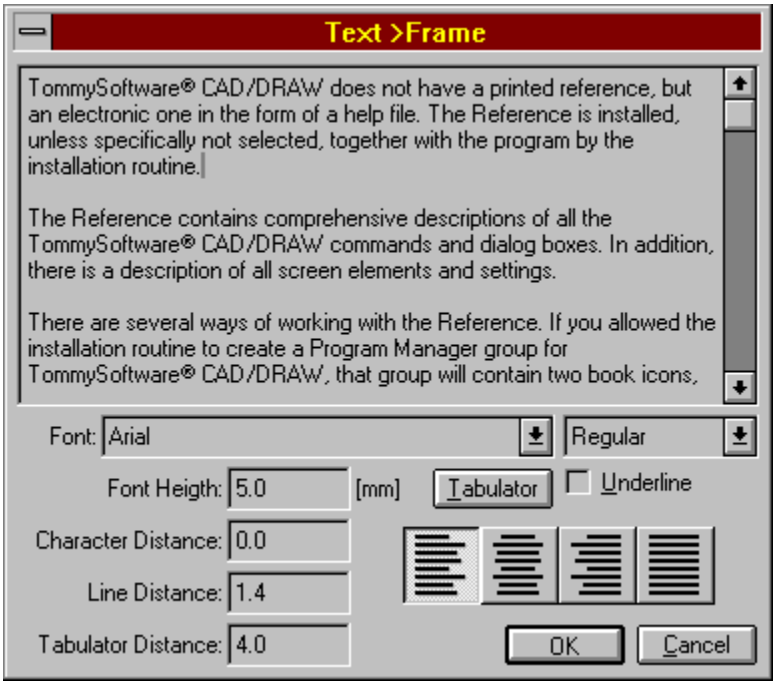
Associated commands:

Text>Standard

Text>Reference

"Enter Frame Text" Dialog

How can I access information on this dialog?



Associated command:

Text>Frame

"Font Selection" Dialog

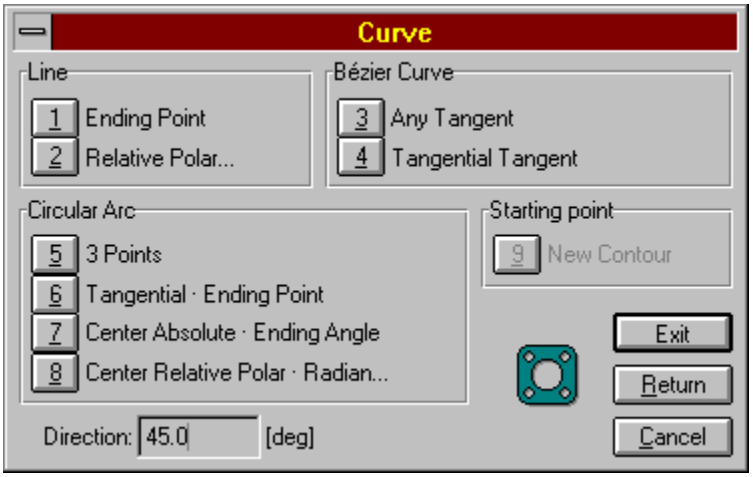
How can I access information on this dialog?

Associated command:

No command-specific usage

"Curve Entry" Dialog

How can I access information on this dialog?



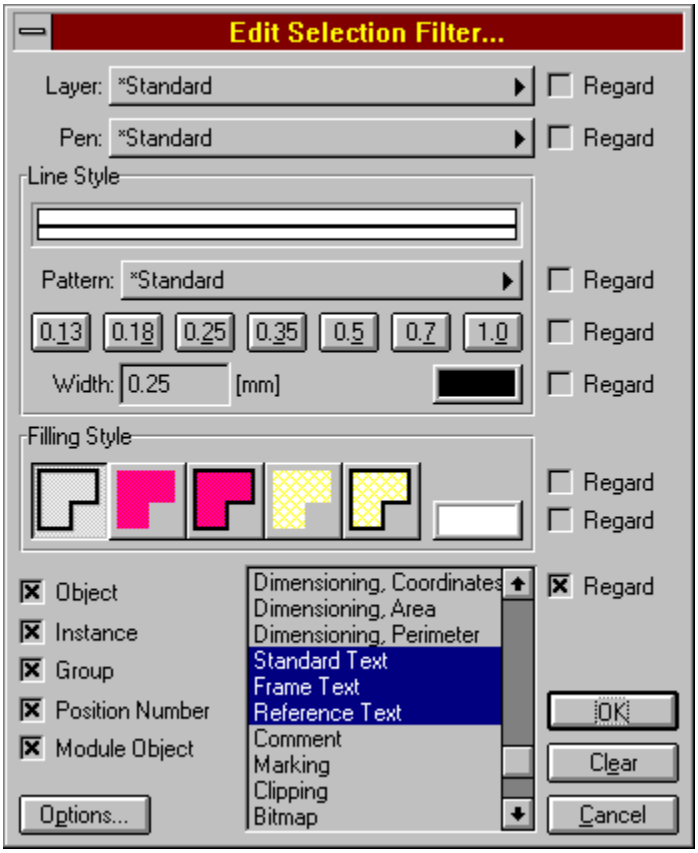
Associated commands:

Curve

Surface

"Selection Filter" Dialog

How can I access information on this dialog?



By activating a "Regard" check box only those objects showing the corresponding property can be selected, i.e. these objects are filtered out. If the "Regard" check box next to the list box is activated only those objects can be selected that are marked in the list box.

By clearing one of the check boxes in the lower left corner of the dialog the relevant object type will be completely excluded (in spite of all other dialog settings) from the selection.

To select all objects showing the fill mode "Outline" and the line color "Red" click the "Outline" button in the "Filling Style" group and select "Red" as the line color by clicking the color button in the "Line Style" group. The corresponding "Regard" check boxes are activated automatically. After applying the "OK" button press the F10 Key to select all objects, which means in this case selection of all objects showing the fill mode "Outline" and the line color "Red".

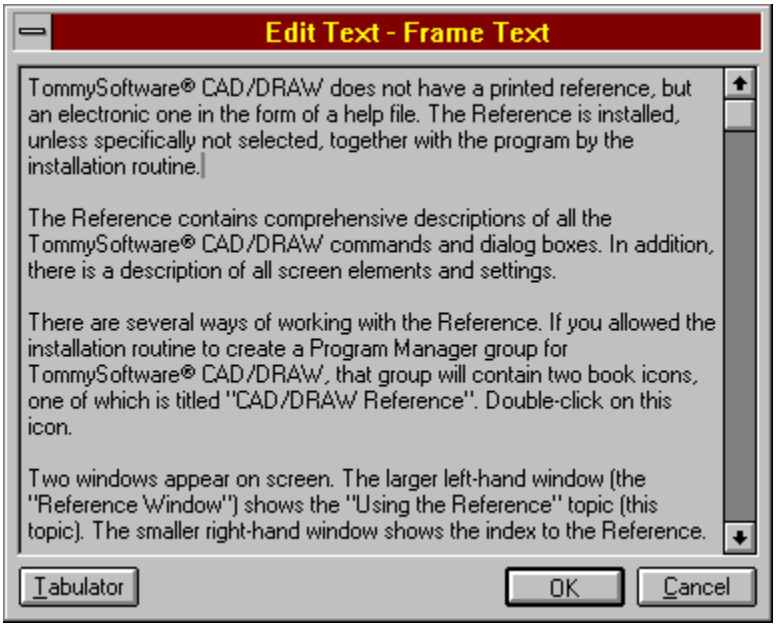
The filter remains active until it is explicitely deactivated, i.e. even if you select a new command the current selection filter continues to remain active.

Associated command:

Object Selection

"Edit Text" Dialog

How can I access information on this dialog?

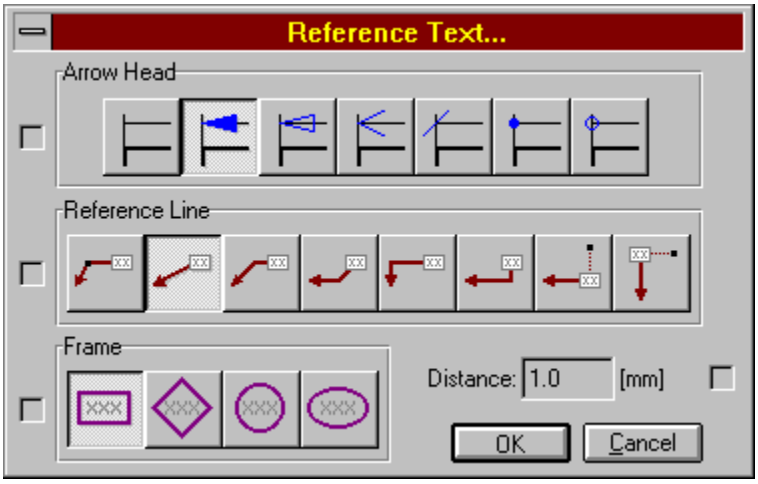


Associated command:

Edit Text

"Reference Text" Dialog

How can I access information on this dialog?



The check boxes at the left and right of the dialog are only shown when altering the properties of more than one object at once. In this case the properties shown in the dialog to start with are the properties of the first single line of text underneath the selected objects.

If only one object is selected, the check boxes are not visible. The properties shown in the dialog to start with are the properties of the selected object, which can then be altered directly.

Associated command:

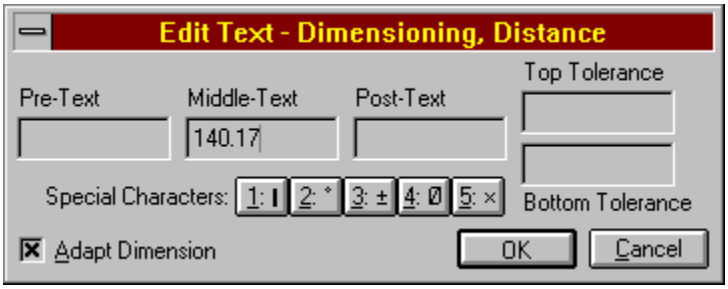
Edit Properties

Possible preceding dialog:

Edit Properties

"Dimension Number and Tolerances" Dialog

How can I access information on this dialog?

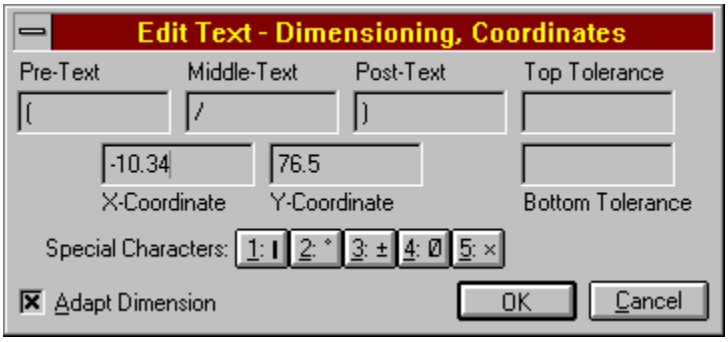


Associated commands:

- Edit Text
- Dimension>Length, Points
- Dimension>Length, Objects
- Dimension>Length, Object - Point
- Dimension>Length, Object - Object
- Dimension>Radius, Points
- Dimension>Radius, Object
- Dimension>Diameter, Points
- Dimension>Diameter, Object
- Dimension>Angle, Points
- Dimension>Angle, Line - Line
- Dimension>Angle, Circle Part
- Dimension>Arc Length, Points
- Dimension>Arc Length, Circle Part
- Calculate>Area
- Calculate>Perimeter / Length

"Coordinates and Tolerances" Dialog

How can I access information on this dialog?



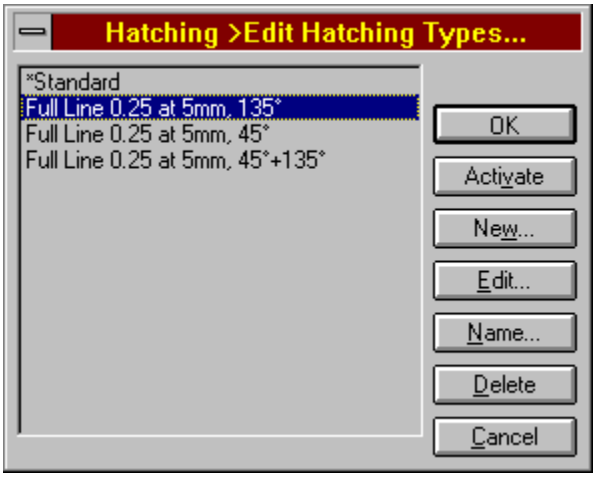
Associated commands:

Edit Text

Dimension>Coordinates

"Define Hatching Types" Dialog

How can I access information on this dialog?

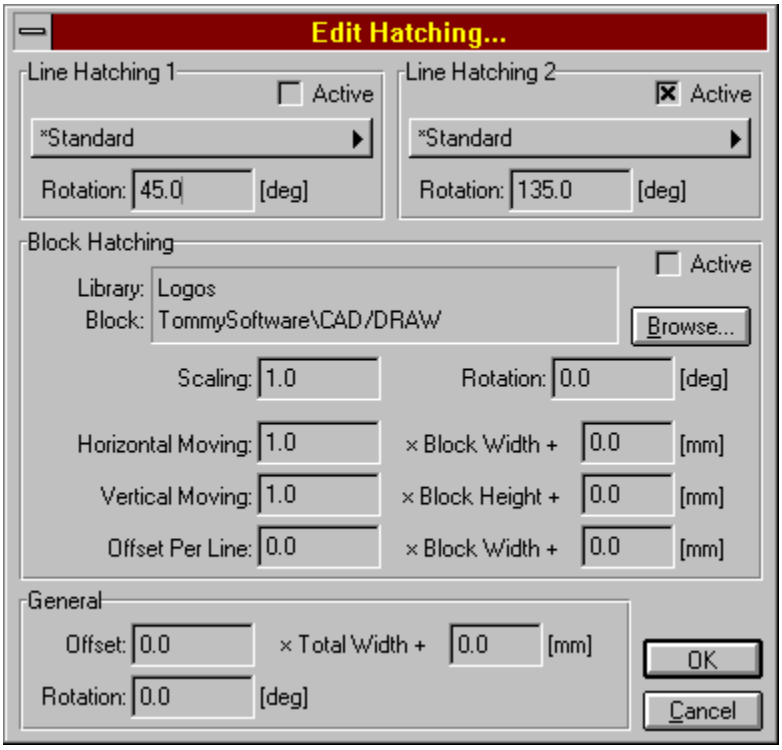


Associated command:

Hatching>Edit Hatching Types

"Edit Hatching Types" Dialog

How can I access information on this dialog?



Associated command:

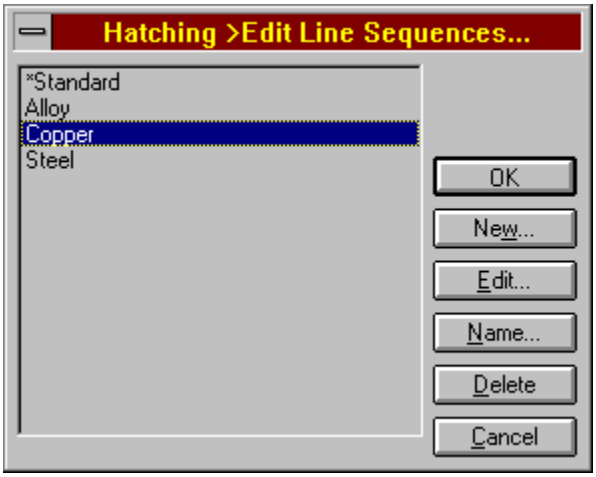
Hatching>Edit Hatching Types

Possible preceding dialog:

Define Hatching Types

"Define Line Sequences" Dialog

How can I access information on this dialog?

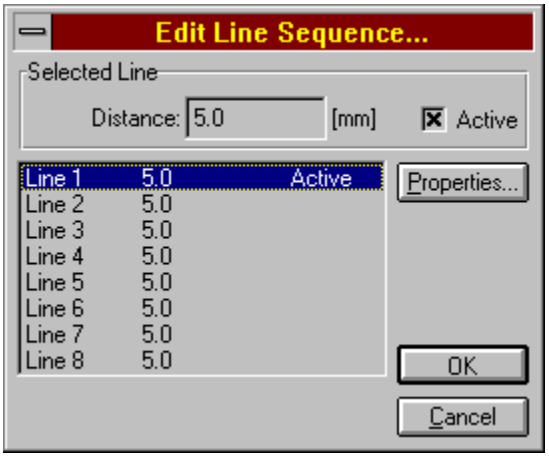


Associated command:

Hatching>Edit Line Sequences

"Edit Line Sequences" Dialog

How can I access information on this dialog?



Associated command:

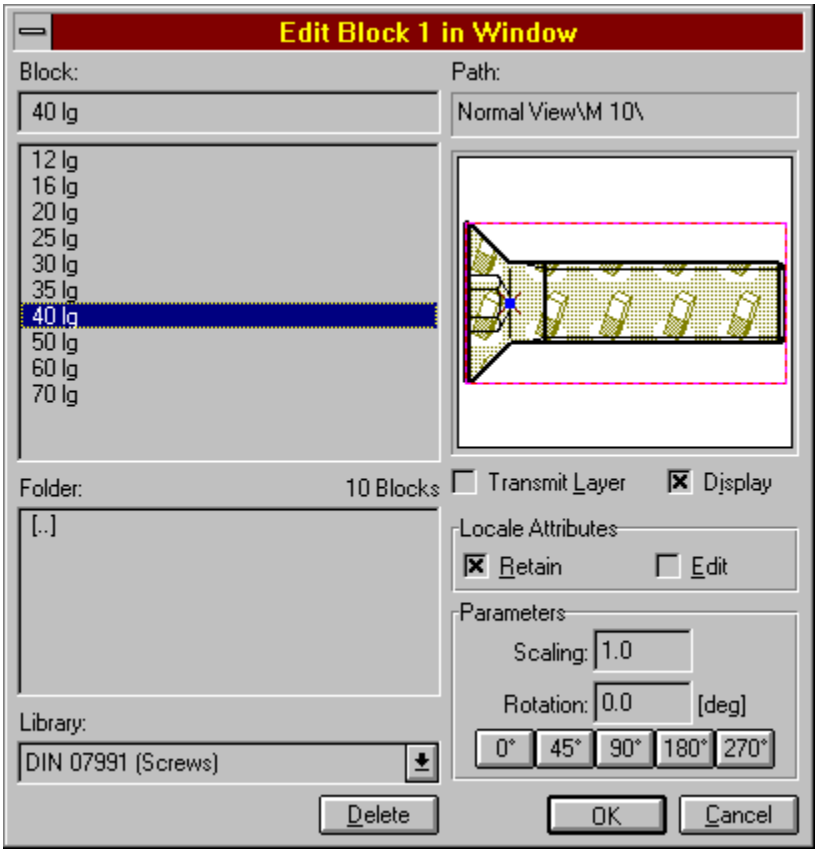
Hatching>Edit Line Sequences

Possible preceding dialog:

Define Line Sequences

"Insert Block" Dialog

How can I access information on this dialog?



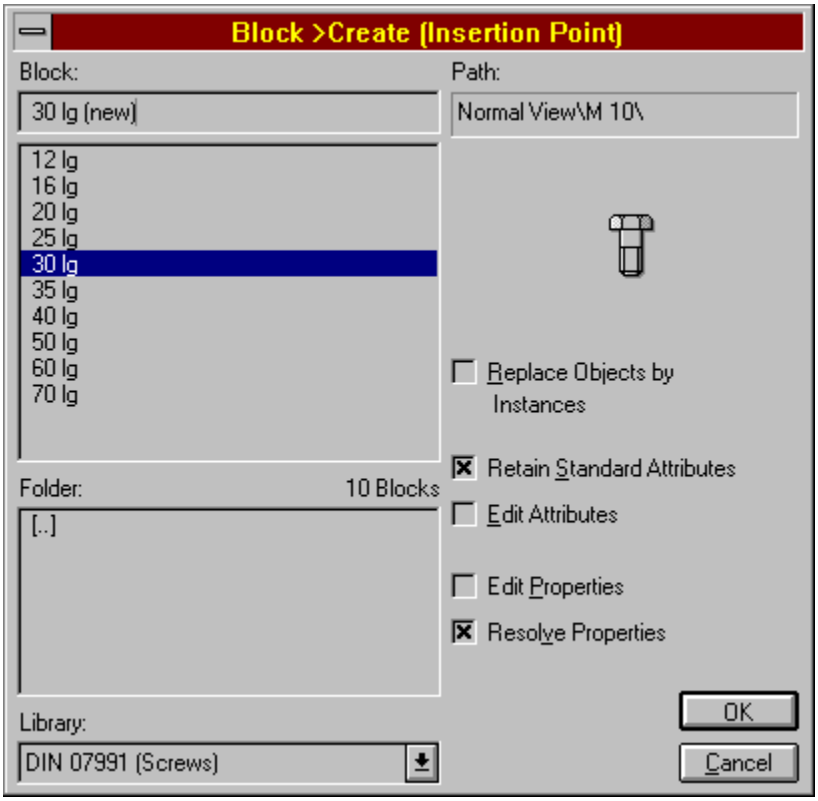
Associated commands:

Block>Insert

Define Block in Window

"Generate Block" Dialog

How can I access information on this dialog?



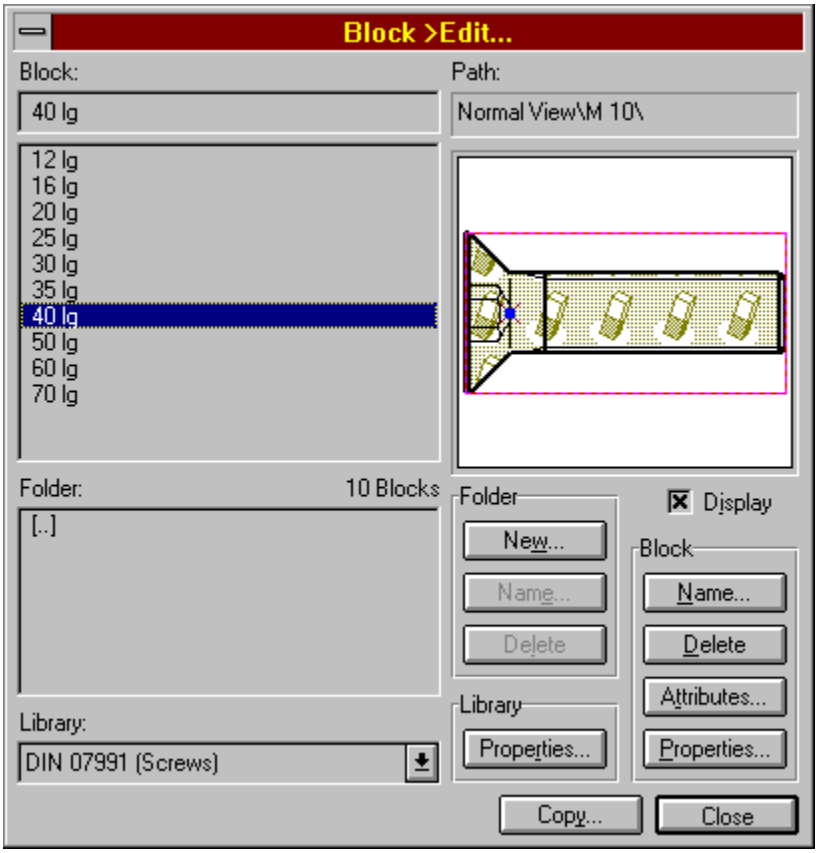
Associated commands:

Block>Create (Insertion Point)

Block>Create (Frame)

"Edit Block" Dialog

How can I access information on this dialog?

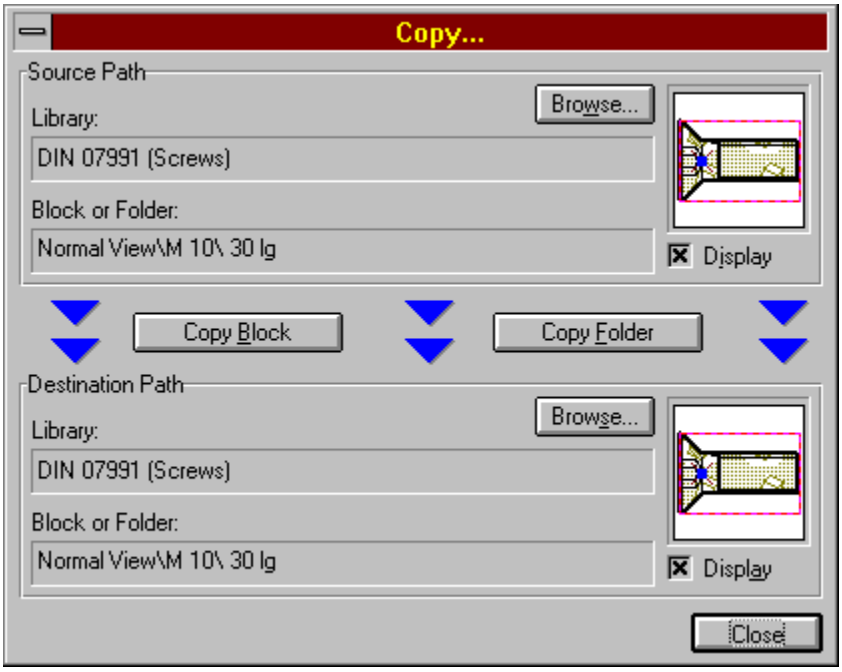


Associated command:

Block>Edit

"Copy Block" Dialog

How can I access information on this dialog?



Associated command:

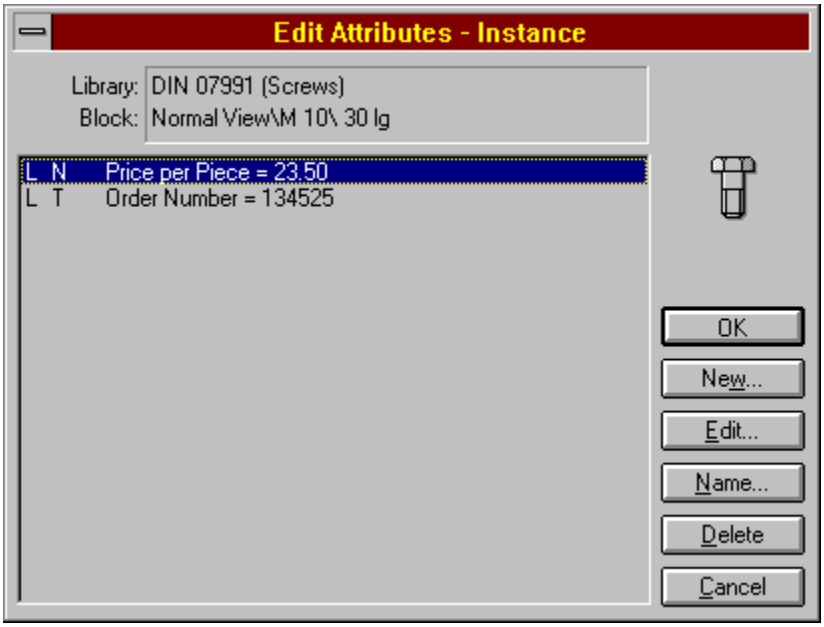
Block>Edit

Possible preceding dialog:

Edit Block

"Define Attributes" Dialog

How can I access information on this dialog?

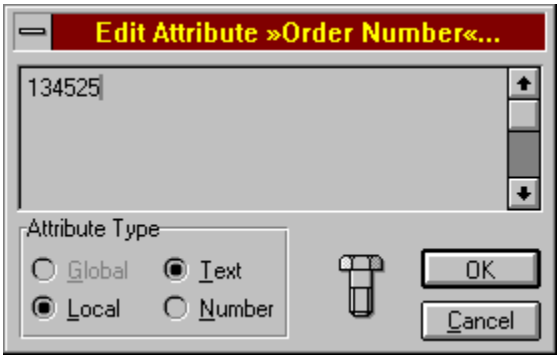


Associated command:

Edit Text

"Edit Attributes" Dialog

How can I access information on this dialog?



Associated command:

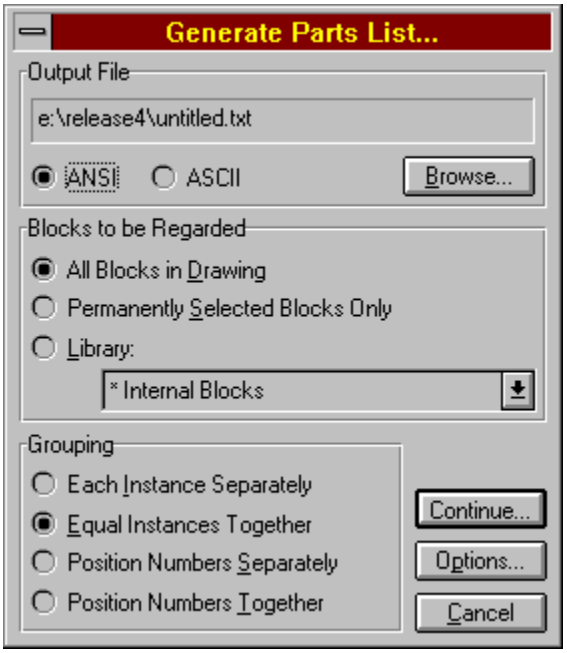
Edit Text

Possible preceding dialog:

Define Attributes

"Generate Parts List" Dialog

How can I access information on this dialog?

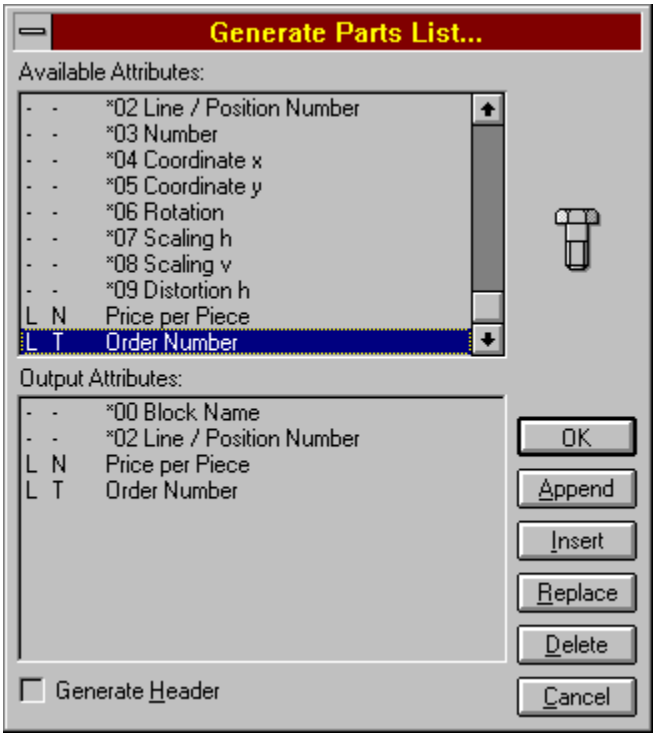


Associated command:

Generate Parts List

"Choose Parts List Attributes" Dialog

How can I access information on this dialog?



Associated command:

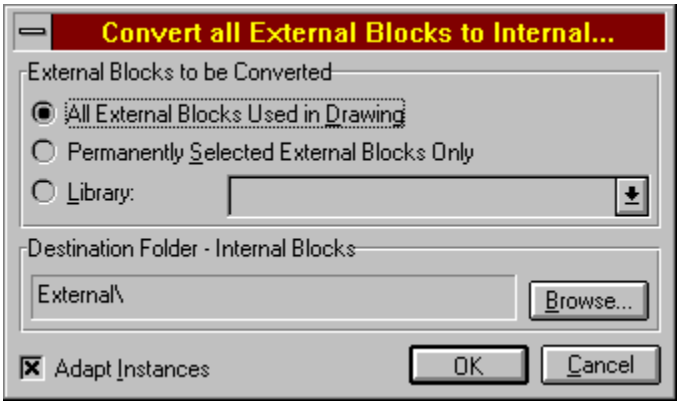
Generate Parts List

Possible preceding dialog:

Generate Parts List

"Convert External Blocks" Dialog

How can I access information on this dialog?

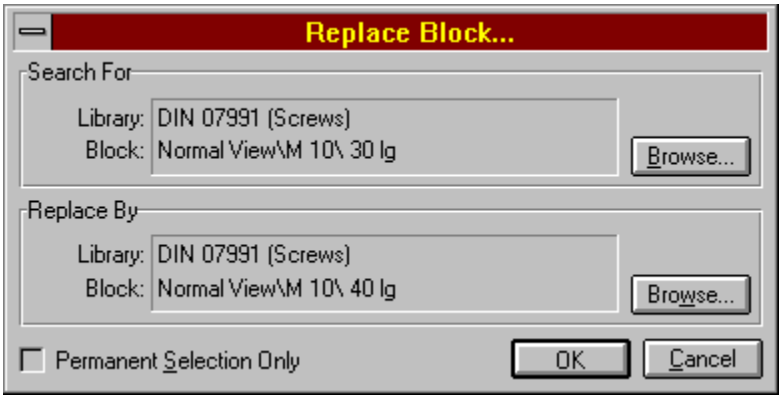


Associated command:

Convert External Blocks

"Replace Block Name" Dialog

How can I access information on this dialog?

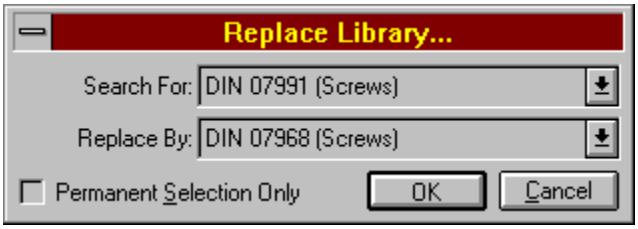


Associated command:

Replace Block Name

"Replace Library Name" Dialog

How can I access information on this dialog?

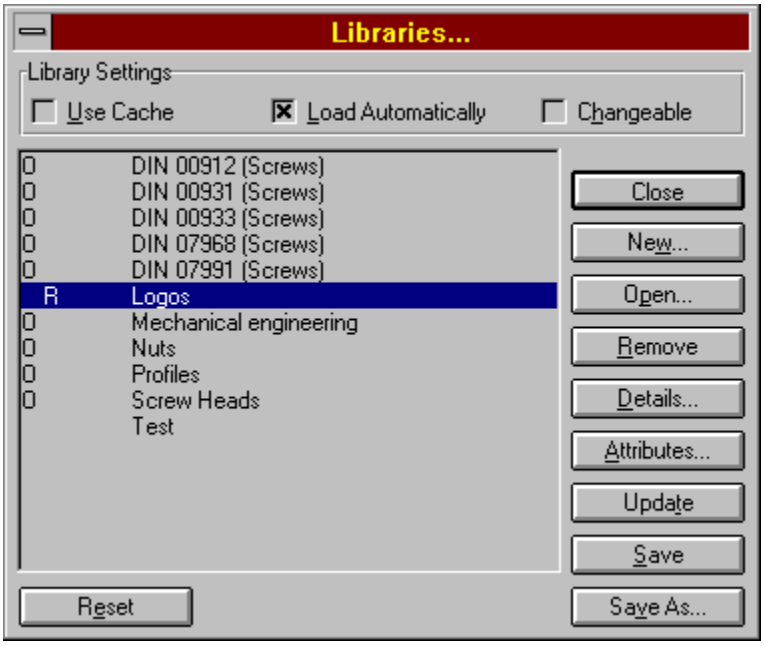


Associated command:

Replace Library Name

"Libraries" Dialog

How can I access information on this dialog?

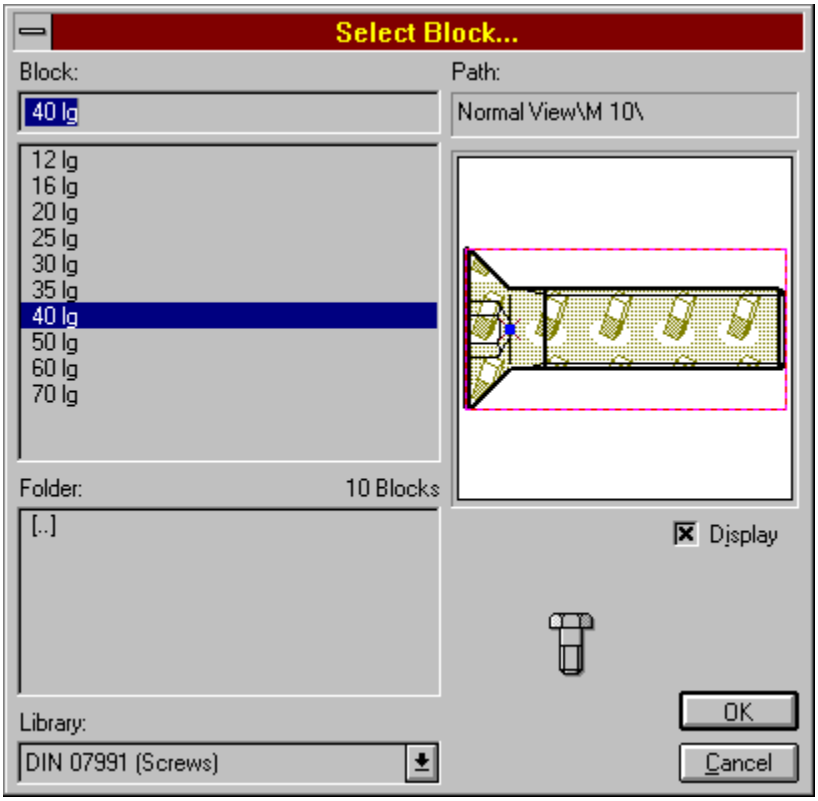


Associated command:

Libraries

"Select Block" Dialog

How can I access information on this dialog?



Associated commands:

No command-specific usage

Possible preceding dialogs:

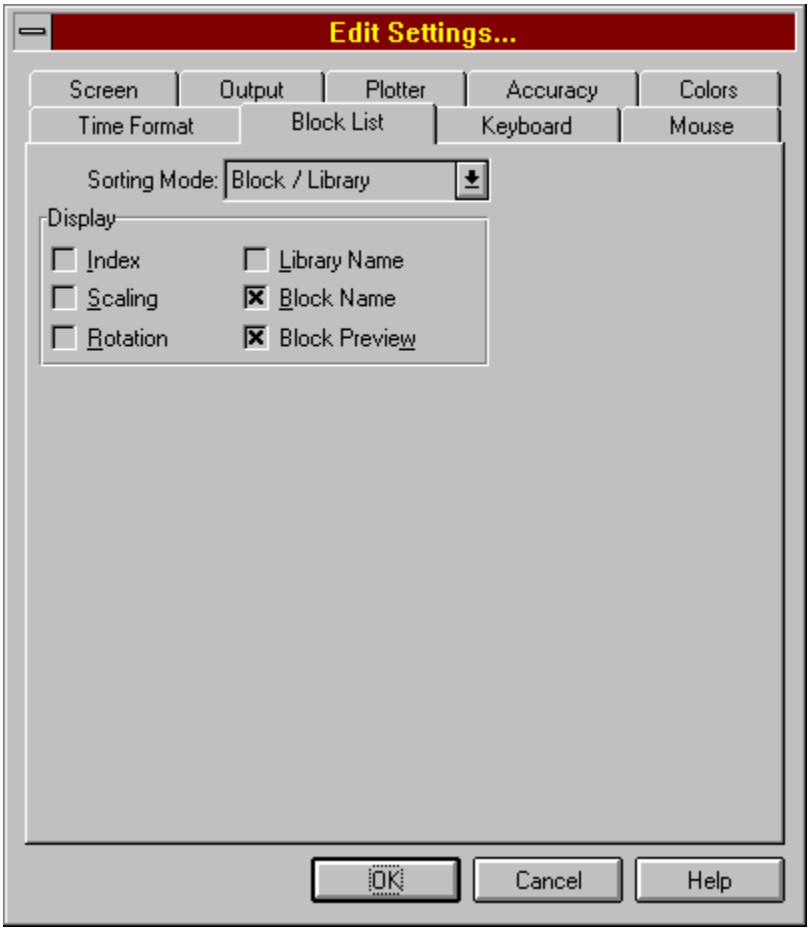
Instance

Copy Block

Replace Block Name

"Settings Block List" Dialog

How can I access information on this dialog?

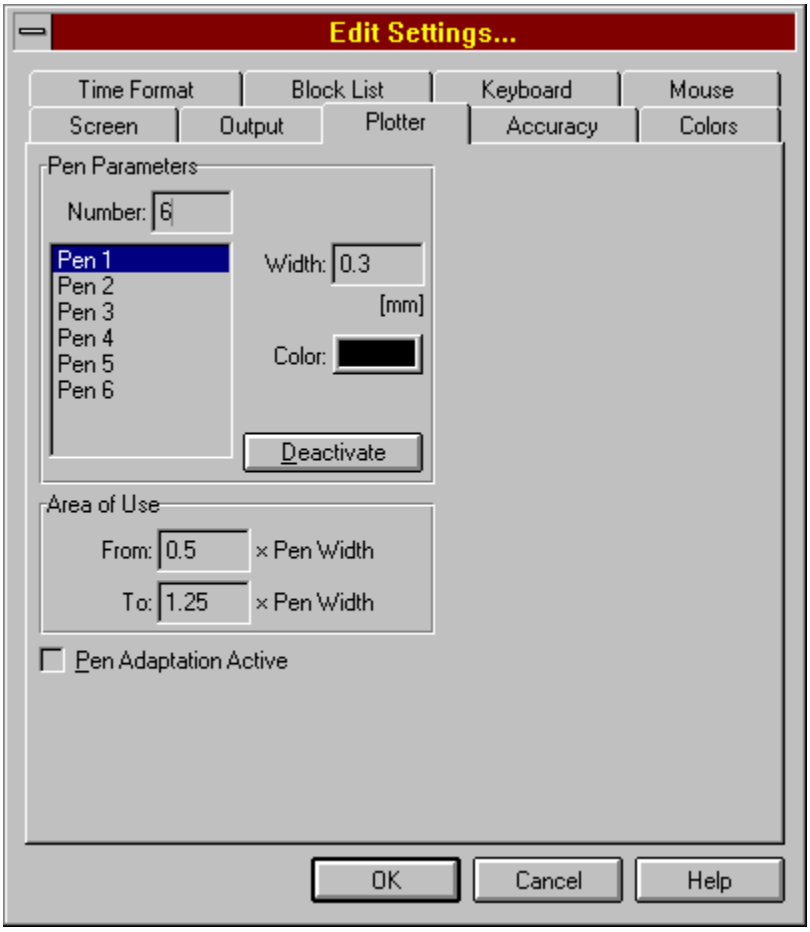


Associated command:

Settings>Block List

"Plotter Settings" Dialog

How can I access information on this dialog?



Before you alter the settings in this dialog, it is *essential* that you read the section on Notes on Plotter Output fully. It contains important information on the correct use of plotters.

Associated commands:

Print Drawing

Print Section

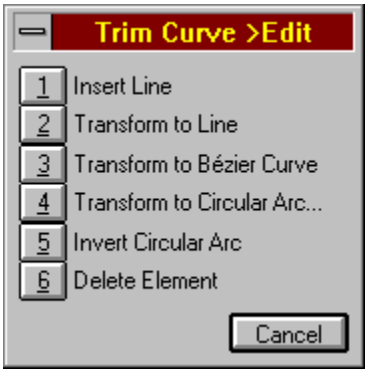
Settings>Plotter

Possible preceding dialog:

Print Drawing

"Edit Curve" Dialog

How can I access information on this dialog?



Associated command:

Trim Curve>Edit

"Message" Dialog

How can I access information on this dialog?

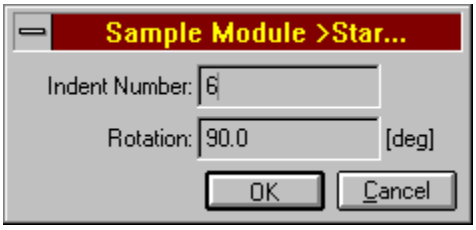
This type of dialog can appear for many reasons. Whatever the message, the program is interrupted until it is closed.

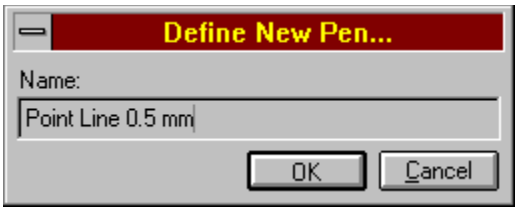
Associated command:

No command-specific usage

"General Entry" Dialog

How can I access information on this dialog?





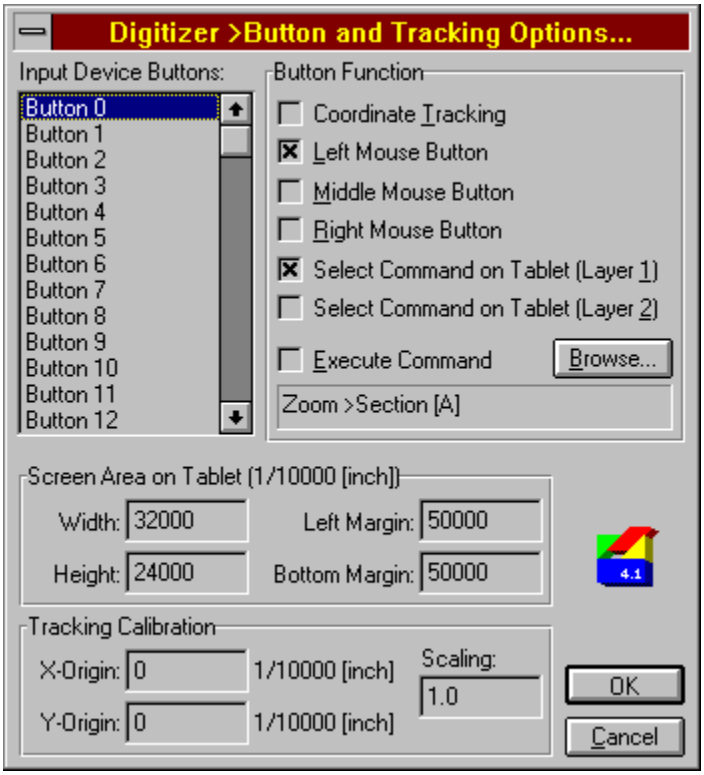
This, or a similar Dialog can appear for many reasons. Whatever the message, the program is interrupted until the required text or values are entered and it is closed.

Associated command:

No command-specific usage

"Digitizer Button and Tracking Options" Dialog

How can I access information on this dialog?

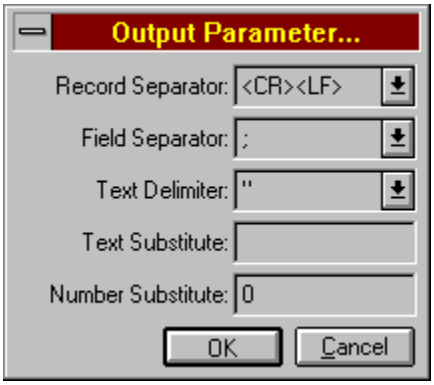


Associated command:

Digitizer>Button and Tracking Options

"Parts List Options" Dialog

How can I access information on this dialog?



Associated command:

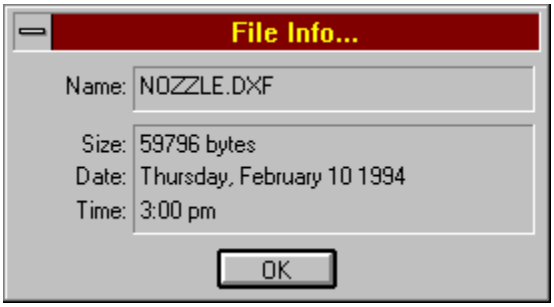
Generate Parts List

Possible preceding dialog:

Generate Parts List

"File Info" Dialog

How can I access information on this dialog?



Associated command:

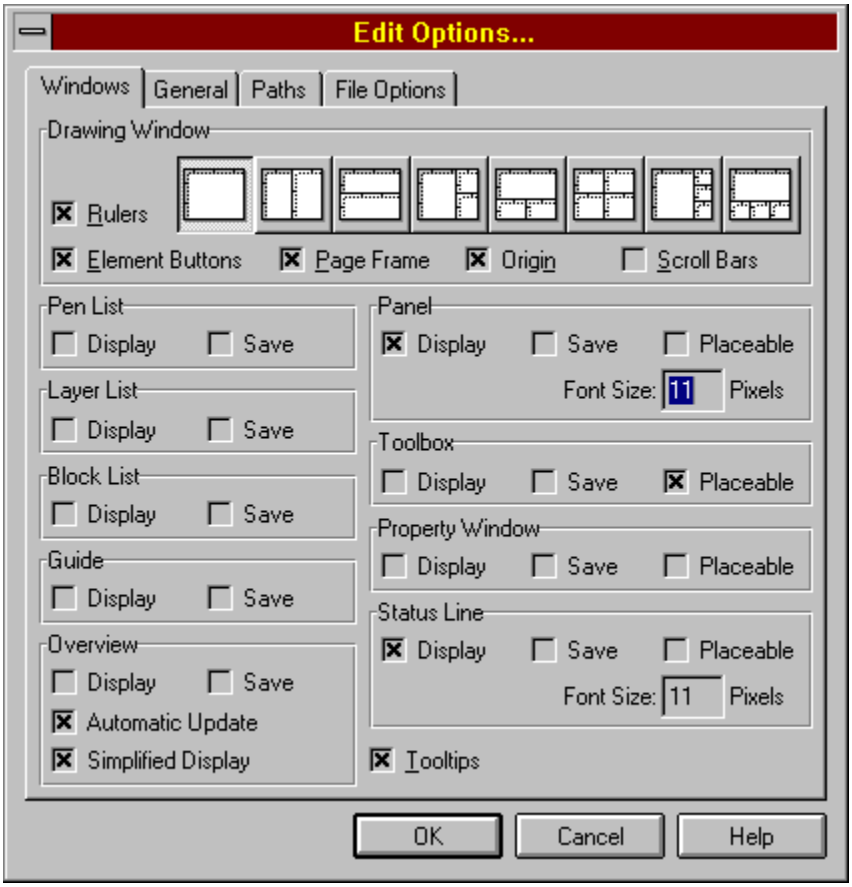
No command-specific usage

Possible preceding dialog:

File Selection

"Window Layout" Dialog

How can I access information on this dialog?

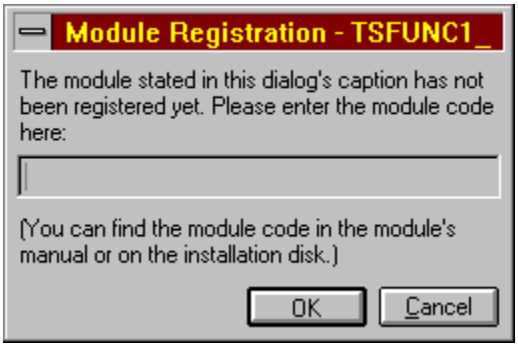


Associated command:

Options>Windows

"Register Module" Dialog

How can I access information on this dialog?



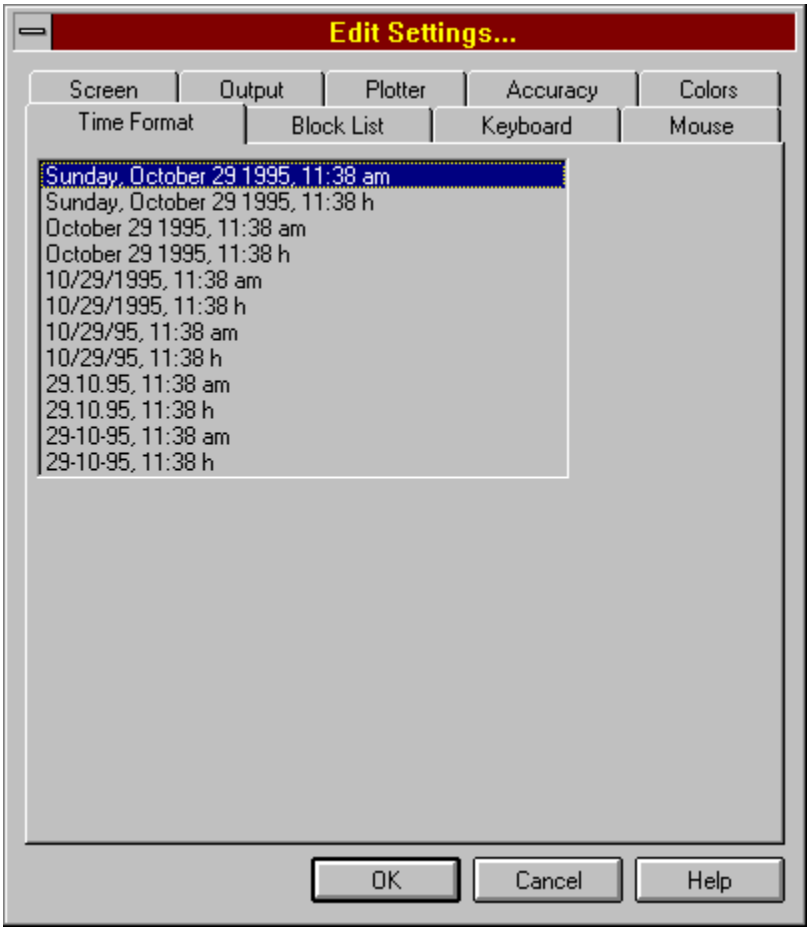
This dialog appears during start-up if TommySoftware® CAD/DRAW detects a module that has to be registered, but has not been registered yet. In this dialog, you have to enter the correct module-code in order to be able to use that module.

Associated command:

No command-specific usage

"Time Format Settings" Dialog

How can I access information on this dialog?

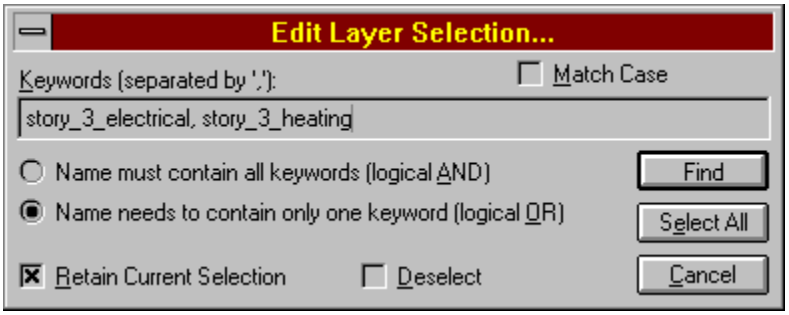


Associated command:

Settings>Time Format

"Layer Selection" Dialog

How can I access information on this dialog?



Associated command:

No command-specific usage

Click the left mouse button on any element in the illustrated dialog window to see an explanation of that element. Explanations for an element are available when the mouse pointer changes from an insertion mark (I) to a hand with a pointing finger (
).

After left-clicking an element, a floating window with information about the clicked dialog element appears. If this window contains jumps, they can be selected as usual. To close this window, left-click again, or press ESC or any other key.

In some cases, the illustration may show some elements that are currently not visible in the application. This is *not* an error, but is neccessary to show and explain all possible dialog window appearances within one illustration.

For information on the command that called this dialog window, please use one of the jumps listed at the end of this topic.

Press the F1 key to get further information on using the help system.

Clicking on this button will close the dialog accepting all changes. Any changes or operations specified will be carried out.

Clicking on this button will close the dialog, accepting any changes made in the dialog.

Clicking on this button will close the dialog, without accepting any changes. Any following operation will not be carried out.

Clicking on this button displays the help text for this dialog window.

This command or function is only available in Level 2 of TommySoftware® CAD/DRAW 4.

This command or function is only available in the "Test before you buy" version of TommySoftware® CAD/DRAW 4.

The layer is chosen from this list. All currently defined layers are included in the list.

If this check box is enabled the specified layer will be applied to all selected objects. It is enabled automatically as soon as a layer is chosen.

This check box shows if and how the layer transmission is to be changed. If the check box is gray, no alterations are made. If it is marked the layer transmission is enabled for all selected objects, otherwise it is disabled.

The pen is chosen from this list. All currently defined pens are included in the list.

If this check box is enabled the specified pen will be applied to all selected objects. It is enabled automatically as soon as a pen is chosen.

This check box shows if and how the pen transmission is to be changed. If the check box is gray, no alterations are made. If it is marked the pen transmission is enabled for all selected objects, otherwise it is disabled.

An example of the current line pattern is shown in this field.

The line pattern is chosen from this list. All currently defined line patterns are included in the list.

If this check box is enabled the specified line pattern will be applied to all selected objects. It is enabled automatically as soon as a line pattern is chosen.

This check box shows if and how the line pattern transmission is to be changed. If the check box is gray, no alterations are made. If it is marked the line pattern transmission is enabled for all selected objects, otherwise it is disabled.

The line width is entered into this field. The width is specified in the current units.

This button sets the line width to 0.13 mm.

This button sets the line width to 0.18 mm.

This button sets the line width to 0.25 mm.

This button sets the line width to 0.35 mm.

This button sets the line width to 0.5 mm.

This button sets the line width to 0.7 mm.

This button sets the line width to 1,0 mm.

If this check box is enabled the specified line width will be applied to all selected objects. It is enabled automatically as soon as the line width is changed.

This check box shows if and how the line width transmission is to be changed. If the check box is gray, no alterations are made. If it is marked the line width transmission is enabled for all selected objects, otherwise it is disabled.

The line color is specified from this color field. Clicking in this field with the left mouse button calls the Choose Color dialog, where the correct color can be chosen.

If this check box is enabled the specified line color will be applied to all selected objects. It is enabled automatically as soon as the line color is changed.

This check box shows if and how the line color transmission is to be changed. If the check box is gray, no alterations are made. If it is marked the line color transmission is enabled for all selected objects, otherwise it is disabled.

This icon represents "Outline" fill mode. An object with this fill mode will only have its outline drawn.

This icon represents "Filling" fill mode. An object with this fill mode will only have its filling drawn. Objects which have no filling (e.g. lines) are not visible if this filling mode is applied to them.

This icon represents "Filling & Outline" fill mode. An object with this fill mode will be drawn with both its filling and outline. Objects which do not have fillings (e.g. lines) only have their outlines drawn.

This icon represents "Eraser" fill mode. An object with this fill mode will only have its filling drawn in background color. Objects which have no filling (e.g. lines) are not visible if this filling mode is applied to them.

This icon represents "Eraser & Outline" fill mode. An object with this fill mode will be drawn with both a filling in background color and outline. Objects which do not have fillings (e.g. lines) only have their outlines drawn.

If this check box is enabled the specified fill mode will be applied to all selected objects. It is enabled automatically as soon as the fill mode is changed.

This check box shows if and how the fill mode transmission is to be changed. If the check box is gray, no alterations are made. If it is marked the fill mode transmission is enabled for all selected objects, otherwise it is disabled.

The fill color is chosen from this color field. Clicking in this field with the left mouse button calls the Choose Color dialog, where the correct color can be chosen.

If this check box is enabled the specified fill color will be applied to all selected objects. It is enabled automatically as soon as the fill color is changed.

This check box shows if and how the fill color transmission is to be changed. If the check box is gray, no alterations are made. If it is marked the fill color transmission is enabled for all selected objects, otherwise it is disabled.

If this check box is enabled a copy of the selected objects will be created and initialized with the edited properties. Otherwise the objects themselves will be altered.

In most cases, one or more dialogs where object-specific parameters can be specified can be called with these buttons. The following special dialogs are possible:

- Standard Text
- Frame Text
- Reference Text
- Zigzag Line
- Dimension Line
- General Dimension Parameters
- Dimension with Line
- Dimension without Line
- Instance
- Bitmap

If the button shows the text "Special..." (three points at the end), only one special dialog is available. This is called when the button is pressed. If the button shows the text "Special >" (an arrow at the end), then several dialogs are available. When the button is pressed, a small menu appears from which the dialog can be chosen. If no special dialog is available, then the button is not visible.

By clicking this tab the standard object properties or the properties of dimension lines are displayed and can be edited.

By clicking this tab the text object properties or the properties of dimension texts are displayed and can be edited.

The font to be used is chosen from this list. All available TrueType fonts and all loaded TommySoftware® CAD/DRAW fonts are shown. If the Adobe Type Manager is present and active, all available PostScript and device fonts will also be listed.

The font style to be used can be chosen from this list. Different styles are available, according to the font chosen. TommySoftware® CAD/DRAW fonts do not have styles.

If this check box is enabled, the font is displayed underlined. This option is not available for TommySoftware® CAD/DRAW fonts.

If this check box is enabled, the font parameters (name, style and underlining) are transferred to all objects. It changes automatically, as soon as the font parameters are changed.

The font size to be used is entered in this field. The value is entered in the current line unit. Note that a unit can be specified immediately after the value, e.g. `10pt`.

If this check box is enabled, the font size specified is transferred to all objects. It changes automatically, as soon as the font size is changed.

The compression factor to use for the font, relative to 1.0, is entered in this field.

If this check box is enabled, the specified compression factor is applied to all objects. It changes automatically, as soon as the compression factor is changed.

The font rotation angle to be used is entered in this field. The value is entered in the current angle unit.

If this check box is enabled, the rotation angle specified is transferred to all objects. It changes automatically as soon as the rotation angle is changed.

The slope angle for the font is entered in this field. The value is entered in the current angle unit.

If this check box is enabled, the specified slope angle is used for all objects. It changes automatically, as soon as the slope angle is changed.

The spacing between characters is specified in this field. The value is entered relative to the font height. Values between -100 and +100 are allowed.
Note that there is no "internal" character spacing for TommySoftware® CAD/DRAW's own fonts. A character spacing of at least 0.1 should be used with TommySoftware® CAD/DRAW fonts. On the other hand, if TrueType fonts are used, the value should be left at 0.0.

If this check box is enabled, the specified character spacing is used for all objects. It changes automatically, as soon as the character spacing is changed.

The tab spacing for the font is entered in this field. The value is specified relative to the font height. Values between -100 and +100 are allowed.

If this check box is enabled, the specified tab spacing is applied to all objects. It changes automatically, as soon as the tab spacing is changed.

The line spacing is entered here. The value is entered relative to the font height. Values between -100 and +100 are allowed.

If this check box is enabled, the specified line spacing is applied to all objects. It changes automatically, as soon as the line spacing is changed.

The X position of the text is entered here. The value is entered in the current length unit.
The X position, together with the Y position, describes the text's "fixed point". According to the text mode, this is the left, center or right point on the base line of the text.

If this check box is enabled, the specified X position is applied to all objects. It changes automatically, as soon as the X position is changed.

The Y position of the text is entered here. The value is entered in the current length unit.
The Y position, together with the X position, describes the text's "fixed point". According to the text mode, this is the left, center or right point on the base line of the text.

If this check box is enabled, the specified Y position is applied to all objects. It changes automatically, as soon as the Y position is changed.

This icon represents the text mode "Aligned Left".

This icon represents the text mode "Centered".

This icon represents the text mode "Aligned Right".

This icon represents the text mode "Justified".

If this check box is enabled, the specified text mode is applied to all objects. It changes automatically as soon as the text mode is changed.

The spacing between zags to be used is entered in this field. The value is entered in the current line unit.

If this check box is enabled, the specified spacing between zags is applied to all objects. It changes automatically, as soon as the spacing between the zags is changed.

This icon represents the arrow style "No Symbol". If the adjacent check box is enabled, no symbol is shown at the start of dimension lines.

This icon represents the arrow style "Filled Arrow". If the adjacent check box is enabled at the start of the dimension line a filled arrow with an opening angle of 20° is shown.

This icon represents the arrow style "Unfilled Arrow ". If the adjacent check box is enabled at the start of the dimension line an unfilled arrow with an opening angle of 20° is displayed/

This icon represents the arrow style "Open Arrow". If the adjacent check box is enabled at the start of the dimension line an open arrow with an opening angle of 60° is displayed.

This icon represents the arrow style "Sloping Line". If the adjacent check box is enabled at the start of the dimension line a 45° sloping line is displayed.

This icon represents the arrow style "Point". If the adjacent check box is enabled at the start of the dimension line a filled circle is displayed.

This icon represents the arrow style "Circle". If the adjacent check box is enabled at the start of the dimension line an unfilled circle is displayed.

If this check box is enabled the arrow style specified for the start of dimension lines is applied to all objects. It changes automatically, as soon as the arrow style for the start of dimension lines is changed.

The orientation for the arrow at the start of dimension lines can be chosen from this list. There are three possibilities:

"Not Rotated"
: The arrow is not rotated, i.e. it points from inside to outside. The dimension line ends at the extension line.

"Rotated"
: The arrow is rotated, i.e. it points from outside to inside. The dimension line runs over the extension lines.

"Automatic"
: The arrow is normally shown unrotated. However, if the dimension line is shorter than 5 mm, the arrow is automatically rotated.

If this check box is enabled the specified dimension line arrow orientation is applied to all objects. It changes automatically as soon as the orientation of the dimension line arrow is changed.

This icon represents the arrow style "No Symbol". If the adjacent check box is enabled, no symbol is shown at the end of the dimension line.

This icon represents the arrow style "Filled Arrow". If the adjacent check box is enabled a filled arrow with an opening angle of 20° is shown at the end of the dimension line.

This icon represents the arrow style "Unfilled Arrow ". If the adjacent check box is enabled an unfilled arrow with an opening angle of 20° is shown at the end of the dimension line.

This icon represents the arrow style "Open Arrow ". If the adjacent check box is enabled an open arrow with an opening angle of 60° is shown at the end of the dimension line.

This icon represents the arrow style "Sloping Line". If the adjacent check box is enabled an arrow sloping at 45° is shown at the end of the dimension line.

This icon represents the arrow style "Point". If the adjacent check box is enabled a filled circle is shown at the end of the dimension line.

This icon represents the arrow style "Circle". If the adjacent check box is enabled an unfilled circle is shown at the end of the dimension line.

If this check box is enabled, the specified arrow type for the ends of dimension lines is applied to all objects. It changes automatically, as soon as the arrow style for the end of dimension lines is changed.

The orientation for the arrow at the end of dimension lines can be chosen from this list. There are three possibilities:

"Not Rotated"
: The arrow is not rotated, i.e. it points from inside to outside. The dimension line ends at the extension line.

"Rotated"
: The arrow is rotated, i.e. it points from outside to inside. The dimension line runs over the extension lines.

"Automatic"
: The arrow is normally shown unrotated. However, if the dimension line is shorter than 5 mm, the arrow is automatically rotated.

If this check box is enabled the specified orientation of the arrow at the end of the dimension line is applied to all objects. It changes automatically as soon as the orientation of the arrow at the end of the dimension line is changed.

The font to be used is chosen from this list. All available TrueType fonts and all loaded TommySoftware® CAD/DRAW fonts are shown. If the Adobe Type Manager is present and active, all available PostScript and device fonts will also be listed.

The font style to be used is chosen from this list. Various styles are available, according to the font selected. TommySoftware® CAD/DRAW fonts do not have any special styles.

If this check box is enabled, the specified font parameters (name and style) are applied to all objects. It changes automatically, as soon as the font parameters are changed.

This list is used to select the desired coordinate system. It lists all coordinate systems that are currently defined.
If the selected coordinate system uses a distorted display for isometric or dimetric design, all dimensions based on this coordinate system will be adapted accordingly. See also Coordinate Systems>Edit.

If this check box is enabled, the specified coordinate system is applied to all objects. It changes automatically, as soon as a coordinate system is selected.

The font size for the main dimension is entered in this field. The value is entered in the current line unit. Note that a unit can be specified immediately after the value, e.g. `10pt`.

If this check box is enabled, the specified font size for the main dimension is applied to all objects. It changes automatically as soon as the font size for the main dimension is changed.

The font size for the tolerances is entered in this field. The value is entered in the current line unit. Note that a unit can be specified immediately after the value, e.g. `10pt`.

If this check box is enabled, the font size specified for the tolerances is applied to all objects. It changes automatically as soon as the font size for the tolerances is changed.

The accuracy for measurement display is entered in this field. Values between 0 and 9 are allowed. The accuracy value determines the maximum number of decimal places if floating point decimals are in use (e.g. if "3" is entered, 1/3 is shown as `1.333` and 1,0002 is shown as `1.0`). With fractions the value shows the maximum negative power to which the fractions are shown (e.g. if "4" is entered, fractions up to 1/16 are shown accurately; 7/3 is shown as `2 5/16` and 7/2 is shown as `3 1/2`).

If this check box is enabled, the specified accuracy is applied to all objects. It changes automatically as soon as the accuracy is changed.

Clicking on this button calls the Dimension Parameters Options dialog, in which some system-wide dimension options can be specified. The changes made here affect *all* the dimensions in the drawing.

If this check box is enabled an extension line is drawn at the start of the dimension. The extension line begins at the start point of the dimension and extends about 1 mm over the dimension line.

If this check box is enabled the dimension line display setting is for the start of dimensions is applied to all objects. It changes automatically as soon as the dimension line display setting is changed.

If this check box is enabled an extension line is drawn at the end of the dimension line. The extension line begins at the end point of the dimension and runs about 1 mm over the end of the dimension line.

If this check box is enabled the extension line display settings for the end of the dimension line are applied to all objects.

If this check box is enabled, dimension lines, the arrows at their ends and (if present) extension lines are drawn. Otherwise, only the dimension itself is shown.

If this check box is enabled, the dimension line display setting is applied to all objects. It changes automatically as soon as the dimension line display settings are changed.

This icon represents "Full Length, Right Angled" dimension line mode. This means that all dimension lines are full length and that arrows and extension lines can be shown at both ends. The extension lines are at right angles to the dimension line.

This icon represents "Full Length, Random" dimension line mode. This means that are dimension lines are full length and that arrows and extension lines can be shown at both ends. The angle between the dimension line and the extension line is determined by the dimension line's position details.

This mode can only be recommended, if the dimension line is displayed parallel to the dimension. In all other cases, sloping extension lines lead to visually dreadful results.

This icon represents "Partial Length, Right Angled" dimension line mode. This means that the dimension line does not extend the full length and that an arrow and extension line can only be shown at one end (the start of the dimension line). The extension line is always at right angles to the dimension line.

If this check box is enabled, the dimension line mode setting is applied to all objects. It changes automatically, as soon as the dimension line mode is changed.

This icon represents "Parallel" dimension line direction mode. The dimension line runs in the same direction as the length being measured. The dimension line therefore runs parallel to the length being measured.

If a dimension which is parallel is to be repositioned, it should not be done by changing this setting, but by using the command Edit Dimension Line>Rotate.

This icon represents "Horizontal" dimension direction mode. This means that the dimension is calculated horizontally. This means that the dimension line also runs horizontally. If the dimension is rotated afterwards, the dimension direction is also rotated, i.e. does not necessarily remain horizontal.

If a dimension which is parallel is to be repositioned, it should not be done by changing this setting, but by using the command Edit Dimension Line>Rotate.

This icon represents "Vertical" dimension direction mode. This means that the dimension direction is calculated vertically. This means that the dimension line runs vertically. If it is rotated afterwards, the dimension direction is rotated accordingly and does not necessarily remain vertical.

If a dimension which is parallel is to be repositioned, it should not be done by changing this setting, but by using the command Edit Dimension Line>Rotate.

This icon represents "Any" dimension direction mode. This means that the dimension can be calculated in any direction and the dimension line will then run in that direction. This direction is determined by entering a point. If the dimension is rotated later, then the dimension direction is rotated correspondingly

If a dimension which is parallel is to be repositioned, it should not be done by changing this setting, but by using the command Edit Dimension Line>Rotate.

If this check box is enabled, the dimension direction mode setting is applied to all objects. It changes automatically as soon as the dimension direction mode is changed.

This icon represents "Any" dimension line spacing mode. This means that any value can be entered for the spacing between the object and the dimension line.

If a dimension line which is currently a fixed or multiple distance from the object is to be repositioned, it should not be done by changing this setting, but by using the command Edit Dimension Line>Position.

This icon represents "Fixed Distance" dimension line spacing mode. This means that the dimension line's distance from the measurement object should be entered in the lower field. When the measurement is made, the only thing which can be specified is which side of the object the dimension line should be placed on.

If a dimension line which is currently a fixed or multiple distance from the object is to be repositioned, it should not be done by changing this setting, but by using the command Edit Dimension Line>Position.

This icon represents "Multiple Spacing" dimension line spacing mode. This means that the distance between the measurement object and the dimension line is always a multiple of the value entered in the lower field. When carrying out the measurement, only the factor to be used and the side on which to place the dimension line can be chosen.

If a dimension line which is currently a fixed or multiple distance from the measurement object is to be repositioned, it should not be done by changing this setting, but by using the command Edit Dimension Line>Position.

If this check box is enabled, the dimension line spacing mode is applied to all objects. It changes automatically, as soon as the dimension line spacing mode is changed.

The dimension line spacing to be used is entered in this field. The value is entered in the current line unit. The dimension line spacing is affected by the dimension line spacing mode

If this check box is enabled, the dimension line spacing setting is applied to all objects. It changes automatically, as soon as the dimension line spacing is changed.

If this check box is enabled, the dimension is automatically updated as soon as either the measurement object or the underlying coordinate system are changed.

If dimensions are to be updated when automatic updating is not enabled, the command Edit Dimension>Update can be used.

If this check box is enabled, the setting for updating dimensions is applied to all objects. It changes automatically as soon as the update setting is altered

If this check box is enabled the dimension is placed at the center of the dimension line. The only thing that can be specified is its distance from the dimension line.

Repositioning a centered dimension should not be done by changing this setting, but by using the command Edit Dimension>Position.

If this check box is enabled the dimension centering setting is applied to all objects. It changes automatically as soon as the dimension centering setting is changed.

If this check box is enabled the dimension is automatically placed close to the dimension line. Its position on the dimension line can still be determined.

If a closely positioned dimension is to be repositioned , it should not be done by changing this setting, but by using the command Edit Dimension>Position.

If this check box is enabled, the setting for the distance between dimensions and objects is applied to all objects. It changes automatically as soon as the dimension is changed.

If this check box is enabled, the dimension can be shown rotated. The angle of rotation to be used is determined by entering a point. Otherwise, the dimension runs parallel to the dimension line.

If a dimension which is not currently rotated is to be rotated, it should not be done by changing this setting, but by using the command Edit Dimension>Rotate.

If this check box is enabled, the dimension rotation setting is applied to all objects. It changes automatically, as soon as the dimension's rotation is changed.

If this check box is enabled a dialog appears after the dimension has been entered, where extra dimension text and the tolerances can be edited.
The dimension text and tolerances of an existing dimension can be viewed and changed using Edit Text.

Clicking on this button calls the General Dimension Parameters dialog, in which further dimension parameters can be specified. The changes made in this dialog affect all subsequently generated dimensions.

The color is displayed in its dithered variant in this color field. This means that it is visible, if no palette is used for output.

The color is displayed in its solid variant in this color field. This means that it is visible if a palette is used for output.

The red portion (in RGB mode) or cyan portion (in CMY mode) is entered as a percentage in this field. The mouse can be used to change the value by clicking on the arrow. The change can then be seen in the adjacent color field.

The green portion (in RGB mode) or magenta portion (in CMY mode) is entered as a percentage in this field. The mouse can be used to change the value by clicking on the arrow. The change can then be seen in the adjacent color field.

The blue portion (in RGB mode) or yellow portion (in CMY mode) is entered as a percentage in this field. The mouse can be used to change the value by clicking on the arrow. The change can then be seen in the adjacent color field.

The hue portion is entered as an angle in this field. The mouse can be used to change the value by clicking on the arrow. The change can then be seen in the adjacent color field.

The saturation portion is entered as a percentage in this field. The mouse can be used to change the value by clicking on the arrow. The change can then be seen in the adjacent color field.

The lumination portion is entered as a percentage in this field. The mouse can be used to change the value by clicking on the arrow. The change can then be seen in the adjacent color field.

If this radio button is enabled, the above values are interpreted as the color proportions of an RGB color. This is the usual way of displaying color as far as computers are concerned.

If this radio button is enabled, the above values are interpreted as the color proportions of a CMY color. This is the usual way of mixing colors in printing.

These color fields contain 32 standard colors. They can be changed by clicking in the color field, which avoids the difficult process of entering the color proportions as figures.

This list contains the names of all currently defined colors. If a color which is already defined is specified in the field above, the appropriate entry is specified.

If this check box is enabled, the colors in the list will be sorted based on their names. Else, they will be sorted based on their color values.

A new color can be defined with this button. It is defined using the current color setting.

With this button, the currently selected color in the list can be redefined using the current color setting. Colors whose name begins with a '*' cannot be redefined.

With this button, the currently selected color in the list can be renamed. A dialog appears, in which the name can be edited. Colors whose name begins with a '*' cannot be changed.

With this button, the color selected from the list can be deleted. Colors whose name begins with a '*' cannot be deleted.

This text shows the path of the directory whose contents are shown in the file list. This path is always shown in front of the file name in the "Name" field.

The name of the file to be used is entered in this field. If there is a file in the list whose name begins with the character entered, this file is put at the top of the list.

The extension for the files to be displayed can be selected from this list. There are usually only two or three file extensions available. If a standard ending (e.g. `.T4G` or `.T4L`) is chosen, it is forced. This means that the chosen ending is applied to all files.
For example, if a drawing file called `MYFILE` is entered, the extension `.T4G` will be used, the resulting file will be called `MYFILE.T4G`.

The ending specified here does not show the file format to be read or written. For example, if the command Save Drawing As has been used to select the ending `.*` the drawing will still be saved in TVG 4.0 format, but with the selected extension.

All the files in the current directory with the specified extension are shown in this list. Clicking once on a file name puts that name in the "Name" field, where it can be changed. Double-clicking on a file name selects that file and closes the dialog as if the "OK" button had been clicked on

All the sub-directories in the current directory are shown in this list. Double-clicking on a directory name will open or close that directory.

All available disk drives are shown in this list. Selecting a disk drive name makes that drive the active drive, and the path shows current root directory of that drive.

Clicking on this button displays brief information about the file whose name is in the "Name" field. If the program recognizes the file format, the file details are shown (see also the View Details dialog). Otherwise, only the name, size and creation date are shown (see also the File Info dialog).

Clicking on this button displays a list of all currently defined standard paths for selection. See Options>Paths.

By clicking on this button you can edit the options for the current file operation. See Choose T4G Elements and Output Settings.

Clicking on this button accepts the current file name. If a standard ending has been chosen from the list of extensions (e.g. `.T4G` or `.T4L`) this is forced. This means that the file will use the selected extension, whichever extension is entered.
The program checks whether the file already exists. If the file to be loaded does not exist, then an error message appears and the dialog remains open. If a file with the same name as the one to be saved already exists, a message appears and you can choose whether or not to overwrite the existing file.

Clicking on this button will close the dialog, without loading or saving a file.

These color fields contain 32 standard colors. A color can be chosen by simply clicking in a color field.

This list shows the names of the 32 standard colors. A color can be chosen by simply clicking in the list.

This text shows the current printer.

The number of copies to be made - between 1 and 250 - is shown in this field. If a multi-page document is printed, all copies of one page are printed together, as this is quicker.

The scaling for the printed output can be entered here. All the drawing objects are scaled by this factor. This also affects the line widths and the length of dashes in lines. Dimensions are not however changed.

The text for a header to be printed on each page can be entered in this field. Control characters can be used within the text to print variable data. Control characters always begin with the percent character '%'. To print a single percentage character, two percent symbols must be entered one after another. The following control characters are available:

%Z	or	%z	Drawing File Name
%P	or	%p	Page Number (if printing on several pages)
%F	or	%f	Current Scaling Factor
%D	or	%d	Current Date
%T	or	%t	Current Time
%A	or	%a	Name of the Author
%N	or	%n	Name of the Change Author

The form in which the date and time are output can be changed with the command Settings>Time Format.

The horizontal position for the header can be specified in this list. It is always in the upper line of the page's printable area.

The text for a footer to be printed on each page can be entered in this field. Control characters can be used within the text to print variable data. Control characters always begin with the percent character '%'. To print a single percentage character, two percent symbols must be entered one after another. The following control characters are available::

%Z	or	%z	Drawing File Name
%P	or	%p	Page Number (if printing on several pages)
%F	or	%f	Current Scaling Factor
%D	or	%d	Current Date
%T	or	%t	Current Time
%A	or	%a	Name of the Author
%N	or	%n	Name of the Change Author

The form in which the date and time are output can be changed with the command Settings>Time Format.

The horizontal position for the footer can be specified in this list. It is always in the lower line of the page's printable area.

The distance of the upper margin of the printable area from the edge of the page can be entered here. The value is entered in the current line unit.
Most printers cannot print in an area between 5 and 20 mm from the edge of the page. If a smaller value is entered here, then the smallest possible margin is used.

The distance of the lower margin of the printable area from the edge of the page can be entered here. The value is entered in the current line unit.
Most printers cannot print in an area between 5 and 20 mm from the edge of the page. If a smaller value is entered here, then the smallest possible margin is used.

The distance of the left margin of the printable area from the edge of the page can be entered here. The value is entered in the current line unit.
Most printers cannot print in an area between 5 and 20 mm from the edge of the page. If a smaller value is entered here, then the smallest possible margin is used.

The distance of the right margin of the printable area from the edge of the page can be entered here. The value is entered in the current line unit.
Most printers cannot print in an area between 5 and 20 mm from the edge of the page. If a smaller value is entered here, then the smallest possible margin is used.

If this check box is enabled, crop marks to make it easier to cut the pages are printed in each corner of the page. This is particularly important if a drawing is printed on several sheets which are then cut up and aligned.

The options for how many pages to print on are chosen from this list. There are three options:

"Single Page"
: The drawing is printed on a single page. If the drawing is larger than the page, then it is chopped off at the edges. The center of the current drawing is printed at the center of the page.

"Multiple Pages"
: The drawing is printed on multiple pages. As many pages as are needed for the whole drawing are printed. The print-outs overlap by one printer point. Empty pages within the drawing are output as well.

"Fit to Page"
: The drawing is printed on one page, scaled proportionally to fit the page. The print-out is automatically centered on the page. The scaling also affects the line width and the length of the dashes in dashed lines, but not the dimension values.

If this check box is enabled the print-out is centered with respect to all the pages. Otherwise its position is not changed.

If this check box is enabled, only those objects and instances which are currently selected are printed.

If this check box is enabled, the print-out is in draft mode. The exact effect depends on the printer and driver used. In general, draft mode delivers a poorer quality print out at a higher speed.

This icon represents a portrait page. If the adjacent check box is enabled, portrait pages are used for printing. This setting can be changed independent of the current drawing's page format. This can reduce the number of pages needed when printing on multiple pages.

This icon represents a landscape page. If the adjacent check box is enabled, the print-out is done on landscape pages. This setting can be changed independent of the current drawing's page format. This can reduce the number of pages needed when printing on multiple pages.

A rough preview of the printed result is shown in this area. The colored area represents the objects to be printed or the drawing. The rectangles show the number of pages required. All changes to the settings are reflected in this preview.

The number of pages needed for the print-out are shown in this area. Empty pages are included in the calculation.

Clicking on this button calls a dialog, in which the selected printer's setup can be viewed and changed. Also, a different printer can be selected.
The functionality of this dialog depends on the selected printer. Note that the page orientation, print quality and margin settings are over ridden by TommySoftware® CAD/DRAW.

This button can be used to call a dialog, in which output options can be specified. The following options are available:

- Output Settings
- Plotter Settings

This button calls the Choose T4G Elements dialog where the options of the currently selected file operation can be altered.

This button resets the options of all file operations to their standard values.

In this list, the file operation whose options are to be edited is selected. The following file operations are available:

"Open Drawing"
Opening of a drawing or a duplicate.

"Save Drawing"
Saving of a drawing or duplicate, and autosave.

"TVG 4.0 Import"
Import of a drawing.

"TVG 4.0 Export"
Export of a drawing.

"New Drawing"
Creation of a new drawing, i.e. reading of the standard elements from `RELEASE4.T4G` or the user-dependent element file.

"Save Standard Elements"
Saving of the standard elements to `RELEASE4.T4G` or the user-dependent element file.

"Open Settings"
Opening of a setting file, also loading of the standard settings from `RELEASE4.INF` or the user-dependent setting file.

"Save Settings"
Saving of a setting file, also saving of the standard settings to `RELEASE4.INF` or the user-dependent setting file.

If this radio button is selected, the elements affected by this setting are added. This means that the new elements are added to the existing ones, which remain unchanged.
This almost always results in a renumbering of the new elements. However, their names remain the same.

If this radio button is selected, then the elements which are affected by this setting are replaced. This means the existing elements are deleted and replaced by the new ones.

This button restores the standard options of the current file operation.

This button stores the selected options as standard options for the current file operation.

This list displays all available TVG 4.0 file elements. Each line of the list represents one element which can be switched on and off by clicking. If an item is selected the corresponding file element will be loaded or saved, respectively.

See TVG 4.0 File Elements and File Format for details.

If this radio button is selected, the drawing will be saved as a duplicate. A file with the same name as the current drawing, but with the extension `.DUP`.

If this radio button is selected, the drawing is saved to the current drawing file. Be careful when using this command, as it is very easy to overwrite existing files.

The interval between autosaves can be entered in this field. The time is entered in minutes. Times between 1 and 60 minutes are allowed.

If this check box is enabled, autosave is on. The other settings are only effective when autosave is on.

The file name of the bitmap is displayed in this area.

If this check box is enabled, the specified bitmap file name is used for all bitmap objects. It is automatically reset as soon as "Select" button is used to choose a bitmap file.

Some information on the bitmap is displayed in this area. The information is not always available.

The value to be used for the matrix element M11 can be entered in this field. Together with the elements M12, M21 and M22 this determines the size, direction and rotation of the bitmap which the bitmap object refers to.

If this check box is enabled the specified matrix element M11 is used for the display matrix of all bitmap objects. It is automatically reset as soon as the matrix element M11 is changed.

The value to be used for the matrix element M12 can be entered in this field. Together with the elements M11, M21 and M22 this determines the size, direction and rotation of the bitmap which the bitmap object refers to.

If this check box is enabled the specified matrix element M12 is used for the display matrix of all bitmap objects. It is automatically reset as soon as the matrix element M12 is changed.

The value to be used for the matrix element M21 can be entered in this field. Together with the elements M11, M12 and M22 this determines the size, direction and rotation of the bitmap which the bitmap object refers to.

If this check box is enabled the specified matrix element M21 is used for the display matrix of all bitmap objects. It is automatically reset as soon as the matrix element M21 is changed.

The value to be used for the matrix element M22 can be entered in this field. Together with the elements M11, M12 and M21 this determines the size, direction and rotation of the bitmap which the bitmap object refers to.

If this check box is enabled the specified matrix element M22 is used for the display matrix of all bitmap objects. It is automatically reset as soon as the matrix element M22 is changed.

A value for the matrix element M31 can be entered here. This determines the X coordinates for the position of the bitmap's insertion point. The value entered is in the current length unit.

If this check box is enabled the specified matrix element M31 is used for the display matrices of all bitmap objects. It is automatically reset as soon as the matrix element M31 is altered.

A value for the matrix element M32 can be entered here. This determines the Y coordinates for the position of the bitmap's insertion point. The value entered is in the current length unit.

If this check box is enabled the specified matrix element M32 is used for the display matrices of all bitmap objects. It is automatically reset as soon as the matrix element M32 is altered.

Clicking on this button calls the File Selection dialog, where the bitmap file name can be chosen.

The width and height of the resulting bitmap in pixels are shown here.

The resolution of the resulting bitmap in dpi (points per inch) is shown here.

The memory requirement for the resulting bitmap is shown in this area (in KB). The memory requirement depends on the height, width and color depth. Bitmaps are stored uncompressed.

If this radio button is selected, the bitmap's size is determined by the given resolution.

If this radio button is selected, the bitmap's size is determined by the given height and width. The drawing will be scaled proportional to fit into that size.

If this radio button is selected, the bitmap's size is determined by the given height and width. The drawing will be scaled non-proportional to fit into that size.

The horizontal resolution to be used when creating the bitmap is entered here. The value is entered in the current unit (points per millimeter or points per inch).

The vertical resolution to be used when creating the bitmap is entered here. The value is entered in the current unit (points per millimeter or points per inch).

The width in Pixels be used when creating the bitmap is entered here.

The height in Pixels to be used when creating the bitmap is entered here.

If this radio button is selected, the adjacent resolutions are shown in points per inch (2.54 cm). This is the usual method of displaying the resolution of bitmaps.

If this radio button is selected, the adjacent resolutions are shown in points per millimeter.

The color depth for the bitmap is chosen from this list. The color depth is shown in bits per pixel. The following color depths can be chosen.

"1 Bit (2 Colors)"
: Monochrome bitmap.

"4 Bit (16 Colors)"
: 16 color bitmap. The first 16 colors from the Windows color palette are used.

"8 Bit (256 Colors)"
: 256 color bitmap. Either the standard colors from the Windows palette or a special 256 color rainbow palette, containing all color combinations in 20% steps, can be used.

"24 Bit (True Colors)"
: True color bitmap.

If this check box is enabled (only possible if 8 bit color depth is chosen), the colors are shown as undithered colors from a rainbow palette. This gives a "shaper" appearance, but the colors can not be shown exactly.

Clicking on this button calls the Output Settings dialog, in which the options for bitmap export can be specified.

The color for the comment arrow can be specified from this color field. Clicking in this field with the left mouse button calls the Choose Standard Color dialog, where the correct color can be chosen.

The background color for comment fields can be specified using this color field. Clicking in this field with the left mouse button calls the Choose Standard Color dialog, where the correct color can be chosen.

The frame color for comment fields can be specified from this dialog. Clicking in this field with the left mouse button calls the Choose Standard Color dialog, where the correct color can be chosen.

The text color for comments can be specified using this color field. Clicking in this field with the left mouse button calls the Choose Standard Color dialog, where the correct color can be chosen.

The width (in pixels) of the comment field is specified in this field.

The width (in pixels) of the comment arrow is specified in this field. The size of the arrowhead is worked out relative to this width.

If this check box is enabled, comments are shown on screen. Comments are drawn in the same order as other objects; they can be covered by objects or other comments.

The name of the file whose details are currently displayed (and can possibly be changed) is shown in this area.

The size of the file in bytes whose details are currently displayed (and can possibly be changed) is shown in this area.

The title of the file is shown in this area and can possibly be changed. The title can be up to 63 characters long.

This title serves to identify library and font files. If the title of a library is changed, all existing references to this library (and instances of blocks from it) become invalid. They may have to be updated using the command Replace Block Name oder Replace Library Name.

The theme of the file is shown in this area and can possibly be changed. It can be up to 63 characters long.

The name of the file's author is shown in this area. The name which appears here is that of the registered user who created the file.

The time at which the file was created is shown in this area. The format can be specified using the command Settings>Time Format.

The name of the change author is shown in this area and can possibly be changed. It can be up to 63 characters long. Initially the name of the registered user is shown here.

The time when the file was last changed is shown in this area. The format can be changed using the command Settings>Time Format.

A comment about the drawing is shown in this area and can possibly be changed. It can be up to 255 characters long.

The page format is chosen from this list. Several US and DIN / ISO formats are available.
Custom sizes can be added by entering the dimensions in the relevant fields.

This icon represents "Custom" page format. It is automatically activated if the values in the "Width" and "Height" fields are changed.

This icon represents "Portrait" page format. If the adjacent check box is enabled, a page of the size specified above in portrait format is used.

This icon represents "Landscape" page format. If the adjacent check box is enabled, a page of the size specified above in landscape format is used.

The page width can be entered in this field. The value is entered in the length unit. Note that a unit can be specified immediately after the value, e.g. `8.5in`.
If the value in this field is changed, then the page format and the page orientation are both set to "custom".

The height of the page can be entered in this field. The value is entered in the length unit. Note that a unit can be specified immediately after the value, e.g. `14in`.
If the value in this field is changed, then the page format and the page orientation are both set to "custom".

If this check box is enabled, layer transmission settings are handled identical for screen display and printer or plotter output.

The layer to be changed is chosen from this list. Any currently defined layer can be chosen. Several layers can be selected by holding down the CONTROL key while selecting layers. If the SHIFT key is held down, then areas can be selected.

Double-clicking a layer exits the dialog making the selected layer the active layer. All modifications will be maintained.

The icons show the current layer status of each layer.

If the currently selected layer transmits any properties these properties are shown here in text form. The transmission information can be changed by clicking on the "Output" / "Screen" button or the "Properties" button respectively. If several layers are selected nothing is displayed.

Clicking on this button exits the dialog box making the currently chosen layer the active layer. All modifications will be maintained.

A new layer can be defined by clicking on this button. A dialog appears in which the name of the new layer is entered.
The new layer initially uses the same layer settings as the currently selected layer in the list. Therefore, a layer must be selected when a new one is defined.

Clicking on this button calls the Edit Properties dialog, in which the current layer settings for screen display can be changed.

Clicking on this button calls the Edit Properties dialog, in which the current layer settings for printer or plotter output can be changed.

Clicking on this button calls a dialog in which the name of the selected layer can be edited. The name of the "*Standard" layer cannot be changed.

Clicking on this button deletes the selected layer. This action has to be confirmed. The "*Standard" Layer cannot be deleted.

Clicking on this button calls the Layer Selection dialog, which you for example can use to select layers containing certain keywords.

After clicking this button a small menu appears where you can choose between importing a new layer status or exporting the current layer status. Afterwards a File Selection Dialog appears where the name of the drawing file can be chosen.

This function allows you to manage and quickly access several layer status settings, e.g. one for screen display and another for output to printer or plotter. This is especially useful when dealing with complex drawings which require frequent changes of the layer status settings.

You can use this function to import layer settings from any drawing, even from those that contain different layer names. In such a case objects belonging to no longer existing layers would be set to the "*Standard" layer. But actually this function makes only sense when importing layer settings with identical layer names but different status settings.

Clicking on this button allows to export a textual description of all currently defined layers. A File Selection Dialog appears where the file name of the text file can be chosen.

If this check button is set, the layer currently selected in the list is displayed on screen, i.e. all objects and instances in that layer are displayed and can be selected.

If the check button is cleared, the layer is not displayed on screen, i.e. the objects and instances in that layer are not displayed and consequently can not be selected or changed.

Several layers can be selected at the same time. In this case the check button is initially displayed gray indicating that there is no change yet. If you set or clear the check box this change is applied to *all* currently selected layers.

If an instance lies in a hidden layer, the block to which that instance refers is *completely* invisible even if individual objects of the block lie in visible layers.
On the other hand, if an instance is in a visible layer, and some objects of the block to which the instance refers are in hidden layers, only those objects are invisible.

If this check button is set, the layer currently selected in the list will be output to printer or plotter, i.e. all objects and instances in that layer will be output.

If the check button is cleared, the layer will not be output to printer or plotter, i.e. the objects and instances in that layer will not be output.

Several layers can be selected at the same time. In this case the check button is initially displayed gray indicating that there is no change yet. If you set or clear the check box this change is applied to *all* currently selected layers.

If an instance lies in a non-output layer, the block to which that instance refers is *completely* invisible even if individual objects of the block lie in output layer layers.
On the other hand, if an instance is in an output layer, and some objects of the block to which the instance refers are in non-output layers, only those objects are invisible.

If this check button is set, the layer which is currently selected from the list is frozen, that is, the objects and instances in the layer can not be selected and consequently can not be changed.

If the check button is cleared, the layer is not frozen, that is the objects and instances in that layer can be selected and changed.

Several layers can be selected at the same time. In this case the check button is initially displayed gray indicating that there is no change yet. If you set or clear the check box this change is applied to *all* currently selected layers.

If this check button is set, the layer which is currently selected from the list is ignored, that is, the objects and instances in the layer are not used for snapping.

If the check button is cleared, the layer is not ignored, that is the objects and instances in that layer will be used for snapping.

Several layers can be selected at the same time. In this case the check button is initially displayed gray indicating that there is no change yet. If you set or clear the check box this change is applied to *all* currently selected layers.

If this check button is set, the layer which is currently selected from the list will be shaded, that is, the objects and instances in the layer are displayed in a gray color.

If the check button is cleared, the layer will not be shaded, that is, the objects and instances in that layer are displayed using their own colors.

Several layers can be selected at the same time. In this case the check button is initially displayed gray indicating that there is no change yet. If you set or clear the check box this change is applied to *all* currently selected layers.

If this check box is enabled, the specified pen is transmitted. It changes automatically, as soon as a pen is selected.

If this check box is enabled, the selected line pattern is transmitted. It changes automatically, as soon as a line pattern is selected.

If this check box is enabled, the selected line width is transmitted. It changes automatically, as soon as the line width is changed.

If this check box is enabled, the selected line color is transmitted. It changes automatically, as soon as the line color is changed.

If this check box is enabled, the specified fill mode is transmitted. It changes automatically, as soon as the fill mode is changed.

If this check box is enabled, the selected fill color is transmitted. It changes automatically, as soon as the fill color is changed.

The layer to which all markings generated from now on are to be assigned can be chosen from this list. As well as all currently defined layers, the entry "Current" can be chosen. If this is chosen, then the markings are assigned to the current layer.

The layer to which newly generated surfaces are to be assigned (see Trim Surface>Generate Surface) can be chosen from this list. As well as all currently defined layers, the entry "Current" can be chosen. If this is chosen, then the newly generated surfaces are assigned to the current layer.

The layer to which all newly generated dimensions are to be assigned can be chosen from this list. As well as all currently defined layers, the entry "Current" can be chosen. If this is chosen, then the dimensions are assigned to the current layer.

The layer to which all newly generated texts are to be assigned can be chosen from this list. As well as all currently defined layers, the entry "Current" can be chosen. If this is chosen, then texts are assigned to the current layer.

The layer to which hatchings generated from this point on are to be allocated can be chosen from this list. As well as all currently defined layers, the entry "Current" can be chosen. If this is chosen, then the hatchings are assigned to the current layer.

The layer to which all instances (references to blocks) generated from this point should be assigned can be chosen from this list. As well as all currently defined layers, the entry "Current" can be chosen. If this is chosen, then instances are assigned to the current layer.

The layer to which all geometry objects generated from this point should be assigned can be chosen from this list. As well as all currently defined layers, the entry "Current" can be chosen. If this is chosen, then instances are assigned to the current layer.

The coordinate system to be changed can be selected from this list. All currently defined coordinate systems are included in the list.

The current settings of the coordinate system selected from the list are shown in text form in this area.

Clicking on this button calls the Coordinate System View dialog in which the view settings for the selected coordinate system can be changed.

Clicking on this button calls the Coordinate System Origin dialog, in which the selected coordinate system's origin settings can be changed.

Clicking on this button calls the Coordinate System Units dialog, in which the unit settings of the current coordinate system can be changed.

Clicking on this button calls the Coordinate System Numbers dialog, in which the selected coordinate system's numbering can be changed.

Clicking on this button calls the Coordinate System Display Grid dialog, in which the selected coordinate system's display grid settings can be changed.

Clicking on this button calls the Coordinate System Position Grid dialog in which the selected coordinate system's position grid settings can be changed.

Clicking on this button exits the dialog box making the currently chosen coordinate system the active coordinate system. All modifications will be maintained.

A new coordinate system can be defined by clicking on this button. This calls a dialog in which the number and name of the new coordinate system are entered. The number used here cannot already be in use and must be between 1 and 40.
Initially, the new coordinate system uses the same settings as the one currently selected from the list. A coordinate system must therefore be selected when a new one is defined.

Clicking on this button calls a dialog in which the name of the selected coordinate system can be edited. The name of the coordinate system called "*Standard" cannot be changed.

Clicking on this button deletes the selected coordinate system. This command must be confirmed with the dialog which then appears. The coordinate system called "*Standard" cannot be deleted.

The coordinate display mode is chosen from this list. This determines whether the objects should be displayed "normally" or "distorted". These options are available:

"Cartesian"
: The objects and instances are shown undistorted. The aspect ratios and angles used on screen are the same as those used for output.

"Isometric, Left View"
: The objects and instances in the drawing are distorted (by 30° vertically and scaled horizontally by 1/cos(30°)) so that the left view of an isometric representation appears undistorted. Both the aspect ratios and the angles used on the screen differ from those used for output.

"Isometric, Right View"
: The objects and instances in the drawing are distorted (by -30° vertically and scaled horizontally by 1/cos(30°)) so that the right view of an isometric representation appears undistorted. Both the aspect ratios and the angles used on the screen differ from those used for output.

"Isometric, Plan View"
: The objects and instances in the drawing are distorted (rotated by 30°, distorted by 30° vertically and scaled horizontally by 1/cos(30°)) so that the plan view of an isometric representation appears undistorted. Both the aspect ratios and the angles used on the screen differ from those used for output.

"Dimetric 1, Left View"

"Dimetric 1, Right View"

"Dimetric 1, Plan View"
: Similar to isometric representation, but for dimetric representation with the axes -7° and 41.5°.

"Dimetric 2, Left View"

"Dimetric 2, Right View"

"Dimetric 2, Plan View"
: Similar to isometric representation, but for dimetric representation with the axes 7° and 41.5°.

This mode works in a similar way when calculating dimensions. Dimensions are calculated according to the representation mode of their reference coordinate systems. This means that correct dimensions can be calculated when working in isometric or dimetric view.

DIN standard 5 uses an angle of 42° instead of the 41.5° shown here. Using 42° does not give an even line grating, which you can easily check mathematically. An angle of 41.5° gives a *perfect* line grid.

The rotation angle for the coordinate system, in the angle unit, can be entered in this field.

Rotation of the coordinate system gives the rotation angle in a positive sense relative to the position of the page. Because the *coordinate system* is always displayed unrotated, with a rotation angle of 30° the screen display will be rotated by-30°.

This button sets the scale to 50:1.

This button sets the scale to 20:1.

This button sets the scale to 10:1.

This button sets the scale to 5:1.

This button sets the scale to 2:1.

This button sets the scale to 1:1.

This button sets the scale to 1:2.

This button sets the scale to 1:5.

This button sets the scale to 1:10.

This button sets the scale to 1:20.

This button sets the scale to 1:50.

The scale to be used can be entered in these fields. Neither value may be 0 or negative.

The position of the origin can be chosen from this list. As well as some positions which are always shown relative to the page, the entry "Defined Position" can be chosen, and the values in the fields will be used.

The X coordinate of the origin can be entered here. The value is entered in the length unit.
Changing the value in this field automatically changes the coordinate position entry to "defined position".

The Y coordinate of the origin can be entered here. The value is entered in the length unit.
Changing the value in this field automatically changes the coordinate position entry to "defined position".

The length unit to be used can be chosen from this list. Both the full description and the abbreviation for the unit are shown.
The length unit is used for all dimension-related input and output. These are coordinates, lengths, radii and surfaces. In particular it is used for the calculation and display of length-related dimensions.
When entering values, the current unit can be overridden by entering a different specific unit.

The line unit to be used can be chosen from this list. Both the full name and the abbreviation for the unit are shown.
The line unit is used for all inputs and outputs which are not dependent on the scale. Above all, these are line widths and text sizes, but also the origin position or dimension line spacing.
When entering values, the current unit can be overridden by entering a different specific unit.

The angle unit to be used can be chosen from this list. Both the full name and the abbreviation of each unit are shown.
The angle unit is used for all angle input and output, including those implied in a term (e.g. `sin(45)`).
When entering values, the current unit can be overridden by entering a different specific unit.

The form in which numeric values for length measurements are to be shown is chosen from this list. There are numerous possibilities:

"Standard (Floating Point)"
: The numbers are shown as usual with a floating decimal separator and decimal fractions, e.g. 10.75. Zeroes at the end of the number are suppressed down to one 0 after the separator.

Standard (Fraction)
: Numbers are displayed as a mixed fraction, e.g. 10 3/4 (10.75). The fraction is reduced automatically.

Foot Inch (Floating Point)
: Numbers are given in feet and inches, which means that the origin must be present in inches. The number of feet is always a whole number and the number of inches is given as a decimal fraction. 170.75 is therefore output as 14'2.75". Zeroes at the end of the number are suppressed down to one 0 after the separator.

"Foot Inch (Fraction)"
: Numbers are given in feet and inches, which means that the origin must be present in inches. The number of feet is always a whole number and the number of inches is given as a mixed fraction. 170.75 is therefore output as 14'2 3/4". The fraction is reduced automatically.

"Yard Foot Inch (Floating Point)"
: Numbers are given in yards, feet and inches, which means that the origin must be present in inches. The number of yards and feet is always a whole number and the number of inches is given as a decimal fraction. 170.75 is therefore output as 4yd2'2.75". Zeroes at the end of the number are suppressed down to one 0 after the separator.

"Yard Foot Inch (Fraction)"
: Numbers are given in yards, feet and inches, which means that the origin must be present in inches. The number of yards and feet is always a whole number and the number of inches is given as a mixed fraction. 170.75 is therefore output as 4yd2'2 3/4". The fraction is reduced automatically.

"Degree Minute Second (Floating Point)"
: Numbers are given in degrees, minutes and seconds, which means that the origin must be present in degrees. The number of degrees and minutes is always a whole number and the number of seconds is given as a decimal fraction. 1.372 is therefore output as 1°22'19.2". Zeroes at the end of the number are suppressed down to one 0 after the separator.

"Degree Minute Second (Fraction)"
: Numbers are given in degrees, minutes and seconds, which means that the origin must be present in degrees. The number of degrees and minutes is always a whole number and the number of seconds is given as a mixed fraction. 1.372 is therefore output as 1°22'19 3/16". The fraction is reduced automatically.

"Architectural (Floating Point)"
: Numbers are given as usual. If the numeric value is below 1.0, it will be multiplied by 100 before output. For this mode, the origin should be present in meters.

In edit field, numbers will to be displayed in a different form understood by the program. In "Yard Foot Inch (Fraction)" mode, the value 170.75 will be displayed as (4yd+2ft+(2+3/4)in).

The form in which numeric values for angle measurements are to be shown is chosen from this list. There are numerous possibilities:

"Standard (Floating Point)"
: The numbers are shown as usual with a floating decimal separator and decimal fractions, e.g. 10.75. Zeroes at the end of the number are suppressed down to one 0 after the separator.

"Standard (Fraction)"
: Numbers are displayed as a mixed fraction, e.g. 10 3/4 (10.75). The fraction is reduced automatically.

"Foot Inch (Floating Point)"
: Numbers are given in feet and inches, which means that the origin must be present in inches. The number of feet is always a whole number and the number of inches is given as a decimal fraction. 170.75 is therefore output as 14'2.75". Zeroes at the end of the number are suppressed down to one 0 after the separator.

"Foot Inch (Fraction)"
: Numbers are given in feet and inches, which means that the origin must be present in inches. The number of feet is always a whole number and the number of inches is given as a mixed fraction. 170.75 is therefore output as 14'2 3/4". The fraction is reduced automatically.

"Yard Foot Inch (Floating Point)"
: Numbers are given in yards, feet and inches, which means that the origin must be present in inches. The number of yards and feet is always a whole number and the number of inches is given as a decimal fraction. 170.75 is therefore output as 4yd2'2.75". Zeroes at the end of the number are suppressed down to one 0 after the separator.

"Yard Foot Inch (Fraction)"
: Numbers are given in yards, feet and inches, which means that the origin must be present in inches. The number of yards and feet is always a whole number and the number of inches is given as a mixed fraction. 170.75 is therefore output as 4yd2'2 3/4". The fraction is reduced automatically.

"Degree Minute Second (Floating Point)"
: Numbers are given in degrees, minutes and seconds, which means that the origin must be present in degrees. The number of degrees and minutes is always a whole number and the number of seconds is given as a decimal fraction. 1.372 is therefore output as 1°22'19.2". Zeroes at the end of the number are suppressed down to one 0 after the separator.

"Degree Minute Second (Fraction)"
: Numbers are given in degrees, minutes and seconds, which means that the origin must be present in degrees. The number of degrees and minutes is always a whole number and the number of seconds is given as a mixed fraction. 1.372 is therefore output as 1°22'19 3/16". The fraction is reduced automatically.

"Architectural (Floating Point)"
: Numbers are given as usual. If the numeric value is below 1.0, it will be multiplied by 100 before output. For this mode, the origin should be present in meters.

In edit field, numbers will to be displayed in a different form understood by the program. In "Yard Foot Inch (Fraction)" mode, the value 170.75 will be displayed as (4yd+2ft+(2+3/4)in).

The accuracy to be used for output can be entered in this field. Values between 0 and 9 are allowed.
This value shows the maximum number of decimal places to be shown if floating point display is in use (e.g. 3 would mean that 1/3 is output as 1.333 , 1.0002 as 1.0).
With fractional display it shows the maximum power to which the fraction should be shown (e.g. 4 means that fractions up to 1/16 are output precisely, i.e. 7/3 is shown as 2 5/16, and 7/2 is shown as 3 1/2).

This icon represents "No Grid" grid type. If the adjacent check box is enabled, the display grid will be inactive regardless of the spacing specified.

This icon represents "Cartesian" grid type. If the adjacent check box is enabled, a cartesian display grid is used. The horizontal and vertical grid point spacing is as entered below.

This icon represents "Isometric" grid type. If the adjacent check box is enabled, an isometric display grid is used. The "horizontal" spacing is shown on the right axis, and the "vertical" spacing on the left axis. If this grid type is active, two straight lines are added to the cross hair which shows the isometric view axes (30° and -30°) It helps with orientation.

This icon represents "Dimetric 1" grid type. If the adjacent check box is enabled, a dimetric display grid is used. The "horizontal" spacing is shown on the left axis, the "vertical" spacing on the right axis.
If this grid type is active, two straight lines which show the dimetric axes (-7° and 41.5°). They help orientation.

This icon represents "Dimetric 2" grid type. If the adjacent check box is enabled, a dimetric display grid is used. The "horizontal" spacing is shown on the right axis, and the "vertical" spacing on the left axis. If this grid type is active, two straight lines which show the dimetric axes (7° and -41.5°). They help orientation.

The horizontal spacing between grid points can be entered in this field. The value is entered in the length unit.

This button sets the horizontal grid point spacing to 0.0 units, this means that the horizontal grid is no longer active.

This button sets the horizontal grid point spacing to 0.01 units.

This button sets the horizontal grid point spacing to 0.05 units.

This button sets the horizontal grid point spacing to 0.1 units.

This button sets the horizontal grid point spacing to 0.25 units.

This button sets the horizontal grid point spacing to 0.5 units.

This button sets the horizontal grid point spacing to 1,0 units.

This button sets the horizontal grid point spacing to 2,0 units.

This button sets the horizontal grid point spacing to 2,5 units.

This button sets the horizontal grid point spacing to 5,0 units.

This button sets the horizontal grid point spacing to 10.0 units.

The vertical grid point spacing can be entered in this field. The value is entered in the length unit.

This button sets the vertical grid point spacing to 0.0 units, this means that the vertical grid is no longer active.

This button sets the vertical grid point spacing to 0.01 units.

This button sets the vertical grid point spacing to 0.05 units.

This button sets the vertical grid point spacing to 0.1 units.

This button sets the vertical grid point spacing to 0.25 units.

This button sets the vertical grid point spacing to 0.5 units.

This button sets the vertical grid point spacing to 1,0 units.

This button sets the vertical grid point spacing to 2,0 units.

This button sets the vertical grid point spacing to 2,5 units.

This button sets the vertical grid point spacing to 5,0 units.

This button sets the vertical grid point spacing to 10.0 units.

This icon represents "No Grid" grid type. If the adjacent check box is enabled, the position grid is not active, independent of the specified spacing.

This icon represents "Cartesian" grid type. If the adjacent check box is enabled, a cartesian position grid is used. The horizontal and vertical grid point spacing is as entered below.

This icon represents "Isometric" grid type. If the adjacent check box is enabled, an isometric position grid is used. The "horizontal" spacing is shown on the right axis, and the "vertical" spacing on the left axis.

This icon represents "Dimetric 1" grid type. If the adjacent check box is enabled, a dimetric grid is used for the position grid. The "horizontal" spacing is shown on the left axis, the "vertical" spacing on the right axis.

This icon represents "Dimetric 2" grid type. If the adjacent check box is enabled, a dimetric grid is used for the position grid. The "horizontal" spacing is shown on the right axis, and the "vertical" spacing on the left axis.

If this check box is enabled, pen settings are handled identical for screen display and printer or plotter output.

The pen to be changed can be chosen from this list. All the currently defined pens are included in the list. Several pens can be selected by holding down the CONTROL key while selecting pens. If the SHIFT key is held down, then areas can be selected.

Double-clicking a pen exits the dialog making the selected pen the active pen. All modifications will be maintained.

The current settings of the selected pen are shown in this area.

Clicking on this button calls the Edit Properties dialog in which the pen output properties (printer, plotter, clipboard, metafile and bitmap) can be specified.

Clicking on this button calls the Edit Properties dialog, in which the screen display properties for the pen can be specified.

Clicking on this button, calls a dialog in which a layer can be selected. This layer is then set as the current layer if the pen is chosen.

Clicking on this button exits the dialog box making the currently chosen pen the active pen. All modifications will be maintained.

A new pen can be defined by clicking on this button. This calls a dialog in which the name of the new pen is entered.
The new pen initially has the same properties as the pen selected from the list. Therefore, a pen must be selected before defining a new one.

Clicking on this button calls a dialog in which the name of the selected pen can be edited. The name of the pen "*Standard" cannot be changed.

Clicking on this button deletes the selected pen. This command must be confirmed with the dialog which then appears. The pen named "*Standard" cannot be deleted.

Clicking on this button allows to export a textual description of all currently defined pens. A File Selection Dialog appears where the file name of the text file can be chosen.

The pen to be used from this point for dimension lines can be chosen from this list. As well as all the currently defined pens, the entry "*Current" is offered. If this is selected, then whichever pen is selected at that time is used for dimension lines.

The pen to be used from now on for all dimension texts can be selected from this list. As well as all the currently defined pens, the entry "*Current" is offered. If it is selected, whichever pen is currently selected is used.

The pen to be used for all text objects created from this point. As well as all the currently defined pens, the entry "*Current" is offered. If it is selected, whichever pen is currently selected is used.

The pen to be used for the reference lines of all reference texts created from this point. As well as all the currently defined pens, the entry "*Current" is offered. If it is selected, whichever pen is currently selected is used.

The pen to be used for all geometry objects created from this point. As well as all the currently defined pens, the entry "*Current" is offered. If it is selected, whichever pen is currently selected is used.

The line pattern to be changed can be selected from this list. All currently defined line patterns are included in the list.

An example of the current line pattern is displayed here.

A new line pattern can be defined by clicking on this button. This calls a dialog in which the name of the new line pattern is entered.
The new line pattern initially uses the same settings as the line pattern currently selected from the list. Therefore a line pattern must be selected when a new one is defined.

Clicking on this button calls the Edit Line Pattern dialog in which the currently selected line pattern can be edited.

Clicking on this button calls a dialog in which the name of the selected line pattern can be edited. The name of the "*Standard" line pattern cannot be changed.

Clicking on this button deletes the selected line pattern. This command must be confirmed with the dialog which then appears. The "*Standard" line pattern cannot be deleted.

The current line pattern is shown in this field. The display adapts to all the changes made in this dialog.

The dash lengths of the current line pattern are shown in this field.

The new or changed dash length is entered in this field. The value entered here must be between 100 and 10000.

If this radio button is selected, the dash lengths are given in 1/100 of the line width i.e. dependent on the line width. This is, for example, required by line patterns according to DIN 15.

If this radio button is selected, the dash lengths are given in 1/100 mm units. This means that they are independent of the line width in use.

The current number of dash pairs is shown in this area. If the number is odd, three plus symbols ("+++") appear after the number. They show that the last value was used, but has not (yet) been used. When another value is entered so that the number of dashes is even again, all the values are used.

Clicking on this button adds the value in the field to the end of the list of dash lengths. Up to 16 line lengths per line pattern can be entered.

Clicking on this button inserts the value from the field into the list before the currently selected entry in the list of dash lengths. Up to 16 line lengths per line pattern can be entered.

Clicking on this button replaces the currently selected entry in the list of line lengths with the value from the field.

Clicking on this button deletes the current entry from the list of dash lengths.

The distance of the extension line from the measured object's edge is entered in this field. The value is entered in the current length unit. Negative values result in extension lines which cross the object.

The minimum size for the end symbol on the dimension line is entered in this field. As the size of the end symbol is worked out from the width of the dimension line, the minimum line width which should be used as the basis of the calculation should be entered here. The value is entered in the current length unit.

If all end symbols (regardless of the dimension line width) should be the same size, the "Minimum size" can be set to the corresponding line width (e.g. `0.25mm`) and the "Size factor" to `0`.

The size factor for the end symbols of dimension lines can be entered in this field. The standard size (which is independent of the dimension line width) is multiplied by this factor.

If all end symbols (regardless of the dimension line width) should be the same size, the "Minimum size" can be set to the corresponding line width (e.g. `0.25mm`) and the "Size factor" to `0`.

This icon represents "Minimum Number of Places" display mode. If the adjacent radio button is selected, only as many decimal places will be displayed as absolutely needed. Whole numbers are shown without decimal places.

This icon represents "At Least One Place" number display mode. If the adjacent radio button is selected, only as many decimal places as needed, but at least one decimal place, are displayed after dimensions. Whole numbers are displayed with one zero after the decimal point.

This icon represents the "Display All Places" number display mode. If the adjacent radio button is selected, dimension numbers are always shown with as many decimal places as specified in the accuracy setting. Whole number are shown with however many decimal places as specified.

This icon represents the "Radial Extension Lines" arc-length dimension display mode. If the adjacent radio button is selected, arc-length dimensions are always shown with radial extension lines.

This icon represents the "Parallel Extension Lines" arc-length dimension display mode. If the adjacent radio button is selected, arc-length dimensions are always shown with parallel extension lines.

This value shows in KB how much memory is reserved and huch much of it is currently used.

This number shows how many objects and instances are in the current drawing.

This value shows the memory required for the current drawing's objects and instances (in KB). This includes the memory required for the undo information.

This number shows the number of blocks in the current drawing.

This value shows the memory required (in KB) for the blocks in the current drawing.

This number shows the number of blocks currently in the block cache.

This value shows the amount of memory (in KB) required for the blocks currently in the block cache.

This number shows the number of font characters currently in memory.

This value shows the amount of memory (in KB) required for the font characters in memory.

This number shows the number of bitmaps currently in memory.

This value shows the memory required for the bitmaps currently in memory.

This area shows whether or not there is useable graphic information on the Clipboard. "Windows Metafile (WMF)", "Enhanced Windows Metafile (EMF)" and "TommySoftware® CAD/DRAW (TVG 4.0)" formats are supported. If multiple formats are available, TommySoftware® CAD/DRAW's own format, which is more precise, will be used.

This value shows the horizontal screen resolution in pixels.

This value shows the useable width (in mm) of the screen area, as specified with the command Options>General.

This value shows the useable width (in inches) of the screen area, as specified with the command Options>General.

This value shows the vertical screen resolution in pixels.

This value shows the height (in mm) of the useable screen area as specified with the command Options>General.

This value shows the height (in inches) of the useable screen area as specified with the command Options>General.

This value shows the color depth (in bits per pixel) used for screen display. It determines how many colors can be displayed at the same time.

This value shows how many colors can be shown on screen at the same time.

A standard monitor size can be chosen from this list. This automatically sets the "Height" and "Width" values.

The width of the useable screen area can be entered in this field. The value is entered in either mm or inches (see the adjacent radio button).
This value is used to display the drawing at its original size. In most cases, it is sufficient to click on one of the above buttons to match this value to the size of monitor size being used.

The height of the useable screen area can be entered in this field. The value is entered in either mm or inches (see the adjacent radio button).
This value is used to display the drawing at its original size. In most cases, it is sufficient to click on one of the above buttons to match this value to the size of monitor size being used.

If this radio button is selected, the values in the adjacent field are in mm.

If this radio button is selected, the values in the adjacent field are in inches.

The size of system memory (in KB) to reserve can be entered in this field. Values between 256 and 1024 are allowed.
System memory is used for internal calculations, e.g. when hatching, generating surfaces or creating equidistants.

The maximum number of undo levels can be entered in this field. Values between 1 and 100 are allowed.

The maximum number of undoable actions has a large effect on the amount of memory required. We recommend values between 3 and 5. This allows a reasonable number of operations to be undone (it is in any case difficult to remember more than 5 operations) and keeps the memory requirement within reasonable limits.

If this check box is set, a dialog will appear at program start-up, offering the possibility to start the CAD/DRAW tour, the tutorial or the reference.

This button displays the current standard path for setting files. These files usually end with `.INF`. The default path serves a "starting value" for all file selection dialogs, from which settings files are selected. Clicking on this button calls a menu, in which one of the eight possible pathes can be edited or deleted. The first path in this list is the "standard path", the others can be accessed from within File Selection Dialog in form of a menu.

This button displays the current standard path for drawing files. These files usually end with .T4G (or .DUP). The default path serves a "starting value" for all file selection dialogs, from which drawing files are selected.
Clicking on this button calls a menu, in which one of the eight possible pathes can be edited or deleted. The first path in this list is the "standard path", the others can be accessed from within File Selection Dialog in form of a menu.

This button displays the current standard path for library files. These files usually end in .T4L. The default path serves a "starting value" for all file selection dialogs, from which library files are selected. Clicking on this button calls a menu, in which one of the eight possible pathes can be edited or deleted. The first path in this list is the "standard path", the others can be accessed from within File Selection Dialog in form of a menu.

This button displays the current standard path for enhanced metafiles. These files usually end in `.EMF`.
The default path serves a "starting value" for all file selection dialogs, from which enhanced metafiles are selected.
Clicking on this button calls a menu, in which one of the eight possible pathes can be edited or deleted.
The first path in this list is the "standard path", the others can be accessed from within File Selection Dialog in form of a menu.

This button displays the current standard path for metafiles. These files usually end in `.WMF`. The default path serves a "starting value" for all file selection dialogs, from which metafiles are selected.
Clicking on this button calls a menu, in which one of the eight possible pathes can be edited or deleted. The first path in this list is the "standard path", the others can be accessed from within File Selection Dialog in form of a menu.

This button displays the current standard path for bitmap files. These files usually end in `.BMP`. The default path serves a "starting value" for all file selection dialogs, from which bitmap files are selected. Clicking on this button calls a menu, in which one of the eight possible pathes can be edited or deleted. The first path in this list is the "standard path", the others can be accessed from within File Selection Dialog in form of a menu.

Clicking on this button appends the value in the edit field to the list of angles.

Clicking on this button inserts the value in the edit field into the list of angles before the currently selected entry.

Clicking on this button deletes the currently selected entry from the list of angles.

This list shows all angles that are active when the custom mode is active.

Enter values into this edit field in order to add them to the list of angles.

If one of these radio button if active, all multiples of the corresponding angle are active.

If this radio button is active, the angles displayed in the list beside are active.

If this radio button is selected, all lines and object edges are drawn through and displayed with a width of one pixel

If this radio button is selected, all lines and object outlines are shown with their own line patterns, but always with a width of one pixel.

If this radio button is selected, all lines and object outlines are displayed with their own line patterns and original line widths. The line width is calculated independently of the zoom, i.e. it is always shown as if the current zoom factor was 100%.

This setting has proved to be the best compromise between quick redraws and recognizability of the line width.

If this radio button is selected, all lines and object outlines are shown with their own line pattern and their original line width. The line width is adapted to the current zoom factor.

If this check box is enabled, all objects and instances in the drawing are displayed. Otherwise, only the geometry (if selected) is displayed.

If this check box is enabled all objects in the drawing are displayed. Otherwise, only instances are shown.

If this check box is enabled all object outlines are shown. Otherwise, only fillings are shown.

If this check box is enabled all object fillings are shown. Otherwise, only object outlines are shown.

If this check box is enabled text and dimension text are shown in the selected font. Otherwise text is displayed in a small font which shows the name of the text and the text itself in the form "[Font Name]Text". This can drastically reduce redraw time when using complicated fonts.

If this check box is enabled instances are displayed completely. Otherwise, just a text in a small font with the name of the library and the block in the form "(Library Name)Block Name" is shown.
This setting also affects groups, as they are displayed via instances.

If this check box is enabled, erasers are displayed in color to make them easier to localize. Otherwise, they are visible only indirectly, because of their effect.

If this check box is enabled, variables inside texts and dimensions are evaluated.

If this check box is enabled, bitmap objects are displayed. Otherwise, just a text in a small font with the name of the bitmap file in the form `(Bitmap File Name)` is shown.

If this check box is enabled, markings are displayed on the screen.

If this check box is enabled, borders of bitmaps and clipping surfaces are displayed.

If this check box is enabled a rectangular frame appears around selected objects.

If this check box is enabled the outlines of selected objects are displayed as dashed lines.

If this check box is enabled, all definition points in the drawing are shown by a small colored square. This also applies to definition points within blocks.

If this radio button is selected, a comma is used as the decimal separator for floating point numbers. In this case 7/4 is shown as 1,75.
Either a point or a comma can be used as the separator when entering numbers, i.e. you can enter `1,75` or `1.75`.

If this radio button is selected, a point is used as the decimal separator for floating point numbers. In this case 7/4 is shown as 1.75.
Only points can be used as the separator when entering numbers, i.e. you can only enter `1.75`. Entering `1,75` causes an error.

Clicking on this button, sets the screen display of the drawing to a low value.

Clicking on this button, sets the screen display of the drawing to a medium value.

Clicking on this button, sets the screen display of the drawing to a high value.

The accuracy for the screen display of the drawing can be entered in this field, using the current line units. The accuracy indicates the average line length which should be used when transforming circles, ellipses, splines and Bézier curves to polylines. The smaller the number, the larger the number of lines and the higher the accuracy.

Clicking on this button sets the output accuracy for the drawing to a low value.

Clicking on this button sets the output accuracy for the drawing to a medium value.

Clicking on this button sets the output accuracy for the drawing to a high value.

The accuracy for the output of the drawing can be entered in this field, using the current line units. The accuracy indicates the average line length which should be used when transforming circles, ellipses, splines and Bézier curves to polylines. The smaller the number, the larger the number of lines and the higher the accuracy.

Clicking on this button sets the accuracy for object transformation to a low value.

Clicking on this button sets the accuracy for object transformation to a medium value.

Clicking on this button sets the accuracy for object transformation to a high value.

The accuracy to be used when transforming objects can be entered in this field, using the current line units.
The accuracy indicates the average line length which should be used when transforming circles, ellipses, splines and Bézier curves to polylines. The smaller the number, the larger the number of lines and the higher the accuracy.

A value for the display size of TrueType fonts can be entered here. The value is given in screen pixels. If a font on screen is smaller than the given value, it is displayed as a filled box.

If this check box is enabled and 8-bit color screen display is used, a special rainbow color palette is used. The colors, in this case, are not dithered.

If this check box is enabled, TrueType-fonts are converted to surfaces for display purposes. This may be necessary with some older video card drivers which do not fully support TrueType-fonts.

If this check box is enabled then the output properties, rather than the specified screen display properties, are used to display pens on screen. This can be used to give a kind of preview of the output.

This check box should not normally be enabled, as otherwise the advantages of the pen concept are lost.

If this radio button is selected, the page margin is used as the basis for calculating the area to be entered.

If this radio button is selected, the frame around all objects and instances to be output is used to calculate the size.

A value can be entered into this field to show how much the output area should deviate from the page margin. The value is entered in the current length unit
If a positive value is entered, then the area is enlarged in every direction by the value. If a negative value is entered, then the area is reduced. The value only has an effect if the "Page margin" check box is enabled.

The value by which the output area should vary from the object frame can be entered in this field.
If a positive value is entered, the area is enlarged in every direction by this amount, if a negative value is entered, the area is reduced. This is only effective if the "Object Frame" check box is enabled.

If this check box is enabled the output area is output as a rectangle. The properties set with the adjacent "Properties" button are used.

Clicking on this button calls the Edit Properties dialog, in which the properties to be used if the area is output can be specified.

If this check box is enabled the display grid is included in output. The properties specified with the help of the "Properties" button are used. The form of the individual grid points is determined with the help of the three marking fields "90° Cross", " 45° Cross" and "Circle", the size in the "Size" field.

Clicking on this button calls the Edit Properties dialog, in which the properties to be used if the display grid is to be output.

If this check box is enabled grid points a.o. are shown by a cross with the angles 0° and 90°.

If this check box is enabled grid points a.o. are shown by a cross with the angles 45° and 135°.

If this check box is enabled grid points a.o. are shown by a cross.

The size for the grid points can be entered here. The value is in the line unit. With circles, this value shows the diameter, with crosses the half line lengths.

If this check box is enabled markings are included in the output. The properties specified with the help of the "Properties" button are used. The form of the individual markings is determined with the help of the three marking fields "90° Cross", " 45° Cross" and "Circle", the size in the "Size" field.

Clicking on this button calls the Edit Properties dialog, in which the properties to be used if the markings are output can be specified.

If this check box is enabled markings a.o. are shown by a line cross with the angles 0° and 90°.

If this check box is enabled markings a.o. are shown by a line cross with the angles 45° and 135°.

If this check box is enabled markings a.o. are shown by a circle.

The size of the markings can be entered in this field. The value is in the line unit. With circles, this value gives the diameter; with crosses the half line length.

If this check box is enabled geometry objects are output. This is independent of the geometry screen display.

If this check box is enabled bitmap objects are output. This is independent of the bitmap object screen display.

If this check box is enabled TrueType fonts are converted to surfaces for output. This is necessary with some older printer drivers which may not support TrueType fonts properly.

If this check box is enabled the screen display properties are used for output instead of the usual output properties. This can be used, for example, to export the drawing as shown on screen to a bitmap or metafile.

This check box should normally be cleared, as the advantages of the pen concept are lost otherwise.

This icon represents the "Round" line end-style.

This icon represents the "Square" line end-style.

This icon represents the "Flat" line end-style.

This icon represents the "Round" line join mode.

This icon represents the "Bevel" line join mode.

This icon represents the "Miter" line join mode.

In this button, the color used for selection frames is shown.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog in which a default color can be chosen.

In this button, the color used for the display of selected objects is shown.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used to show possible results during variant choice is shown.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used to show selected points is shown.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used to draw displayed but unsolicited definition points is shown.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used to show markings is shown.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used to show objects while they are being drawn is shown. This color is also used to draw the frame when selecting objects in an area.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the background color for the drawing window and display areas in dialogs is shown. The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used to show the large crosshair in the active window is shown. The color displayed is linked with the drawing window's background color so the crosshair is only shown in this color if the window background is white.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used to show the small crosshair in the inactive drawing windows is shown. The color displayed is linked with the drawing window's background color so the crosshair is only shown in this color if the window background is white.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used to display the page frame and page symbol is shown.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used to show block names and text displayed in a small font is shown. For example, this is the case if an instance cannot be resolved or a particular font is unavailable.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used to show erasers is shown. If erasers are displayed, they are filled with the background color and outlined and hatched in the color chosen here.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used for assistant lines is shown. Guiding lines are displayed during object input for orientation purposes.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used for the display of darkened layers is shown.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used for the icons symbolizing sub-menus in the Pop Up Menu, toolbox and status window is shown.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used to display the icons symbolizing commands in the pop up menu, the toolbox and the status window is shown.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

In this button, the color used to display highlighted texts is shown.
The color can be changed by clicking onto this button. This calls the Select Standard Color dialog, from which another standard color can be chosen.

Clicking on this button resets all colors to their defaults. These are the colors which you see in this illustration. These colors are matched to each other and meant for use with a white background. They are also suitable for use with a black background.

This list shows all the keys which have commands assigned to them with descriptions of the commands.

Clicking on this button restores the Standard Key Assignments.

Clicking on this button deletes the key assignment shown in the list.

Clicking on this button allows to export a textual description of all currently defined keys. A File Selection Dialog appears where the file name of the text file can be chosen.

If this radio button is selected, clicking the middle mouse button calls the pop up menu which appears at the current mouse position.

If this radio button is selected, clicking the middle mouse button in a drawing window calls the standard menu in text form, which appears at the current mouse position.

If the menu bar is active (see Main Window), the standard menu can not be used at the same time because this would cause problems with the Windows system resources.

If this radio button is selected, the middle mouse button mimics the SHIFT key. It controls the snapping function and when selecting objects with the middle mouse button pressed an area can be entered.

If this radio button is selected, the middle mouse button mimics the CONTROL key. It therefore controls the duplicate function, and also enables several objects to be selected one after another.

If this radio button is selected, clicking the right mouse button calls the pop up menu, which appears at the current mouse position.

If this radio button is selected, clicking the right mouse button in a drawing window calls the standard menu, which appears at the current mouse position

If the menu bar is active (see Main Window), the standard menu can not be used at the same time because this would cause problems with the Windows system resources.

A description of the command chosen is displayed in this area. This command can only be assigned to one button.

These buttons symbolize the keyboard keys which can have commands assigned to them. Pressing a button assigns the command shown above to the button and the dialog is closed.
If the "Shift" check box is enabled at this time, then the command is assigned to the key combination SHIFT+CHOSEN KEY.
If the "Info Mode" check box is enabled at this time, the assignment is not changed; instead the command currently assigned to the key is shown. The dialog is not closed.

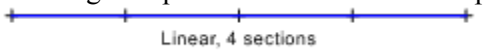
If this checkbox is enabled when a key is selected, the command is not assigned to the key itself, but to the key combination SHIFT+SELECTED KEY.

If this check box is enabled when the key is selected, the submenu containing the command is assigned to the key. This check box is only available if the selected command is contained in a submenu.

If this checkbox is enabled when a key is selected, no command is assigned to the key; instead, the command currently assigned to that key is shown.

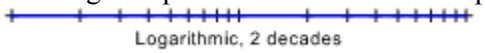
The number of sections for linear division is entered here. It can be between 2 and 999.

If this radio button is selected, the length chosen is divided into multiple sections of equal length. Markings are placed at the start and end points of the chosen length (see graphic).



The decade number for logarithmic division is entered here. It can be between 1 and 99.

If this radio button is selected, the selected length is divided logartihmically into multiple decades. Markings are placed at the start and end points of the length (see graphic).



The logarithmic division always begins at the start point, i.e. the start point is the zero point. If, for example, a logarithmic ruler is to be created, then the origin of the ruler must be specified as the start point of the length to be divided up.

The number of divisions for division using the golden section is entered here. It can be between 1 and 999.

If this radio button is selected, the selected length is divided into multiple sections according to the golden section. Markings are placed at the start and end points of the length. (see graphic).

Golden section, 6 divisions

Division according to the golden section begins at the start point, i.e. the rest of the first division is at the start point, then the rest of the second division and so on. The rest of the last division lies at the end point.

The length of the first section in the geometric progression is entered here. The value entered is in the current length unit.

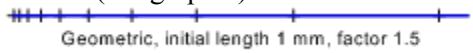
If this radio button is selected, the geometric progression is calculated with a fixed factor. It is automatically set, as soon as the factor is altered.

A fixed factor for the geometric progression can be entered here.

If this radio button is selected, the geometric progression is calculated with a fixed number of sections. It is automatically set as soon as the fixed number is altered.

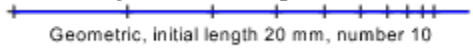
A fixed number of sections for the geometric progression can be entered here. It can be between 1 and 999.

If this radio button is selected, the selected length is divided according to a geometric progression. The way in which the length is divided depends on whether a fixed factor or number of sections has been entered (see graphic).



Geometric, initial length 1 mm, factor 1.5

If a fixed factor is entered, the selected length can not be used fully in most cases. In these cases the rest of the length to the end point remains unused.



Geometric, initial length 20 mm, number 10

If a fixed number is entered, an unending geometric progression is used as the basis for the calculation as the calculation would otherwise be too complex. Because of this, the length, especially with small numbers of sections, is only partly used.

The length of the first section in the arithmetic progression is entered here. The value entered is in the current length unit.

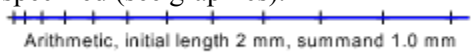
If this radio button is selected, the arithmetic progression is calculated using a fixed summand. It is automatically set as soon as the fixed summand is altered.

The fixed summand for the arithmetic progression can be entered here. The value entered is in the current length unit.

If this radio button is selected, the arithmetic progression is calculated using a fixed number of sections. It is automatically enabled, as soon as the fixed number is changed.

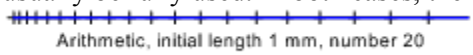
A fixed number of sections for the arithmetic progression can be entered here. It can be between 1 and 999.

If this radio button is selected, the selected length is divided according to an arithmetic progression. Exactly how it is divided depends on whether a fixed summand or a fixed number of sections have been specified (see graphics).



Arithmetic, initial length 2 mm, summand 1.0 mm

If a fixed, negative summand is entered, at some stage the steps will become negative. If this happens before the end point is reached, then the progression is broken. In addition, the specified length can not usually be fully used. In both cases, the rest of the length to the endpoint remains unused.



Arithmetic, initial length 1 mm, number 20

If a fixed number of steps is entered, it is usually only possible to divide the length up completely if the sections become negative towards the end, i.e. the progression runs slightly beyond the end point. In this case, only the sections which lie on the length are used. This means that the number of sections can be less than desired.

A value can be entered into this field, which will be transferred to the underlying list.

All the figures used during division on the basis of relationships between figures are listed here.

The sum of all the numeric values in the list is shown here. This makes it easier to fill up to a particular value (e.g. 100 if percentages are used).

Clicking on this button deletes the currently selected value from the list.

Clicking on this button, transfers the value from the above field into the list, above the currently selected value. The list can hold up to 20 numbers.

Clicking on this button puts the value in the above field into the list instead of the current value.

Clicking on this button adds the value in the above field to the list of values. The list can contain a maximum of 20 numbers.

If this radio button is selected, the length is divided up according to the relationship of the numbers in the list (see graphic).

Relation, values 30 : 45 : 15 : 10

If this check box is enabled the objects to be copied during a multiple copying operation are rotated according to their position relative to the reference object. This checkbox is not available for all multiple copying operations.

The substitute font can be chosen from this list. This font is used as a substitute for any of TommySoftware® CAD/DRAW's own fonts which are unavailable. Any TrueType font installed on the system or any available TommySoftware® CAD/DRAW font can be used.

This list shows the name of all currently available TommySoftware® CAD/DRAW fonts are shown. The fonts shown here are automatically loaded when the program is next started.
The fonts may be marked by letters in front of their names. The letters show operations which are carried out when the dialog is closed by clicking on "OK". The following letters may occur:

'O' stands for Open. A font marked like this is loaded into memory and made available. See the "Open" button.

'R' stands for Remove. A font marked like this is removed from memory and is then no longer available. See the "Remove" button.

Clicking on this button enables a new TommySoftware® CAD/DRAW font to be added to the list. A File Selection Dialog in which the font to be used can be selected.
After selecting the font file, the title of the font is placed in the list. It is then marked with an 'O' (for "Open"). The font itself is loaded when the dialog is closed by clicking on "OK".

Clicking on this button, marks the currently selected font in the list with an 'R' (for "Remove"). It is removed from memory when the dialog is closed by clicking on "OK". If the button is clicked again the marking is removed and the font is not removed from memory.

Clicking on this button allows the details of the currently selected font to be viewed. To do this, the View Details dialog appears.

This icon represents "Outline" fill mode. An object with this fill mode will only have its outline drawn. If this mode is selected, all subsequently generated text objects will use this filling mode. This is particularly a good idea if the text is to be output on a plotter.

This icon represents "Filling" fill mode. An object with this fill mode will only have its filling drawn. Objects which have no filling (e.g. lines) are not visible if this filling mode is applied to them.
If this mode is selected, all subsequently generated text objects will use this filling mode. This is particularly a good idea when using TrueType fonts and output on a dot matrix, inkjet or laser printer.

This icon represents "Filling & Outline" fill mode. An object with this fill mode will be drawn with both its filling and outline. Objects which do not have fillings (e.g. lines) only have their outlines drawn.
If this mode is selected, all subsequently generated text objects will use this filling mode.

This icon represents "Eraser" fill mode. An object with this fill mode will only have its filling drawn in background color. Objects which have no filling (e.g. lines) are not visible if this filling mode is applied to them.
If this mode is selected, all subsequently generated text objects will use this filling mode.

This icon represents "Eraser & Outline" fill mode. An object with this fill mode will be drawn with both a filling in background color and outline. Objects which do not have fillings (e.g. lines) only have their outlines drawn.
If this mode is selected, all subsequently generated text objects will use this filling mode.

If this radio button is selected, subsequently generated text objects do not have a particular fill mode assigned to them, but instead use the fill mode determined by the pen used to draw them.

Clicking on this button closes the dialog and carries out all specified changes. All operations which are displayed with a letter in the font list are carried out at this time, i.e. the fonts are either loaded into or removed from memory.

The name of the library is displayed in this area.

If this check box is enabled, the specified library name is used for all instances. It is automatically reset as soon as the library name is changed or the "Select" button is used to choose a block.

The name of the block is displayed in this area.

If this check box is enabled the specified block name is used for all instances. It is reset automatically as soon as the block name is altered or the "Select" button is used to choose a block.

The value to be used for the matrix element M11 can be entered in this field. Together with the elements M12, M21 and M22 this determines the size, direction and rotation of the block which the instance refers to.

If this check box is enabled the specified matrix element M11 is used for the display matrix of all instances. It is automatically reset as soon as the matrix element M11 is changed.

The value to be used for the matrix element M12 can be entered in this field. Together with the elements M11, M21 and M22 this determines the size, direction and rotation of the block which the instance refers to.

If this check box is enabled the specified matrix element M12 is used for the display matrix of all instances. It is automatically reset as soon as the matrix element M12 is changed.

The value to be used for the matrix element M21 can be entered in this field. Together with the elements M11, M12 and M22 this determines the size, direction and rotation of the block which the instance refers to.

If this check box is enabled the specified matrix element M21 is used for the display matrix of all instances. It is automatically reset as soon as the matrix element M21 is changed.

The value to be used for the matrix element M22 can be entered in this field. Together with the elements M11, M12 and M21 this determines the size, direction and rotation of the block which the instance refers to.

If this check box is enabled the specified matrix element M22 is used for the display matrix of all instances. It is automatically reset as soon as the matrix element M22 is changed.

A value for the matrix element M31 can be entered here. This determines the X coordinates for the position of the block's insertion point. The value entered is in the current length unit.

If this check box is enabled then the specified matrix element M31 is used for the display matrices of all instances. It is reset as soon as the matrix element M31 is altered.

A value for the matrix element M32 can be entered here. This determines the Y coordinates for the position of the block's insertion point. The value entered is in the current length unit.

If this check box is enabled the matrix element M32 is applied to the display matrices of all instances. It is automatically reset as soon as the matrix element M32 is altered.

Clicking on this button calls the Select Block dialog, where the library name and the block name can be chosen.

Text can be entered into this field. It can contain up to 8000 characters. A paragraph end can be entered by pressing the ENTER key. A tab space can be entered by using the "Tab" button.

Clicking on this button inserts a tab space into the above text. It is shown by a '¬' symbol.

Clicking on this color field calls the Choose Color dialog, in which the display color for the geometry can be chosen. A pale color which is not used in the drawing itself should be chosen, to make it as easy as possible to distinguish geometry objects.

Clicking on this button adds lines to the previously-specified curve. Their end points can be positioned anywhere.

1. *Enter end point*
   Several points can be entered one after another. The curve is extended from the end with a series of lines through all subsequently entered points. These points can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

Clicking on this button adds *one* line to the previously-specified curve. The length and angle are added numerically.

1. *Enter length and angle*
   A dialog in which a length and an angle can be entered appears. After clicking on "OK" a line with the specified length, running at the specified angle, is added to the curve. Clicking on the "Cancel" button reverts to the previous dialog.

Clicking on this button adds Bézier curves to the specified curve. The Bézier curve direction is fully independent of the current curve direction.

1. *Enter angel point 1*
   The first angel point determines, through its position relative to the current curve endpoint, the initial steepness of the Bézier curve, and by its spacing the size and curvature of the Bézier curve. The angel point can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).
2. *Enter curve end point*
   The end point of the Bézier curve can be specified with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).
3. *Enter angel point 2*
   The second angel point determines, through its position relative to the end point of the Bézier curve, the final gradient of the Bézier curve and, by its distance, the size and curvature of the Bézier curve. The angel point can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

Clicking on this button appends Bézier curves to the previously specified curve. The Bézier curve starts at the previous curve's end point which ensures a smooth transition.

1. *Enter angel point 1*
   The first angel point determines, by its distance from the current end point of the curve, the size and curvature of the Bézier curve. The angel point can be entered with the mouse by clicking anywhere in the drawing. A perpendicular is dropped from this point to a straight line running in the current direction of the curve, to determine the final angel point. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).
2. *Enter curve end point*
   The end point of the Bézier curve can be specified with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).
3. *Enter angel point 2*
   The second angel point determines, by its position relative to the end point of the Bézier curve, the final gradient of the Bézier curve and, by its distance, the size and curvature of the Bézier curve. The angel point can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

Clicking on this button adds a circular arc to the previously identified curve. The circular arc is determined by two end point and a point through which it should run.

1. *Enter arc end point*
   The end of the circular arc can be entered with the mouse by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).
2. *Enter point on arc*
   The point through which the circular arc should run, can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

Clicking on this button adds a circular arc to the previously entered curve. The circular arc begins tangential to the specified curve and ends at a specified point.

1. *Enter arc end point*
   The end of the circular arc can be entered with the mouse by clicking at any point in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

Clicking on this button attaches circular arcs to the previously specified curve. They are determined by specifying a center point and the end angle.

1. *Enter center point*
   The center point of the circular arc can be entered with the mouse by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).
2. *Enter end angle*
   The end angle can be entered with the mouse, by clicking anywhere in the drawing. The position clicked on, relative to the center point, determines the end angle. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8). In addition, you can enter an angle. This determines the arc's end angle.

   The appearance of the arc is determined by the current arc direction setting. This can be changed using the command Toggle Arc Direction (F2), with the F2 key or the button in the panel. If the arc direction is positive, then the arc is drawn from the current curve end point to the end angle in a mathematically positive (I.e. anticlockwise) direction. If the arc direction is negative, then the arc is drawn from the current curve end point to the end angle in a mathematically negative (i.e. clockwise) direction (see graphic).

   The final appearance of the arc is shown while it is being drawn, and alters instantly if the arc direction is changed while it is being entered.

The arc direction of existing arcs is unaffected. To change the arc direction of a circular arc within a curve, use the command. Trim Curve>Edit.

Clicking on this button attaches *one* circular arc to the previously specified curve. The position of the center point relative to the previous curve and the arc angle of the circle are then entered directly.

1. *Enter angle, length and arc angle*
   A dialog appears, in which a length, an angle and an arc angle can be entered. After clicking on the "OK" button an arc whose center lies at the specified position relative to the previous curve and whose arc angle is the same as that entered is attached to the curve.
   If the arc angle is positive, the arc is drawn in a mathematically positive sense (i.e. anticlockwise), otherwise in a mathematically negative (i.e. clockwise) sense.
   Clicking on the "Cancel" button reverts to the previous dialog.

Clicking on this button begins a new outline. Before this, the last-entered outline is closed with a line from its start point to its end point.

1. *Enter start point*
   The start point of the new outline can be entered with the mouse, by clicking anywhere in the drawing. Alternately, you can enter numerical coordinates, see Coordinate Entry (F8).

This button is not available when entering a curve, as a curve can consist of multiple outlines.

The current curve direction, i.e. the gradient to the current endpoint of the curve, is shown here. This value is used as a basis, if following curve parts are to be entered tangentialy or relatively polar.
For example, if a Bézier curve is to be rotated by exactly 90 degrees relative to the previous end, this value can be increased by 90 degrees (e.g. by entering `+90` after it), in order to attach the Bézier curve using the operations "Bézier Curve, Tangential Tangents".

Clicking on this button inserts the curve drawn up to this point in the drawing.

Clicking on this button returns to curve entry, as if this dialog had not been called.
If the dialog is closed immediately after entering the start point, the specified start point is ignored, i.e. the process must be started again.

Clicking on this button cancels curve entry, discarding all the entered points.

If this check box is enabled only objects assigned to the specified layer are used. It is automatically enabled as soon as a layer is selected.

If this check box is enabled., only the objects assigned to the specified pen are used. It is automatically enabled as soon as a pen is selected.

If this check box is enabled, only objects which have the specified line pattern are used. It is automatically enabled as soon as a line pattern is selected.

If this check box is enabled, only objects with the specified line width are used. It is automatically enabled as soon as a line width is selected.

If this check box is enabled, only the objects with the specified line color are used. It is automatically enabled as soon as the line color is changed.

If this check box is enabled, only objects with the specified fill mode are used. It is automatically enabled as soon as the fill mode is changed.

If this check box is enabled, only objects with the specified fill color are used. It is automatically enabled as soon as the fill color is changed.

The object types to be used can be chosen from this list. Several object types can be selected by holding down the CONTROL key while selecting object types. If the SHIFT key is held down, then areas can be selected.

If this check box is enabled, only objects of the types selected in the list are used. It is automatically set as soon as object types are selected in the list.

If this check box is set, entities of type "Objects" are used.

If this check box is set, entities of type "Instances" are used.

If this check box is set, entities of type "Group" are used.

If this check box is set, entities of type "Position Number" are used.

If this check box is set, entities of type "Module-Defined" are used.

Clicking on this button resets the dialog to its default values, i.e. all object will be used.

Clicking on this button shows a command-specific dialog displaying further options.

This icon represents the "Bend" reference line mode. If this line mode is active, an additional point can be entered that defines the bending position.

This icon represents the "Straight" reference line mode.

This icon represents the "45° Start, 90° Ending" reference line mode.

This icon represents the "90° Start, 45° Ending" reference line mode.

This icon represents the "Vertical Start, Horizontal Ending" reference line mode.

This icon represents the "Horizontal Start, Vertikal Ending" reference line mode.

This icon represents the "Horizontal Line" reference line mode.

This icon represents the "Vertical Line" reference line mode.

This icon represents the "Rectangle" reference frame mode.

This icon represents the "Rhomb" reference frame mode.

This icon represents the "Circle" reference frame mode.

This icon represents the "Ellipse" reference frame mode.

In this edit field, the frame offset can be entered. This offset determines the additional size of the frame in relation to the reference text's size.

If this check box is enabled the current reference arrow mode is applied to all objects. It changes automatically as soon as the reference arrow mode is changed.

If this check box is enabled the current reference line mode is applied to all objects. It changes automatically as soon as the reference line mode is changed.

If this check box is enabled the current reference frame mode is applied to all objects. It changes automatically as soon as the reference frame mode is changed.

If this check box is enabled the current offset is applied to all objects. It changes automatically as soon as the offset is changed.

The prefix text is entered in this field. This text is output, without a space, in front of the dimension. For example, here the letter R is put in front of a radius measurement.

In this field, the dimension itself can be altered. This alteration is only permanent if the "Adapt Dimension" check box is *not* enabled. Otherwise the dimension is recalculated and overwritten when the dialog is closed.

The suffix is entered in this field. It appears after the dimension (before any tolerances) without a space between it and the number. For example, measurement units can be shown here.

The upper tolerance is entered in this field. This tolerance appears after the suffix, raised by about half a line.

The lower tolerance is entered in this field. This tolerance appears after the suffix, lowered by about half a line.

Clicking on this button inserts the surface symbol (Ansi 128) into the current field. This symbol is only available with TommySoftware® CAD/DRAW's own fonts. In TrueType fonts, Ansi 128 is usually unassigned.

Clicking on this button inserts the degree symbol ° (Ansi 176) into the current field.

Clicking on this button inserts the Plus / Minus character ± (Ansi 177) in to the current field.

Clicking on this button inserts the diameter symbol Ø (Ansi 216) in to the current field.

Clicking on this button inserts the multiply symbol × (Ansi 215) in to the current field.

If this check box is enabled the measurement is updated automatically, adapting to changes in the measurement position, movement of individual definition points or changes in scale.
If the measurement is to be set "by hand", then this check box must be cleared so that the measurement is not immediately overwritten.

The prefix is entered in this field. This prefix appears, without a space, in front of the X coordinates. For example, this is where the opening bracket is placed.

The X coordinate can be changed in this field. This change is only permanent if the "Adapt Dimension" check box is not enabled. Otherwise the X coordinate is automatically recalculated when the dialog is closed.

The mid text can be entered in this field. This text appears, without spaces, between the two coordinates. For example a slash \ may be entered here.

The Y coordinate can be changed in this field. This change is only permanent if the "Adapt Dimension" check box is not enabled. Otherwise the Y coordinate is automatically recalculated when the dialog is closed.

The suffix is entered in this field. This suffix appears, without a space, after the X coordinates. For example, this is where the closing bracket is placed.

If this check box is enabled, the coordinates are automatically updated. They adapt themselves automatically to changes in the position of the dimension or changes in the scale.
If the coordinates are to be set "by hand", then this check box must be cleared so that the coordinates are not immediately overwritten.

The hatching type to be used is chosen from this list. You can choose any currently defined hatching type.

Clicking on this button exits the dialog box making the currently chosen hatching type the active hatching type. All modifications will be maintained.

Clicking on this button enables a new hatching type to be defined. A dialog appears in which the name of the new hatching type can be entered.
The new hatching type initially has the same settings as the hatching type currently chosen from the list. Therefore a hatching type must be selected before a new one can be defined.

Clicking on this button calls the Edit Hatching Type dialog, from which the selected hatching type can be edited.

Clicking on this button calls a dialog in which the name of the selected hatching type can be edited. The name of the "*Standard" hatching type can not be changed.

Clicking on this button deletes the selected hatching type. This choice must be confirmed. The "*Standard" hatching type can not be deleted.

If this check box is enabled, the chosen line sequence 1 will be included in the hatching.

This buttons displays the selected line sequence 1. Clicking on this button calls a list off all currently defined line sequence for selection.

In this edit field, the rotation angle of line sequence 1 can be entered.

If this check box is enabled, the chosen line sequence 2 will be included in the hatching.

This buttons displays the selected line sequence 2. Clicking on this button calls a list off all currently defined line sequence for selection.

In this edit field, the rotation angle of line sequence 2 can be entered.

If this check box is enabled, the chosen block will be included in the hatching.

This area shows the library name of the chosen block.

This area shows the block name of the chosen block.

Clicking this button calls the Select Block dialog, in which a block for the hatching can be chosen.

In this edit field, the scaling factor of the chosen block can be entered.

In this edit field, the rotation angle of the chosen block can be entered.

In this edit field, the first (relative) portion of the horizontal offset can be entered. This offset defines the horizontal movement for each block insertion.

In this edit field, the second (absolute) portion of the horizontal offset can be entered. This offset defines the horizontal movement for each block insertion.

In this edit field, the first (relative) portion of the vertical offset can be entered. This offset defines the vertical movement for each line of block insertion.

In this edit field, the second (absolute) portion of the vertical offset can be entered. This offset defines the vertical movement for each line of block insertion.

In this edit field, the first (relative) portion of the line offset can be entered. This offset defines the horizontal movement for each line of block insertion.

In this edit field, the second (absolute) portion of the line offset can be entered. This offset defines the hoizontal movement for each line of block insertion.

In this edit field, the first (relative) portion of the global offset can be entered. This offset defines the starting point of the hatching relative to the current hatch fixed point.

In this edit field, the second (absolute) portion of the global offset can be entered. This offset defines the starting point of the hatching relative to the current hatch fixed point.

In this edit field, the rotation angle of the whole hatching (in addition to the separate rotation angles) can be entered.

The line sequence to be used is chosen from this list. You can choose any currently defined line sequence.

Clicking on this button enables a new line sequence to be defined. A dialog appears in which the name of the new line sequence can be entered.
The new line sequence initially has the same settings as the line sequence currently chosen from the list. Therefore a line sequence must be selected before a new one can be defined.

Clicking on this button calls the Edit Line Sequence dialog, from which the selected line sequence can be edited.

Clicking on this button calls a dialog in which the name of the selected line sequence can be edited. The name of the "*Standard" line sequence can not be changed.

Clicking on this button deletes the selected line sequence. This choice must be confirmed. The "*Standard" line sequence can not be deleted.

The distance of the selected line to its predecessor is entered here.

If this check is enabled, the selected line is active.

This list presents all possible eight lines of the line sequence for selection. For each line, it shows their distance to its predecessor and whether it is active of not.

Clicking this button calls the Edit Properties dialog, from which the properties of the selected line can be edited.

This area shows the current path. Analog to paths, this path shows nested folders using a ‘\' symbol as a divider.

The library to be used can be chosen from this list. All known libraries (see Libraries) as well as the pseudo-library "*Internal Blocks" are listed. The pseudo-library "* Internal Blocks" contains all the blocks and groups from the current drawing. These are always saved with the drawing, and do *not* form a library.

A folder can be opened or closed from this list. Double-clicking on a folder name opens that folder.
Double-clicking on the entry " [..]" closes the current folder and returns to the one above it.
If an order is to be deleted or renamed, it can be selected by simply clicking on it in the list.

The block to be used can be selected from this list. All the blocks in the selected folder from the library are shown, also those which are not yet in memory.

The name of the block to be used is entered in this field. The name of the block selected in the list is automatically inserted here.
If a name is entered, the program searches for a block in the list whose name matches the characters entered. If a match is found, it is highlighted and selected in the list.
The block name and folder name together can be a maximum of 63 characters.

If this check box is enabled the currently selected block is displayed. This is only possible if the selected block is already in memory.

The currently selected block is displayed in this area. This only happens if the "Display" check box is enabled and the chosen block is in memory.

Three special elements are shown next to the block:

 A small square in the color of a selected definition point (usually Blue) shows the block's insertion point. This point is positioned with the mouse when the block is inserted.

 A crossed-through rectangle in the selection color (usually Red) shows the frame around the block.

 A hatched rectangle in the color of the page margin (usually Violet) shows the "block frame" which shows the character cell of text blocks. With other blocks this is usually identical to the surrounding frame.

If this check box is enabled when a block is inserted, then the layer assignment of the instance will automatically be transmitted to the block itself.

If this check box is enabled when a block is inserted, then all the block's default local attributes are copied to the instance of that block.

If this check box is enabled the Define Attributes dialog appears after inserting the block and can be used to edit the attributes of the generated instances.

The scaling factor of the block relative to its original size can be entered in this field.

The rotation angle of the block relative to its original position can be entered in this field. The value is entered in the current angle unit.

Clicking on this button sets the block's rotation angle to 0°.

Clicking on this button sets the block's rotation angle to 45°.

Clicking on this button sets the block's rotation angle to 90°.

Clicking on this button sets the block's rotation angle to 180°.

Clicking on this button sets the block's rotation angle to 270°.

The number of blocks contained in the current library folder is shown here. A folder can have up to 1000 sub-folders and 1000 blocks; the whole library can hold 1,000,000 blocks.

Clicking on this button removes the assigned block from the current symbol button, i.e. sets it to "(UNDEFINED)". This button is only visible when the allocation of a symbol window button is edited from this dialog.

If this check box is enabled the objects from which the block is made up are deleted and replaced by an instance of the newly created block.

If this check box is enabled the standard attributes for the library are applied to the newly generated block. These attributes can be defined with the Libraries command or with the "Attributes" button.

If this check box is enabled the Define Attributes dialog appears after the block is created; the local and global attributes of the newly generated block can be edited from this dialog.

If this check box is enabled the properties of the objects out of which the block is created are resolved, i.e. made independent of pen and layer.

If this check box is enabled the Edit Properties dialog appears after the block is generated in which the output properties of the new block can be edited.

Clicking on this button enables the name of the currently selected block to be altered. A dialog appears in which the name can be edited. This button is only available if the selected library can be changed and the selected block is in memory.

If the name of a block is changed, all instances which refer to this block will become invalid because they will now refer to an undefined block name. You can either replace the instances with the command Replace Block Name or use the command Edit Properties to change the block name in the instances to match the new block name.

Clicking on this button deletes the currently selected block. This operation must be confirmed. This button is only available if the selected library can be changed and the selected block is in memory.

Clicking on this button calls the Define Attributes dialog, in which the local and global attributes of the selected block can be edited. This button is only available if the selected library can be changed and the selected block is in memory.

Clicking on this button calls the Edit Properties dialog, in which the output properties of the currently selected block can be edited. This button is only available if the selected library can be changed and the selected block is in memory.

By clicking on this button you can create a new folder which will be located in the currently opened folder. A dialog appears in which the new folder's name can be entered. This button is only available if the selected library can be altered and is fully loaded into memory (i.e. the cache is not in use).

Clicking on this button enables the name of the current folder to be changed. A dialog appears in which the name can be edited. This button is only available if the selected library can be altered and is fully loaded into memory (i.e. the cache is not in use).

If the name of a folder is changed, all instances which refer to blocks in that folder will no longer be valid, as they will contain a block name which is no longer defined. You can either replace the instances with the command Replace Block Name or use the command Edit Properties to change the block name in the instances to match the new block name.

Clicking on this button deletes all blocks and sub-folders from the current folder. This command must be confirmed. This button is only available if the selected library can be altered and is fully in memory (i.e. the cache is not used).

Clicking on this button calls the Edit Properties dialog, in which either individual or multiple properties of all the blocks in the current library can be altered. This button is only available if the selected library can be altered and is fully in memory (i.e. the cache is not used).

Clicking on this button calls the Copy Block dialog, where individual blocks or whole folders can be copied.

The name of the library from which the block or folder is to be copied is shown in this area.

The name of the library to which the block or folder is to be copied is shown in this area.

The name of the block or folder is to be copied is shown in this area.

The name of the block or folder is to be created is shown in this area.

Clicking on this button calls the Select Block dialog in which the block or folder to be copied can be selected.

Clicking on this button calls the Select Block dialog in which the name of the copy (block or folder) can be specified.

The block to be copied is shown in this area. This only happens if the "Display" check box is enabled and the block is already in memory.

Three special elements are shown next to the block:

 A small square in the color of a selected definition point (usually Blue) shows the block's insertion point. This point is positioned with the mouse when the block is inserted.

 A crossed-through rectangle in the selection color (usually Red) shows the frame around the block.

 A hatched rectangle in the color of the page margin (usually Violet) shows the "block frame" which shows the character cell of text blocks. With other blocks this is usually identical to the surrounding frame.

The block to be replaced is shown in this area. This only happens if the "Display" check box is enabled and the block is already in memory.

Three special elements are shown next to the block:

 A small square in the color of a selected definition point (usually Blue) shows the block's insertion point. This point is positioned with the mouse when the block is inserted.

 A crossed-through rectangle in the selection color (usually Red) shows the frame around the block.

 A hatched rectangle in the color of the page margin (usually Violet) shows the "block frame" which shows the character cell of text blocks. With other blocks this is usually identical to the surrounding frame.

If this check box is enabled the block which is to be copied is shown. This is only possible if the block is already in memory.

If this check box is enabled then the block to be replaced is shown. This is only possible if the block is already in memory.

Clicking on this button copies an individual block. If the target block already exists, a dialog appears which asks whether it should be replaced or not.
This button is only available if the source block is in memory, and the target library can be changed.

Clicking on this button copies all the blocks (but not the sub-folders) in the current folder. If blocks with the same name already exist in the target folder, then you are prompted as to whether they should be overwritten or if only the other blocks should be copied.
This button is available if the source library is completely in memory (i.e. the cache is not being used) and the target library can be altered.

The name of the library from which the block whose attributes are to be altered comes or which the instance refers to.

The name of the block whose attributes are to be edited are displayed in this area. If the local attributes of an instance are edited, the name of the block to which the instance refers is shown here.
If the standard attributes of a library are altered, the words "Standard Attributes" appear here.

All currently defined local and global attributes of the block or instance with names and content are shown here.
The attributes are marked by letters in front of the attribute names. These letters show the attribute types:

'G' stands for global attribute. Global attributes can only be defined in blocks, not instances. If the attributes of an instance are edited, the attributes of the block to which the instance refers shown, but they cannot be changed.

'L' stands for local attribute. Local attributes can be defined in blocks and instances.

'T' stands for text attribute. Text attributes can contain any text, up to a maximum of 250 characters.

'N' stands for number attribute. Number attributes may be empty or contain text which shows a valid number. They can be up to 250 characters long.

Clicking on this button enables a new attribute to be defined. First of all, a dialog appears where the name of the new attribute can be entered. The name entered here must not already be in use.
If no attribute is selected, then the new attribute initially uses the same settings as the selected attribute. Otherwise the new attribute uses the settings "Local attribute" and "Text attribute".

Clicking on this button calls the Edit Attributes dialog where the selected attributes can be changed.

Clicking on this button calls a dialog in which the name of the selected attribute can be edited.

Clicking on this button deletes the selected attribute. This must be confirmed from the dialog which appears.

The contents of the attribute can be edited from this field. The maximum length of the attribute is 250 characters. If the attribute is a "Number attribute" then the content must be either a valid number or it must be empty.

If this radio button is selected, the attribute is a global attribute. Global attributes can only be defined in blocks, not in instances. They are valid for all instances which refer to the block.
Global attributes are used for information which is the same whenever the block is used. These are mostly details like article numbers, descriptions, prices, weights and so on.

If this radio button is selected, the attribute is a local attribute. Local attributes can be contained in blocks and instances. In blocks, they serve as defaults, i.e. they show which local attributes an instance of the block should have.
Local attributes are used for information which varies according to the use of the block. These are things like color, size, etc.

If this radio button is selected, the attribute is a text attribute. Text attributes can contain any text, up to a maximum length of 250 characters.

If this radio button is selected, the attribute is a number attribute. Number attributes may only contain text which shows a valid number or they must be empty. The maximum length is 250 characters.

If this radio button is selected the part list for all instances of blocks from the selected library is generated.

If this radio button is selected the parts list for all instances in the drawing is generated.

If this radio button is selected the part list for all currently selected instances in the drawing is generated.

The library for whose instances a parts list should be generated can be selected from this list. The list contains all libraries from which at least one block is referenced by an instance in the drawing.

If this radio button is selected, a data entry is created for every instance of a block. If there are several instances of a block, then an entry is created for each instance.
Local attributes and implicit attributes such as position, rotation angle and scaling are only available if this setting is selected.

If this radio button is selected, only one data entry is created for all instances of a block. If several instances of the same block exist, they are counted and the total is available as an implicit attribute. Local attributes and implicit attributes such as position, rotation angle and scaling are *not* available if this setting is selected.

If this radio button is selected, a data entry is created for every position number. If there are identical position numbers, then an entry is created for each positions number.
Local attributes and implicit attributes such as position, rotation angle and scaling are only available if this setting is selected.

If this radio button is selected, only one data entry is created for all identical position numbers. If several identical position numbers exist, they are counted and the total is available as an implicit attribute. Local attributes and implicit attributes such as position, rotation angle and scaling are *not* available if this setting is selected.

The name of the file to which the parts list should be written can be entered in this area. It can be changed by clicking on the "Select" button.

If this radio button is selected the parts lists are generated using Ansi characters, which are used by all Windows programs.

If this radio button is selected the parts lists are generated using the extended ASCII characters, which are used by most DOS programs.

Bear in mind that if the ASCII character set is used, then conversion of the attribute data is needed. This usually causes no problems, but some special characters may not be converted correctly.

Clicking on this button enables a parts list file to be selected. This is done from a File Selection Dialog, in which the file can be specified. This file does not already have to exist.

The characters to be used as record delimiters can be chosen from this list. This is output after each "line" of the parts list.
Control characters are shown in the list in pointed brackets. The following control characters (in various combinations) are available:

<TAB>	Tabulator	Ansi 9
<LF>	Line feed	Ansi 10
<CR>	Carriage return	Ansi 13

The characters to be used as field delimiters can be chosen from this list. This is output after each "column" of the parts list.
<TAB>Control characters are shown in the list in pointed brackets. The following control characters (in various combinations) are available:

<TAB>	Tabulator	Ansi 9
<LF>	Line feed	Ansi 10
<CR>	Carriage return	Ansi 13

The characters to be used as the text delimiter can be chosen from this list. This is output before and after each text attribute. If "None" is chosen, no text delimiter will be output as well as the data field delimiter.

A text can be entered in this field which will be output in the parts list if a text attribute does not exist in a specified block.
Useful substitute texts are, apart from a blank, the texts "-" and "N/A". The substitute texts should be entered without quotes or similar delimiters, as a text delimiter is always output before and after it.

A number can be entered in this field to be output when a particular number attribute is not available in a specified block.
Useful substitute numbers, are, apart from a blank, "0" and "0.0".

Clicking on this button displays the dialog Parts List Options, with allows to edit further options.

Clicking on this button calculates all the attributes available for the parts list with the chosen settings. The Parts List Attributes dialog appears, in which the attributes to be included in the parts list can be selected.

All the attributes available for output in the parts list are included in this list. These are all global (and, if the relevant check box in the previous dialog was enabled) local attributes present in one of the affected blocks or instances.
Some implicit attributes, amongst others those which are products of the display matrix, are listed. The following implicit attributes are possible:

"*0 Block Name"
The name of the block, including all folder names (*Text* type).

"*1 Library Name"
The name of the library, with a blocks and groups this is "* Internal Blocks" (*Text* type).

"*2 Position"
Position in the list, beginning with 1 (*Number* type).

"*3 Number"
Gives the number of instances of the same block. If "Equal instances together" is enabled, this value is always 1. (*Number* type).

"*4 Coordinate x"
Shows the X coordinate of the instance's insertion point in mm relative to the center of the page (*Number* type).

"*5 Coordinate y"
Shows the Y coordinate of the instance's insertion point in mm relative to the center of the page (*Number* type).

"*6 Rotation"
Shows the instance's rotation angle in degrees (*Number* type).

"*7 Scaling h"
Shows the instance's horizontal scaling (*Number* type).

"*8 Scaling v"
Shows the instance's vertical scaling (*Number* type).

"*9 Distortion h"
Shows the horizontal distortion of the instance in degrees (*Number* type).

If two or more attributes in different blocks have the same names, they are only shown once. If these identically named attributes are of different types, the general type (Global or Text) is shown.

This attribute list can have up to 200 attributes and so some of the local attributes may not be available.

This list shows the order in which the attributes should be output in the parts list. The order is the same as the output order, and so this list is not sorted.

Attributes which are not available in all of the affected blocks or instances can also be output. In this case, the values defined as substitute texts or substitute numbers are output.

If this check box is enabled a heading is included as the first line of the parts list. Instead of the content of the attributes this contains their names. As these names are text, a text delimiter is output before and after each name.

Clicking on this button adds the attribute selected in the upper list to the lower list. The lower list can hold up to 200 attributes.

Clicking on this button inserts the attribute selected in the upper list into the lower list before the highlighted attribute. The lower list can hold up to 200 attributes.

Clicking on this button replaces the attribute highlighted in the lower list with the attribute selected in the upper list.

Clicking on this button deletes the highlighted attribute from the lower list.

If this radio button is selected all blocks from the current library of which instances exist in the drawing are transformed to blocks.

If this radio button is selected all the blocks of which instances exist are converted to blocks.

If this radio button is selected all blocks of which instances exist in the drawing are transformed to blocks.

The library whose external blocks should be converted to internal blocks if instances of them exist can be chosen from this list. The list contains all the libraries from which at least one block is instanced in the drawing.

The path to the target folder to which the blocks within the library "* Internal Blocks" should be copied is shown here. It is a good idea to specify a separate folder for each library to avoid name duplication problems.

Clicking on this button calls the Select Block dialog in which the target folder in the "* Internal Blocks" library can be chosen.

If this check box is enabled all affected instances in the drawing are adapted automatically, i.e. the reference is diverted from the original library to the "* Internal Blocks" library.

The name of the library where the block whose instances are to be changed is located is shown in this area.

The name of the library from which the block comes and to which the instances will later refer, is shown in this area.

The name of the blocks whose instances should be altered is shown in this area.

The name of the block to which instances should later refer is shown in this area.

Clicking on this button calls the Select Block dialog in which the block whose instances are to be changed can be chosen.

Clicking on this button calls the Select Block dialog in which the block to which the instances should later refer can be selected.

If this check box is enabled only those instances of the chosen block which are currently selected are altered.

The library where the blocks whose instances are to be altered are chosen from this list. The list contains all the libraries from which at least one block is instanced in the drawing.

The library to which the instances should refer is selected from this list. The block names within the instances remain unchanged, and must therefore be the same as the ones in the original library
All currently recognized libraries (see Libraries) as well as the pseudo-library "* Internal Blocks", which contains blocks and groups, are shown in the list.

If this check box is enabled only those instances from the selected libraries which are currently selected are changed.

If this check box is enabled the library chosen from the list is managed with the help of the block cache. This means that blocks are only loaded into memory if they are actually needed. Especially with large libraries, this can cut down memory use, reducing the load on the program. This check box is only enabled when exactly one library is selected in the list.

If automatic loading is not turned on, and the cache is used, the library can be used (it is included in all the library lists) but its blocks will not be loaded into memory and consequently will not be displayed. If the blocks are required in memory later, either automatic loading can be turned off or the library can be updated (see the "Update" button).

If this check box is enabled blocks from the library selected in the list (if working with the cache) are automatically loaded as soon as they are needed. The check box is only available if exactly one library is selected from the list.

If automatic loading is not turned on, and the cache is used, the library can be used (it is included in all the library lists) but its blocks will not be loaded into memory and consequently will not be displayed. If the blocks are required in memory later, either automatic loading can be turned off or the library can be updated (see the "Update" button).

If this check box is enabled the library selected from the list can be altered; otherwise it cannot be changed. This setting is *not* file level write protection but is exclusively a restriction for the program. The checkbox is only available if exactly one library is selected from the list.

The names of all the currently available (i.e. in memory or at least in the cache) libraries. These libraries are automatically loaded the next time the program is started.
The libraries may have a letter in front of their names. These letters show the operations which will be carried out when the dialog is closed by clicking on "OK". The following letters are used:

'N' stands for "New". A library marked like this is created, i.e. it does not already exist. See the "New" button.

'O' stands for "Open". A library marked like this is loaded into the cache or memory and becomes available. See the "Open" button.

'R' stands for "Remove". A library marked like this is removed from memory and made unavailable. See the "Remove" button.

'U' stands for "Update". A library marked like this is removed from memory and then reloaded. If the cache is used, only those blocks which are actually required are loaded into memory again. See the "Update" button.

'S' stands for "Save". A library marked like this is saved, i.e. all changes carried out in memory are written to the appropriate library file. See the "Save" button.

'+' Indicates that the library has been changed.

Clicking on this button closes the dialog and saves the changes. All the operations shown by letters in the library list are carried out at this time, i.e. the libraries are created, loaded, saved, updated and so on.

Clicking on this button enables a new library to be created. A File Selection Dialog appears from which the library file to be used can be specified.
The Edit Details dialog then appears, in which the details of the library file can be specified. The title of the new library *must* be entered in the "Title" field. This title is used later to identify the library. It can in theory be changed later, but this leads to a lot of extra work as all the drawings which use blocks from this library have to be adapted.
After entering the title of the new library, it appears in the list. It is marked with an 'N' (for "New"). The library itself is created when the dialog is closed by clicking on OK. A newly created library is initially editable and is automatically loaded. The cache is not used.
The library list can hold up to 50 libraries.

Clicking on this button places an existing library in the library list. A File Selection Dialog appears, in which the library file to be included in the list can be specified.
After selecting the library file, its name appears in the list. It is marked with an 'O' (for "Open"). The library itself is loaded when the dialog is closed by clicking on OK. A newly opened library will be automatically loaded and will use the cache, but cannot be edited.
The library list can hold up to 50 libraries.

Clicking on this button marks the library which is currently highlighted in the list with an 'R' (for "Remove"). When the dialog is closed by clicking on OK, the library is removed from memory. Clicking again on the button removes the marking, so that the library is not removed.

If the marked libraries include libraries which have been altered, a question appears, asking whether or not the alterations should be saved before removing the libraries from memory. If this question is answered with "Yes", then all the altered libraries are marked with an 'S' (for "Save"). See the "Save" button.

Clicking on this button enables the details of the library which is currently selected in the list to be edited. This is done from the Edit Details dialog. Only the details of libraries which are already in memory can be edited. The other details can only be viewed.

The title of the library is used to identify it. It can in theory be changed later, but this leads to a lot of extra work as all the drawings which use blocks from this library have to be adapted.

Clicking on this button enables the standard attributes of the library currently selected in the list to be edited. This is done from the Define Attributes dialog. Only the standard attributes of libraries which are already in memory can be altered.

Clicking on this button marks the currently selected libraries in the list with a 'U' (for "Update"). They are removed from memory when you click on "OK" and reloaded immediately. Clicking on this button again removes the marking and the libraries are then not updated.

If a library which uses the cache and is not automatically loaded, then all the required blocks will nonetheless be loaded into memory. Automatic loading is unaffected by this.

Clicking on this button marks all the currently selected libraries from the list with an 'S' (for "Save"). They are saved when the dialog is closed by clicking on "OK", i.e. all the changes are written to the library file. If this button is clicked on again, the markings are removed and the changes are not saved.

If a library which is using the cache is saved, this is done by creating a new library file from the old library file and the blocks in memory. During copying the library file is held twice on the storage medium and the original file is deleted when the save process is over. This can lead to storage capacity problems, especially when working with floppy disks.

Clicking on this button generates a copy of the library selected in the list. A File Selection Dialog appears in which the library file to be copied can be selected. After this, the Edit Details dialog appears in which the details for the library file can be specified. The title of the copied library *must* be entered under "Title". The title of the library is used to identify it. It can in theory be changed later, but this leads to a lot of extra work as all the drawings which use blocks from this library have to be adapted.

The copy is created immediately, i.e. while the dialog is still open. The copied library can therefore be loaded into the list using the "Open" button.

Clicking on this button returns the dialog to the state it was in originally. All the markings are removed, libraries to be created are removed from the list and the library settings are reset to their original values. Alterations to the details and settings of the libraries are not affected by this, i.e. changes remain in effect.

The sort mode for the block window buttons can be selected from this list. The following modes are available:

"No Sort"
: The buttons are not sorted, that is each button keeps its assignment until it is explicitly altered. This setting should always be used when the block window is to be shown on a digitizer.

"Pack Tight"
: The buttons are tightly packed, i.e. gaps are filled by shuffling the buttons.

"Library / Block"
: The buttons are sorted first according to library name and then within libraries according to block names. Gaps are always filled.

"Block / Library"
: The buttons are sorted first according to block names and if there are any, identically named blocks are sorted according to library names as well. Gaps are always filled.

If this check box is enabled the button's number is shown (1.100).

If this check box is enabled the scaling of the block relative to 1.0 is shown.

If this check box is enabled the rotation angle of the block is shown in degrees.

If this check box is enabled the library name of the block is shown.

If this check box is enabled the name of the block including folders is shown.

If this check box is enabled the block itself is shown in a small field inside the button.

Three special elements are shown next to the block:

 A small square in the color of a selected definition point (usually Blue) shows the block's insertion point. This point is positioned with the mouse when the block is inserted.

 A crossed-through rectangle in the selection color (usually Red) shows the frame around the block.

 A hatched rectangle in the color of the page margin (usually Violet) shows the "block frame" which shows the character cell of text blocks. With other blocks this is usually identical to the surrounding frame.

The number of pens available to the plotter is entered in this field. Values between 1 and 8 are allowed, normal values are 2, 6 or 8.

All available pens are shown in this list. The line width and color of the selected pens are shown and can be altered from the fields to the right of the list.

The width of the pen selected from the adjacent list can be changed here. The width is entered in the current length unit.
If the pen width is set to 0.0, the pen is not used. A value of 0.0 should be entered if the corresponding pen is not to be used, e.g. there is no pen in this position in the plotter's pen carousel.

The exact pen width, not the approximate pen width, which should be assigned to the pen in the plotter driver settings

The color of the pen selected in the adjacent list can be selected from this color field. To do this, click the left mouse button in this field to call the Choose Color dialog where the color can be specified.

The actual pen color should be entered here and not the standard color assigned to the pen by the plotter driver settings.

Clicking on this button sets the width of the pen which is currently selected in the adjacent list to 0.0. In addition its color is set to black.
If the pen width is set to 0.0, the pen is not used. A value of 0.0 should be entered if the corresponding pen is not to be used, e.g. there is no pen in this position in the plotter's pen carousel.

The lower limit for using the pens can be entered in this field. The value is relative to the pen width and can be between 0.0 and 1.0. A pen is used to draw lines whose width lies between the two limits for that pen

The upper limit for using the pens can be entered in this field. The value is relative to the pen width and can be between 1.0 and 10^6. A pen is used to draw lines whose width lies between the two limits for that pen

If this check box is enabled pen adaptation is active for output to a printer or plotter.

This check box must be cleared and reset if you wish to use a device other than a plotter for output.

Clicking on this button inserts a line. It is inserted at the intersection closest to the ID point. The line initially has a length of 1 mm and runs from the insertion point to the right.

Clicking on this button converts the element nearest to the ID point to a line. Its start and end points remain unchanged.

Clicking on this button converts the element which lies closest to the ID point in to a Bézier curve. Its start and end points remain unaltered. The two angel points are placed at 1/4 and 3/4 of the distance between the start and end points.
If an element which is already a Bézier curve, the angel points are still moved as described.

Clicking on this button transforms the element nearest to the ID point into a circular arc. Its start and end points remain unchanged. The circular arc's radius is entered numerically.

1. *Enter radius*
   A dialog appears, in which the radius can be entered. After clicking on "OK", the element is converted into a circular arc with the specified radius.

   The appearance of the circular arc is determined by the current arc direction. This is set with the command Toggle Arc Direction (F2), with the F2 key or using the button in the panel. In addition it is influenced by the "direction" in which the curve runs, i.e. which of the two end points was entered first. Finding the correct arc direction for the current curve may be a process of trial and error.

   To get the correct result, it may be necessary to click on the new circular arc and use the "Invert Circular Arc" function to invert it.

If the radius entered is to small to draw a suitable circular arc between the two end points, an error message appears.
If an element which is already a circular arc is clicked on, the radius is still reentered and redrawn as described. This makes it possible to change the radius and arc direction of a circular arc within an existing curve.

Clicking on this button inverts the element nearest to the ID point, if it is a circular arc. Otherwise an error message appears.

Clicking on this button deletes the element which lies nearest to the ID point. The start point of the next element is placed at the start point of the deleted element. Which end of the element is the "start" and which is the "end" depends which "direction" the curve runs in, i.e. which of the two end points was entered first.

If an element should be cut from a curve so that two curves result, the curve must first be cut at one end using the command Edit Curve>Cut. The element can then be deleted.

The message is shown here. It is different from case to case. The size of the dialog adapts itself to the amount of text.

This icon draws attention to something, which is why it is a loud color. It has no other meaning.

Up to three buttons can be visible at once. The text in the buttons and their functions depend on the text above them.
One of the buttons is framed. This button is activated when the ENTER key is pressed. If the ESC key is pressed, then, if there is one, the "Cancel" button is activated.

In this area, the layer and pen assignent of the currently marked object is displayed, if known.

Clicking on this button selects the currently marked object or result. The dialog is closed and the operation is carried out.

Clicking on this button marks the next possible object or result.

Clicking on this button marks the previous possible object or result.

Clicking on this button cancels the selection and closes the dialog.

If this dialog appears during identification, the identification can be carried out again. If this dialog appears during selection of a result, the selection and therefore the whole operation is canceled.

A whole number can be entered into this field. This value lies in a narrow range. If a value outside the range is entered, an error message showing the permitted range appears when the "OK" key is clicked on.

A text, a series of characters, can be entered in this field. The maximum number of characters is usually limited, e.g. to 31 or 63 characters. If you try to enter more characters they will not be accepted.

This field, like every text field, can be used with the clipboard. You can copy a marked area to the clipboard with the key combination CONTROL+INSERT. Text can be pasted from the clipboard into this field using the keyboard combination SHIFT+INSERT.

A floating point number can be entered in this field. This value is normally restricted to a relatively large range, e.g. 10^-10 to 10^10. The value is usually entered in a given unit, which is shown after the field.

In a similar way to coordinate entry, calculations can be carried out in this field. The basic mathematical functions are available. For further information on entering terms, see Coordinate Entry (F8).

The unit for the value is generally shown after the field where floating point numbers are entered.

This unit can be overridden by entering another one in the number entry field, e.g. `30cm`. For further information on entering terms, see Coordinate Entry (F8).

This list shows all buttons of the digitizer's input device. When selecting a button, its assigned functions are displayed in the area right to this list, and can be modified directly.

This list may contain buttons that are not available on the current input device!

If this check box is enabled, the function "Coordinate Tracking" is assigned to the currently selected button in the list, i.e. clicking this button results in a direct coordinate entry to the program, using the current digitizer input device's position.

If more than one function is assigned to a button, all functions will be executed one after another.

If this check box is enabled, the function "Left Mouse Button" is assigned to the currently selected button in the list, i.e. clicking this button has the same effect than clicking the left mouse button. This function will only have effect if the input device is inside the stated screen area.

If more than one function is assigned to a button, all functions will be executed one after another.

If this check box is enabled, the function "Middle Mouse Button" is assigned to the currently selected button in the list, i.e. clicking this button has the same effect than clicking the middle mouse button. This function will only have effect if the input device is inside the stated screen area.

If more than one function is assigned to a button, all functions will be executed one after another.

If this check box is enabled, the function "Right Mouse Button" is assigned to the currently selected button in the list, i.e. clicking this button has the same effect than clicking the right mouse button. This function will only have effect if the input device is inside the stated screen area.

If more than one function is assigned to a button, all functions will be executed one after another.

If this check box is enabled, the function "Select Command on Tablet (Layer 1)" is assigned to the currently selected button in the list, i.e. clicking this button will select the command that is assigned to command layer 1 of the digitizer at the current input device's position. This function will only have effect if the input device is outside the stated screen area.

If more than one function is assigned to a button, all functions will be executed one after another.

If this check box is enabled, the function "Select Command on Tablet (Layer 2)" is assigned to the currently selected button in the list, i.e. clicking this button will select the command that is assigned to command layer 2 of the digitizer at the current input device's position. This function will only have effect if the input device is outside the stated screen area.

If more than one function is assigned to a button, all functions will be executed one after another.

If this check box is enabled, the function "Execute Command" is assigned to the currently selected button in the list, i.e. clicking this button will select the command that was assigned to this button. The currently assigned command is displayed in the area below, and can be altered by pressing the "Browse" button.

If more than one function is assigned to a button, all functions will be executed one after another.

Clicking this button shows the Popup menu, where you can select a command to be assigned to the currently selected button in the list. It will be executed when clicking this button if the "Execute command" check box is enabled.

In this area, the command assigned to the currently selected button in the list is displayed. It will be executed when clicking this button if the "Execute command" check box is enabled.

The width of the digitizer's screen area in 1/10000 inch is entered in this field.

The height of the digitizer's screen area in 1/10000 inch is entered in this field.

The left margin besides the digitizer's screen area in 1/10000 inch is entered in this field.

The bottom margin below the digitizer's screen area in 1/10000 inch is entered in this field.

The x-coordinate of the origin for the coordinate tracking in 1/10000 inch relative to the lower left corner of the digitizer is entered in this field.

The y-coordinate of the origin for the coordinate tracking in 1/10000 inch relative to the lower left corner of the digitizer is entered in this field.

The scaling for the coordinate tracking is entered in this field.

This icon represents the screen layout "One drawing window". If the adjacent radio button is selected, a single drawing window which fills the whole screen is shown.

This icon represents the screen layout "Two drawing windows, side by side". If the adjacent radio button is selected, two equally sized drawing windows are shown next to each other, filling the whole working area.

This icon represents the screen layout "Two drawing windows, above one another". If the adjacent radio button is selected, two equally sized drawing windows are shown one above another, filling the whole working area.

This icon represents the screen layout "One plus two drawing windows, next to one another". If the adjacent radio button is selected, one large drawing window and next to it two small drawing windows are shown, filling the whole working area.

This icon represents the screen layout "One plus two drawing windows, above one another". If the adjacent radio button is selected, one large drawing window is shown with two small drawing windows underneath it, filling the whole working area.

This icon represents the screen layout "Four drawing windows". If the adjacent radio button is selected, for equally sized drawing windows are shown, filling the whole working area.

This icon represents the screen layout "One plus three drawing windows, next to one another". If the adjacent radio button is selected, one large drawing window and three small windows next to it are shown, filling the whole working area.

This icon represents the screen layout "One plus three drawing windows, above one another". If the adjacent radio button is selected, one large drawing window is shown with three small drawing windows underneath it filling the whole working area.

If this check box is enabled, a Ruler is shown in all the drawing windows. Otherwise, just a small field is shown in the upper left of each drawing window.

If this check box is enabled, element buttons are shown is all drawing windows. These buttons can be used to set the current element which is displayed next to the button. In the first drawing window, additional buttons are displayed that can be used to switch the auxiliary windows on and off.

If this check box is enabled, the scroll bars are shown in all drawing windows.

If this check box is enabled, the origin position is displayed in all drawing windows with a symbol.

If this check box is enabled, a dashed line is shown in all the drawing windows to indicate the edge of the page.

If this check box is enabled, ToolTips are active. When moving the mouse above a button, a short textual description of that button is displayed.

If this check box is enabled, the pen list window will be displayed on the screen.

If this check box is enabled, the pen list window position will be saved on exit and restored.

If this check box is enabled, the layer list window will be displayed on the screen.

If this check box is enabled, the layer list window position will be saved on exit and restored.

If this check box is enabled, the block list window will be displayed on the screen.

If this check box is enabled, the block list window position will be saved on exit and restored.

If this check box is enabled, the guide window will be displayed on the screen.

If this check box is enabled, the guide window position will be saved on exit and restored.

If this check box is enabled, the overview window will be displayed on the screen.

If this check box is enabled, the overview window position will be saved on exit and restored.

If this check box is enabled, the overview window will automatically updated when changes occur. Otherwise, it will have be updated manually by moving the mouse above the overview window and pressing SPACE.

If this check box is enabled, the overview window will show a simplified view of the drawing with minimum line widths, only solid lines, no markings, no rulers (even if active in other windows), etc.

If this check box is enabled, the panel will be displayed on the screen.

If this check box is enabled, the panel position will be saved on exit and restored.

If this check box is enabled, the panel will be movable, i.e. can be placed anywhere on the screen.

A value for the font size in the panel can be entered here. The value is given in screen pixels.

If this check box is enabled, the toolbox will be displayed on the screen.

If this check box is enabled, the toolbox position will be saved on exit and restored.

If this check box is enabled, the toolbox will be movable, i.e. can be placed anywhere on the screen.

If this check box is enabled, the property window will be displayed on the screen.

If this check box is enabled, the property window position will be saved on exit and restored.

If this check box is enabled, the property window will be movable, i.e. can be placed anywhere on the screen.

If this check box is enabled, the status line will be displayed on the screen.

If this check box is enabled, the status line position will be saved on exit and restored.

If this check box is enabled, the status line will be movable, i.e. can be placed anywhere on the screen.

A value for the font size in the status line can be entered here. The value is given in screen pixels.

The valid module-code for the module whose name is stated in the caption must be entered here. The format of such a code is "xxxxx-xxxxx-xxxxxxx", where each 'x' represents a number.

This list shows all available display formats for date und time. The current date is used as an example to illustrate those formats.

If this check box is set the search for the specified keywords is case-sensitive.

Use this edit control to enter the keywords (separated by commas).

If you set this radio button only layer names that contain *all* specified keywords will be selected (logical AND).

If you set this radio button each layer name that contains *at least one* specified keyword will be selected (logical OR).

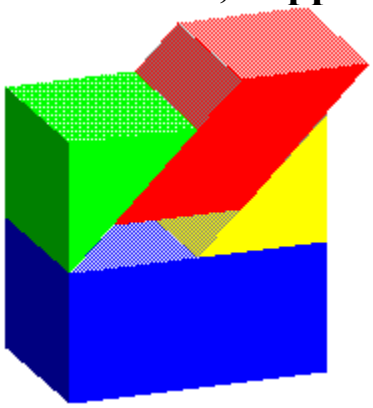
If this check box is set the current selection is retained, i.e. all selections are added to the current selection. Otherwise the current selection is previously removed.

If this check box is set all layer names that match the specified search criteria will be deselected instead of selected. This option is only available if the check box "Retain Current Selection" is set.

After applying this button all layer names that match the specified search criteria will be selected.

If this button is applied all layer names (except "*Standard") will be selected.

Introduction, Support & Information (Information)



The reference contains comprehensive descriptions of all the TommySoftware® CAD/DRAW commands, dialog boxes, and all screen elements. And the reference also gives you an in-depth view of the applications's basic concepts and working methods. Conventions in This Reference contains information on the conventions used in this reference.

There are several ways of working with the reference. If you allowed the installation program to create an icon group for TommySoftware® CAD/DRAW, that group will contain a book icon titled "CAD/DRAW Reference". Double-click this icon. Two windows appear on screen. The larger left-hand window (the "Reference Window") shows the "Introduction, Support & Information" topic (this topic). The smaller right-hand window shows the index to the reference. Any topic chosen in the index window will be displayed in the reference window.

You can also call the reference from within TommySoftware® CAD/DRAW by choosing the command **Contents of Reference** from the **Help** menu.

In addition, you can press the F1 Key at any time in TommySoftware® CAD/DRAW. This displays the reference topic relevant to the current command or dialog in the reference window. As usual, the index window appears to the right. This ensures that you get the required information quickly and easily, without time consuming page turning and searching indexes and tables of contents - just a simple keypress!

Clicking on the "Index" button displays the index window. You can then choose a new topic to be displayed in the reference window.

You can find further help on using TommySoftware® CAD/DRAW in the electronic **CAD/DRAW Tutorial** and the **CAD/DRAW Tour**.

Technical Support and Information

For technical support please send an e-mail to "support@tommysoftware.com". If you have any other questions please send your e-mail to "sales@tommysoftware.com".

On our World Wide Web site (http://www.tommysoftware.com) and in our CompuServe library you can find the latest program versions and upgrades, a collection of import/export filters and converters, the CAD/DRAW Tutorial, the CAD/DRAW Tour, additional documentation, utilities, and current information on our products. And please also try out our special Web offers. Don't miss this opportunity to make the most of the Internet!

If you currently own TommySoftware® CAD/DRAW Level 1 only, you will also find a trial version of TommySoftware® CAD/DRAW 4 Level 2 there for download. This extended application offers numerous additional functions and possibilities. Check it out now!

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All of the vector graphics and tables used in this reference were generated with TommySoftware® CAD/DRAW and either imported via clipboard or as a Windows Metafile into Word for Windows. Markings are output as crosses, in order to generate the point markings quickly and easily.

Conventions in This Reference (Information)

The following conventions are used throughout the reference:

 Very important: You should really read this!

 Note

 Tips & Tricks

Click The word *click* by itself means click the left mouse button once (sometimes also called left-click). Otherwise we say right-click, or double-click.

SHIFT This means the Shift key.

KEY1+KEY2 This means press KEY1 and KEY2 at the same time. For example, ALT+TAB means hold down the ALT key while you press the TAB key.

`x+100+2/3` User input, e.g. text in edit controls.

Italic Strengthening of single words or phrases.

`RELEASE4.T4G`	File name.
"Options"	When referring to elements in dialog boxes the element names are displayed in quotation marks (e.g. "Options" button).
Shape>Modify Objects	This means the command "Modify Objects" in the "Shape" menu of TommySoftware® CAD/DRAW 4.
Object Selection	Underlined, colored text marks a jump to another topic. By left-clicking the underlined text the corresponding topic is displayed. To return to the previous topic apply the "Back" button. If a jump ends with the > symbol (only the case for some entries in the index window), then clicking on that jump displays all subtopics of the clicked topic. This "folding out" of a topic can be reversed by clicking on the " ^ ^ ^ " button at the top of the index window.

If a dialog box is shown in a topic, you can get information on the dialog's elements (e.g. buttons, lists etc.) by left-clicking the relevant part of the picture, see How can I access information on this dialog?

System Requirements (Information)

TommySoftware® CAD/DRAW 4 requires at least the following:

- Intel 80386 processor
- Microsoft Windows 95 or Windows NT 3.51 or higher
- 8 MB RAM available to Windows (not including swap file)
- Graphics card and monitor capable of displaying 640×480 pixels and 16 colors.
- Mouse or other pointing device.

For working with complex drawings and/or imported bitmap graphics, we recommend the following:

- Intel Pentium, 100 MHz processor
- Microsoft Windows NT 3.51
- 16 MB RAM available to Windows (plus a permanent swap file of about the same size)
- Accelerated graphics card and monitor capable of displaying 1024×768 pixels and 256 colors.

Advice for Plotter Output (Information)

Some capabilities of the application (filled surfaces, erasers, clippings, bitmaps, TrueType fonts) cannot be output to or are limited on HPGL or HPGL2 plotters!

This section contains important information for TommySoftware® CAD/DRAW users who wish to output their drawings on an HPGL plotter using the HPPLOT.DRV plotter driver supplied with Windows or obtained from Hewlett Packard. If this applies to you, it is *essential* to read this section completely.

The plotter driver HPPLOT.DRV (in the version 2.30 of March 27, 1990) has many weaknesses. In particular this is evident in the way in which the optimal pen is chosen for the current purpose. When working with differing line widths it is not always possible to see how the choice has been made.

We have carried out extensive tests (using other programs as well) and have come to the conclusion that using this plotter driver will mostly give unusable results.

The resulting output may look correct (the driver, according to Hewlett Packard, is "correct") but the way in which the drawing is produced consists of drawing operations which contradict the plotter's principles. The reason for this lies with a non-optimal choice of pen and its consequences.

We have, because of this odd behavior, added an extra control structure which tells the plotter driver which pen to use. Because this cannot be done in a direct manner (in Windows, a command like "Choose Pen 5" can only be used conditionally), pen choice is governed exclusively by color. The plotter driver is deliberately given false information, to which it reacts in a known manner. This behavior is utilized.

In order to drive your plotter from TommySoftware® CAD/DRAW, proceed as follows.

1. If you have not already done so, install the plotter driver for an HP plotter (e.g. the HP 7550A). Instructions for installing a plotter driver can be found in your Windows documentation or help files.
2. After installing the driver, start its setup dialog. A window with a menu bar should appear.
3. In the Device list, choose the plotter type which you wish to use for output. If you have an HP compatible plotter from another manufacturer, check in its documentation to see which plotter type you should use.
4. At the right of the window appear descriptions of various pens. Note the number of pens which appear here. According to the plotter type, there should be two, six or eight pens.
5. Click on the "Reset" button. The first eight pens should appear in the list with the following colors:

1	Black	p.3
2	Red	p.3
3	Green	p.3
4	Yellow	p.3
5	Blue	p.3
6	Red-Violet	p.3
7	Aqua	p.3
8	Orange	p.3

   If the plotter has less than eight pens, a correspondingly lower number of pens, whose colors should agree with the first of the listed colors, will be shown.
6. If the color allocations do not agree, please alter them accordingly. Do this by choosing a pen from the menu and then choosing the required color from the "Pen Color" menu.
7 In the same way, set the *actual* pen width (or at least an approximate value) and type for each pen in the Type menu. The pen colors *must* remain unchanged even if they do not agree with the actual color

of the pens you are using. Clear the "Draft" check box and enable the "Save" check box. End the plotter setup process by clicking on "OK".

8. Start TommySoftware® CAD/DRAW if it is not already running and choose the Settings>Plotter.command from the "Config" menu. Enter the value obtained in step 4 into the Number field. A maximum of 8 pens can be administered. After this, set the *actual* width and color for each pen.
9. Enable the "Pen Adaptation Active" check box. Note that you *must* clear and then re-enable this check box if you print out on another device in between printing out on the plotter.
10. If you have used TrueType fonts in your drawing, choose the command Settings>Output from the "Config" menu and enable the "TrueType as Surface" check box.
    Note that output of filled surfaces (or characters) takes a long time on a plotter. It is a good idea to use outline fonts where possible. By using the "Fonts" command it is possible to force all subsequently-entered text to be output as outlines.
11. If all the settings altered above are to be saved, choose the Settings>Save As or Settings>Save as Default commands from the "Config" menu.

After carrying out these steps, TommySoftware® CAD/DRAW will have been optimized for plotter output. TommySoftware® CAD/DRAW always selects the optimal pen.
The procedure for choosing a pen ensures that the optimal pen color is always used, the optimal width is chosen next. If a required pen width is not available, then the next pen with a width within a certain tolerance is used. If no such pen is found, then the program attempts to draw the line by making several passes with a narrower pen.

The functionality described here makes deliberate use of weaknesses in the HPPLOT.DRV 2.30 Plotter Driver. We cannot guarantee that it will work with other similar Plotter Drivers. However, if you can carry out the steps described above (especially regarding color allocation), then it will probably be successful.

Although you can use the procedure described here to obtain (in our opinion) better control over an HP plotter, we will keep pressing for improvements to the HPPLOT.DRV plotter driver. If and when a driver becomes available which can perform optimal pen selection on its own, we will let registered users know.

Affected commands:

Print Drawing

Print Section

Using Digitizers (Information)

Working with TommySoftware® CAD/DRAW can be made a lot easier by using a digitizer (also known as "graphics tablet" or "digitizing tablet"). A digitizer enables faster and more accurate input than is possible with a conventional mouse or trackball, and a digitizer can also be used to select commands.

General

All of the major digitizer manufacturers now offer drivers for Windows 95 or Windows NT which enable the digitizer to be used instead of a mouse. Particularly interesting are products which offer an "absolute" mode.

Most digitizers offer a choice of input via a special kind of mouse (normally with at least 4 buttons and a cross hair attached to it) or a pen. Additionally you can often choose between a wired and a wireless model.

Which Input Device?

The question of which input device is best depends on the application. If you often input dimensions from templates, then you should obtain a mouse with an attached cross hair because it can be positioned with great precision. On the other hand, if you do a lot of freehand drawing and enter precise dimensions using the keyboard, then a 3-button pen is preferable as it offers a quicker and more natural way of working.

Which Size?

When working with TommySoftware® CAD/DRAW, a 12"×12" (305×305mm) digitizer is usually sufficient. Smaller digitizers can be used, but they scarcely leave space outside the working area for command fields.

Which Manufacturer?

In principle, you can use any digitizer which has a Windows 95 or Windows NT driver. However, we recommend digitizers that use the WINTAB® driver from LCS/Telegrahpics (e.g .CalComp DrawingBoard and DrawingSlate, Wacom Ultrapad etc.) as they come with an especially powerful driver. These devices also offer a range of extra features, explained below.

Special Functions for WINTAB® Digitizers

In order to use these extra functions, the WINTAB Control Panel must be active. The special functions work with the digitizer and are visible in the TommySoftware® CAD/DRAW menus.

Before using the digitizer in TommySoftware® CAD/DRAW, you should do some preparation. First, setup the WINTAB driver for your digitizer tablet. Set it to absolute coordinate tracking and set the tracking area to the following size and position (depending on the digitizer's size):

8 x 6 inch Digitizer:

Width	3.2 inch
Height	2.4 inch
Left Margin	3.0 inch
Top Margin	depends on tablet size
Right Margin	depends on tablet size

Bottom Margin	1.8 inch

12 x 12 inch Digitizer or larger:

Width	3.2 inch
Height	2.4 inch
Left Margin	5.0 inch
Top Margin	depends on tablet size
Right Margin	depends on tablet size
Bottom Margin	5.8 inch

Next, set the button assigments for your digitizer cursor's buttons in the same dialog. We recommend the following settings:

4-Button Crosshair Cursor:

Button 0	Left Mouse Click
Button 1	Double Left Mouse Click
Button 2	Right Mouse Click
Button 3	Right Mouse Click

3-Button Pen:

Button 0 (tip)	Left Mouse Click
Button 1	Double Left Mouse Click
Button 2	Right Mouse Click

Next, start TommySoftware® CAD/DRAW and assure that the "Configure" menu contains a "Digitizer" sub-menu. This sub-menu will only appear if the digitizer was installed properly. Choose the "Load Assigments" command from that sub-menu and load one of the digitizer assignment files located in the `TSCAD4\SYSTEM` directory. Which file to load depends on the digitizer's size:

8 x 6 inch Digitizer:

`DIGIT06.DIG`

12 x 12 inch Digitizer or larger:

`DIGIT12.DIG`

Finally, you should load and print the command template for your digitizer. This template contains all predefined command fields available. TommySoftware® CAD/DRAW normally uses only a small area in the center of the digitizer to represent the screen. The remaining area is divided into small square fields, each of which is linked to a command. If a field is clicked in, the associated command is carried out as if it had been selected from a menu.

The digitizer templates are normal drawings that can be printed using the "Print Drawing" or the "Print Section" command. Depending on the digitizer's size, load the following drawing file from the `TSCAD4\SYSTEM` directory:

8 x 6 inch Digitizer:

`DIGIT06E.T4G`

12 x 12 inch Digitizer or larger:

`DIGIT12E.T4G`

Print out the drawing with the "Multiple Pages" option enabled, trim the sheets, and fasten them together with sticky tape. Place the digitizer menu under the plastic cover in the lower left corner of your digitizer.

Altering the Command Allocation

If you wish to change or delete the preset command positions, you can do so by using the Define Digitizer Command Field or Delete Digitizer Command Field commands. These commands are called by clicking on the digitizer.

Tracing Templates

A common use for digitizers is tracing templates. If a puck is used, it is possible to trace with an accuracy of up to 1/1000" (± 0.0254mm).

TommySoftware® CAD/DRAW can be used for tracing by reserving a button on the input device for it. Using the Tracking Calibration and Set Tracking Origin commands the digitizer can be set up accurately for this.

This enables dimensions from a template to be transferred accurately and easily to a drawing simply by clicking on the corresponding points.

Affected commands:

 Button and Tracking Options

 Tracking Calibration

 Set Tracking Origin

 Load Assignments

 Save Assignment as

 Information

 Options

 Define Digitizer Command Field

 Delete Digitizer Command Field

Notes for Windows 95 Users (Information)

TommySoftware® CAD/DRAW 4 is a genuine 32 bit application that has been designed to use the full capabilities of Microsoft's modern 32 bit operating systems. Although Windows 95 and Windows NT are not identical in their features, we decided to offer users of Windows NT all capabilities, knowing that Windows 95 users will not be able to use all of them.

As a result, the some capabilities of TommySoftware® CAD/DRAW cannot be fully exploited when running on Windows 95:

 Bitmaps cannot be output if they are distorted or rotated. Windows 95 only allows the output of scaled or reflected bitmaps.

 When using non-solid line patterns, these patterns have to be generated by the application (instead of the driver). To do so, Bezier curves and circular elements will then always be resolved into line sequences.

 When using PostScript fonts by means of the Adobe Type Manager (rather than TrueType fonts), they cannot be displayed outlined, reflected of distorted. Nor can they be resolved or used for gradient fills and clipping.

 The output coordinate range is limited to 16 bits. As a result, the maximum resolution is 254 dpi instead of 2540 dpi.

 Some Windows 95 printer drivers were not able to output reflected bitmaps (even though they should be). Other drivers had heavy problems with clipping surface, especially in connection with bitmaps. Expect to get spool files of 100 MB and more!

Despite these limitations TommySoftware® CAD/DRAW performs also well on Windows 95. But when running TommySoftware® CAD/DRAW on Windows NT all these limitations do not apply and the program's overall performance is considerably better. Therefore we strongly recommend the use of Windows NT!

Affected commands:

 Print Drawing

 Print Section

 Export>Enhanced Metafile (*.EMF)

 Export>Windows Metafile (*.WMF)

 Export>Windows Bitmap (*.BMP)

 Export>Windows Bitmap, Section (*.BMP)

Object Selection (Procedures)

Many commands in TommySoftware® CAD/DRAW require an object selection, where the user selects a group of objects that the current command shall be applied to. You can change the appearance of objects on the screen by means of the command Settings>Screen.

The object selection works similar for all commands. Any number of objects can be chosen. This can be done with various methods, which can be used on their own or in combination with one another:


Objects can be selected by simply clicking on their outlines. If several objects lie at the position clicked on, a small dialog appears at the bottom of the screen, with the help of which the correct object can be picked out.


To select several objects hold down the CTRL key and then click on the outline of each object. If several objects lie at the position clicked on, a small dialog appears at the bottom of the screen, with the help of which the correct object can be picked out. Release the CTRL key to end the selection.


Whole areas can be chosen by holding down the SHIFT key, and specifying two opposite corners of a rectangle. This chooses all the objects which lie wholly within the specified area.

Whether only the objects wholly within the area, or also those which overlap it should be chosen is detemined by the area mode. This is changed with the Shift+F6 Key or the button in the panel.


All the objects in the drawing can be identified by pressing the F10 Key.


If the same group of objects is to be manipulated several times, it can be selected again by pressing the F11 Key.


All the at the moment permanently selected objects can be chosen by pressing the F12 Key.

If several of these choosing methods are to be combined, the CTRL key must be kept held down throughout the entire choosing process. If you then release the CTRL key the current command will be carried out using the selected objects.

All choosing operations are self-inverting. I.e. if an object is chosen twice during a choosing process, it counts as "not chosen". A third selection rechooses it, and so on.

Objects in hidden or frozen layers cannot be selected.


If all the objects in the drawing except two are to be chosen, follow these steps:


Press and hold down the CTRL key


Press the F10 key


Click, one after another, on the two objects not to be chosen

 Release the CTRL key

The object selection is influenced by a selection filter whoose settings can be changed at any time during an object selection by means of the Change Parameters (+ESC) command. The filter remains active until it is explicitely deactivated. For a description of the filter settings, see the chapter Selection Filter.

Point Selection (Procedures)

Some commands in TommySoftware® CAD/DRAW require a point selection, where the user selects a group of definition points that the current command shall be applied to. You can show or hide definition points on the screen by means of the command Settings>Screen.

The point selection works similar for all commands. Any number of points can be chosen. This can be done with various methods, which can be used on their own or in combination with one another:

 Points can be chosen by simply clicking on them.

 To select several points hold down the CTRL key and then click on each point. Release the CTRL key to end the selection.

 Whole areas can be chosen by holding down the SHIFT key, and specifying two opposite corners of a rectangle. This chooses all the points which lie wholly within the specified area.

 All the points in the drawing can be chosen by pressing the F10 Key.

 If the same group of points is to be manipulated several times, it can be selected again by pressing the F11 Key.

If several of these choosing methods are to be combined, the CTRL key must be kept held down throughout the entire choosing process. If you then release the CTRL key the current command will be carried out using the selected points.

All choosing operations are self-inverting. I.e if a point is chosen twice during a choosing process, it counts as "not chosen". A third selection rechooses it, and so on.

Points in hidden or frozen layers cannot be selected.

If all points inside an area plus two additional points are to be chosen, follow these steps:

 Press and hold down the CTRL key

 Press and hold down the SHIFT key

 Enter the two corner points of the area

 Release the SHIFT key

 Click, one after another, on the two additional points

 Release the CTRL key

The point selection is influenced by a selection filter whoose settings can be changed at any time during a point selection by means of the Change Parameters (+ESC) command. The filter remains active until it is explicitely deactivated. For a description of the filter settings, see the chapter Selection Filter.

Change Parameters of Current Command (Procedures)

Most commands of TommySoftware® CAD/DRAW are controlled by a set of parameters that can be edited while the command is active. Some of the commands display a parameter dialog when they are selected from the menu or by any other means, some do not. This depends on whether a command usually requires a new parameter for each usage or not.

In both cases, the command's parameters may be viewed and altered by selecting the Extras>Change Parameters (+ESC) command. This command can also be called by pressing the key combination SHIFT+ESC or by clicking the command's icon in the Panel. Unless you have altered the key assignment, you can also press the 'N' key.

Typical commands that do display a parameter dialog on selection:

- Draw>Line>Parallel, Numerical
- Draw>Line>Angle to Line
- Draw>Polyeder>Standard
- Draw>Circle>Radius - Object - Object
- Geometry>Line>Parallel, Numerical
- Geometry>Line>Angle to Line
- Geometry>Circle>Radius - Object - Object
- Trimming>Round Edge, Outer Arc>Object - Object
- Trimming>Round Edge, Outer Arc>One Edge
- Trimming>Round Edge, Outer Arc>Complete
- Text>Text>Standard
- Text>Text>Reference
- Library>Block>Insert

Typical commands that do not automatically display their parameter dialog, but do still have one:

- Draw>Line>Zigzag Line
- Draw>Freehand Line
- Draw>Hatching>Generated Surface
- Trimming>Trim Surface>Generate Surface
- Text>Dimension Line>Straight
- Text>Dimension Line>Curved
- Text>Dimension>Length, Points
- Text>Dimension>Length, Object
- Text>Dimension>Distance, Object - Point
- Text>Dimension>Distance, Object - Object
- Text>Dimension>Radius, Points
- Text>Dimension>Radius, Object
- Text>Dimension>Diameter, Points
- Text>Dimension>Diameter, Objects
- Text>Dimension>Angle, Points
- Text>Dimension>Angle, Line - Line
- Text>Dimension>Angle, Circle Part
- Text>Dimension>Arc Length, Points
- Text>Dimension>Arc Length, Circle Part
- Text>Dimension>Coordinates
- Text>Calculate>Area
- Text>Calculate>Perimeter / Length
- Library>Position Number>Assign

Most other commands do not own specific parameters, but some allow access to general settings, like the multiline settings (Duplicate / Multiline (F7)) and the Selection Filter.

Main Window (Screen Elements)

The TommySoftware® CAD/DRAW main window serves as a background for the other screen elements. Its main function is to display the menu bar, which provides access to all available commands.

The name of the program and of the current drawing are shown In the main window's title bar. If `(UNTITLED)` is shown here, then the drawing is as yet unnamed.

An asterisk after the drawing name (e.g. `MYFILE*`) indicates that the drawing has been changed since it was last saved.

The size of the Drawing Window changes automatically whenever the size of the main window is altered.

System Menu

The main window has a system menu from which certain settings relating to the main window can be changed. In addition to the usual commands for moving and closing the window and altering its size, the following commands are available:

Display Menu Bar

This command determines whether or not the menu bar is displayed. If the menu entry has a tick against it, the menu bar is displayed.

If the menu bar is not displayed, then pressing the ALT key displays a "floating menu" menu at the current mouse position. It can be used like the normal menu bar.
Turning the menu bar off has the advantage of increasing the available work space on the screen.

If the menu bar is active, the standard menu cannot be used at the same time (see Settings>Mouse Button Usage), as this would exhaust Windows" system resources.

Save Position

If this menu entry has a tick against it, then the position of the main window is saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Drawing Window (Screen Elements)

TommySoftware® CAD/DRAW can work with up to four drawing windows open at the same time. A different view of the same drawing can be displayed in each window. To do this, a separate coordinate system can be allocated to each drawing window (see Coordinate Systems>Edit).

The drawing windows are the program's actual work space. All point entry is carried out in drawing windows. Many commands and keyboard commands are only available when the mouse pointer is within a drawing window.

The mouse position in a drawing window is indicated by a cross hair and not by a pointer. The crosshair makes it easier to recognize horizontal and vertical movement.

The intersection of the crosshair's lines is the current drawing position. The coordinates of this point are shown in the Status Line if it is turned on. In addition, the mouse position can be read from both Rulers.

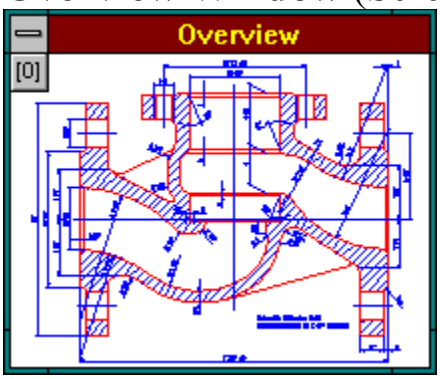
The number of drawing windows is controlled with the command Options>Windows. Several layouts are available which all have one thing in common. All drawing windows in a layout have the same height/width relationship. This has the advantage that the contents of two windows can easily be swapped. (e.g. see Enlarge Window 2).

The size of the drawing windows changes automatically if the size of the main window is altered. Individual drawing windows cannot be resized.

If several drawing windows are visible, then one of them will always be "active". Which window is active can be seen by the way the rulers are displayed. A window is activated by clicking on its rulers with the *left* mouse button. Many commands which modify window-specific settings (e.g. Zoom>Section) always relate to the active window.

If only one drawing window is displayed, it is automatically the active window.

Overview Window (Screen Elements)



Basically, the overview window offers the same possibilities than a "normal" drawing window does. But in addition to that, the overview window can be scaled and placed anywhere on screen, and it can be configured to use a simplified display that leaves out or simplifies some object types. Finally, the automatic window update after each operation can also be disabled in order to save time. For further information on overview window parameters, see Options>Windows.

System Menu

The overview window has a system menu from which certain settings relating to it can be changed. Apart from the usual commands for moving and closing the window the following commands are available:

Rearrange

Pushes the window into one of the four corners of the workspace. The overview window is normally in the lower right corner of the drawing window.

Save Position

If this menu entry has a tick against it, then the overview window position is saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Panel (Screen Elements)

The panel is located on the left of the TommySoftware® CAD/DRAW main window. It is made up of many buttons with which various functions can be activated. In addition, the buttons show the status of the corresponding functions.

In general the setting can be changed by clicking on the button with the left mouse button. Clicking on most of the buttons with with the right mouse button calls a dialog box where you can alter the parameters which apply to the function.

The panel can be turned off to increase the available work space. This is done with the command Options>Windows.

The panel consists of the following elements:

 Snapping functions and Snapping radius (see Snap Modes>Snap Active (F6) and Snap Modes>Radius)

 Snap Mode "Midpoint" (see Snap Modes>Center)

 Snap Mode "Quadrant" (see Snap Modes>Quadrant)

 Snap Mode "Edge" (see Snap Modes>Edge)

 Snap Mode "Corner / End-Point" (see Snap Modes>Corner / End-Point)

 Snap Mode "Intersection" (see Snap Modes>Intersection)

 Snap Mode "Geometry" (see Snap Modes>Geometry)

 Snap Mode "Marking" (see Snap Modes>Marking)

 Snap Mode "Other Point" (see Snap Modes>Other Point)

 Snap Mode "Relative" (see Snap Modes>Relative)

 Orthogonal Mode (see Orthogonal Mode (F5))

 Arc Direction (see Arc Mode>Toggle Arc Direction (F2))

 Arc Mode (see Arc Mode>Toggle Arc Mode (+F2))

 Area Mode (see Toggle Area Mode (+F6))


Duplicate / Multiline (see Duplicate / Multiline (F7))

Geometry Freezing (see Freeze Geometry (+F9))

Geometry Display (see Display Geometry (F9))

Display Grid (see Coordinate Systems>Display Grid Active (F3))

Position Grid (see Coordinate Systems>Position Grid Active (F4))


Current command's icon (see Edit Parameters)


Pen Number 0

Move Pen Display


Pen Numbers 1 to 40 (see Pens>Edit)

Pen Button Display

The buttons with which you select pens display a lot of information simultaneously:

The left color field shows the line color for *screen display*. If the filling mode is either "Filling" or "Eraser", i.e. no outline is drawn, this field will show only a cross.

The right color field shows the fill color for *screen display*. If the filling mode is "Outline", i.e. no filling is drawn, this field will show only a cross.

Between the two color fields, the line width is displayed in millimeters. Below the color fields, the line pattern is displayed. Both the line width and line pattern display are based on the properties for *output*.

System Menu


If moveable, the panel has a system menu from which certain settings relating to it can be changed. Apart from the usual commands for moving and closing the window the following commands are available:

Rearrange

Pushes the window into one of the four corners of the workspace. The panel window is normally in the upper left corner of the drawing window.

Save Position

If this menu entry has a tick against it, then the panel window position is saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Property Window (Screen Elements)

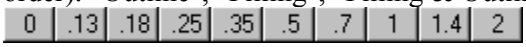
The property window contains several groups of buttons, most of which are used to alter properties. During standard operation, these buttons will directly affect the current properties of pen "*Standard". When drawing without the use of pens, this property window offers the fastest way to select the desired properties.

The following buttons are available in the property windows:

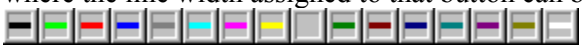
 These five icons represent frequently used commands: Modify Objects, New Drawing, Open Drawing, Save Drawing and Print Drawing.

 This button is used to select a line pattern. When left-clicking it a list of all currently defined line patterns appears.

 These buttons are used to set the filling mode. The available filling modes are (in this order): "Outline", "Filling", "Filling & Outline", "Eraser", "Eraser & Outline".



These buttons are used to select the line width. They do always display the line width in millimeters (independent of the current unit setting)! Clicking with the left mouse button selects the line width, clicking with the right mouse button displays a dialog window where the line width assigned to that button can be edited.

 These buttons are used to select the line color. Clicking with the left mouse button selects the line color, clicking with the right mouse button displays a dialog window where the line color assigned to that button can be edited.

 These buttons are used to select the fill color. Clicking with the left mouse button selects the fill color, clicking with the right mouse button displays a dialog window where the fill color assigned to that button can be edited.

During the execution of the command Modify Objects, the property window works differently. Pressing buttons in the property window will now *directly* alter the properties of the currently selected objects! Since multiple objects with different properties might be selected, the property windows buttons will not be displayed permanently pressed. This would lead to the incorrect assumption that the button state visualizes the objects properties, which is not the case.

System Menu

If moveable, the property window has a system menu from which certain settings relating to it can be changed. Apart from the usual commands for moving and closing the window the following commands are available:

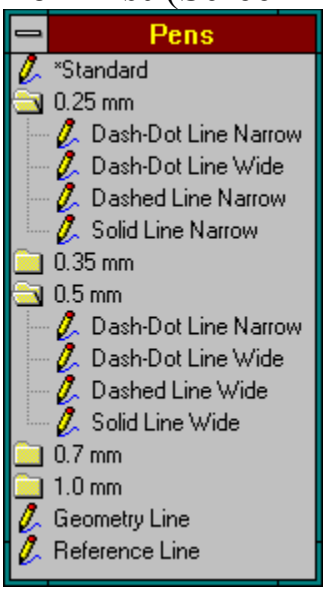
Rearrange

Pushes the window into one of the four corners of the workspace. The property window is normally in the upper left corner of the drawing window.

Save Position

If this menu entry has a tick against it, then the property window position is saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Pen List (Screen Elements)



The pen list shows the complete list of currently defined pens in form of a hierarchical tree-view control.

Entries preceeded by a small pen icon symbolize actual pens, entries preceeded by a small book icon symbolize folders containing further pens. A double-click on any pen makes this pen active, a double-click on a folder open or closes it, respectively.

Clicking with the right mouse button on any pen opens (as usual) the dialog Edit Pens, in which that pen's properties can be edited.

If the command Shape>Modify Objects is active at the time a pen is selected from this list, the chosen pen will be assigned to the currently selected objects. In this case, the active pen is not changed.

System Menu

The pen list has a system menu from which certain settings relating to it can be changed. Apart from the usual commands for moving and closing the window the following commands are available:

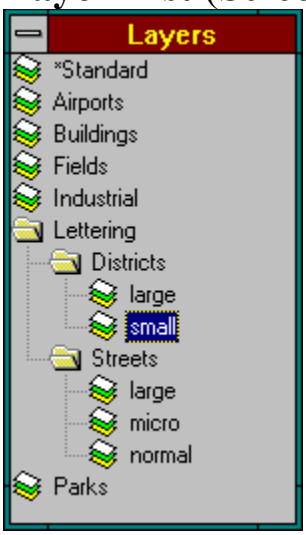
Rearrange

Pushes the window into one of the four corners of the workspace. The pen list is normally in the upper left corner of the drawing window.

Save Position

If this menu entry has a tick against it, then the pen list position is saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Layer List (Screen Elements)



The layer list shows the complete list of currently defined layers in form of a hierarchical tree-view control.

Entries preceeded by a small layer icon symbolize actual layers, entries preceeded by a small book icon symbolize folders containing further layers. A double-click on any layer makes this layer active, a double-click on a folder open or closes it, respectively.

Clicking with the right mouse button on any layer opens (as usual) the dialog Edit Layers, in which that layer's properties can be edited.

If the command Shape>Modify Objects is active at the time a layer is selected from this list, the chosen layer will be assigned to the currently selected objects. In this case, the active layer is not changed.

System Menu

The layer list has a system menu from which certain settings relating to it can be changed. Apart from the usual commands for moving and closing the window the following commands are available:

Rearrange

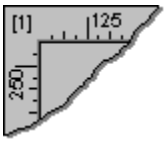
Pushes the window into one of the four corners of the workspace. The layer list is normally in the upper left corner of the drawing window.

Save Position

If this menu entry has a tick against it, then the layer list position is saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Ruler (Screen Elements)

Usually each drawing window has a ruler, which runs along the left and upper edge of the window. It fulfils several purposes.

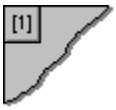


Above all, it serves the purpose of orientation when working with coordinates. You can read from the ruler the current position of the origin and the coordinates of the crosshair.

When working with more than one drawing window, the ruler also shows which window is "active" ist. The ruler of the active window is displayed normally, but the ruler of an inactive window is shown, like unavailable buttons, in a different color.

A drawing window can be acitated by clicking on its ruler with the *left* mouse button. Clicking with the *right* mouse button calls the Define Coordinate Systems dialog, in which the coordinate system for that window can be set up.

In the upper left corner of each window at the rulers' intersection, the number of the window is shown.



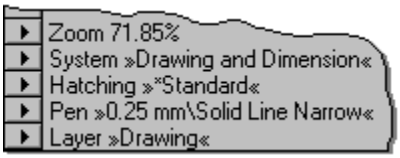
If ruler display is turned off using Options>Windows, a small field in the upper left corner of the window remains active. This field shows which window is active. Clicking on this field with the left or right mouse button has the same effect as clicking on the ruler.

Tne window number is shown in this field.

In the upper right corner of the main drawing window's ruler, a set of window button is display. Each button represents one window out of the following, sorted list:

- Overview Window
- Panel
- Status Line
- Toolbox
- Property Window
- Block List
- Guide Window

Each respective window can be turned on and off by clicking the button with the left mouse button.



In the lower left corner of each drawing window's ruler, the element button are displayed. These button can be used to easily set the current elements. Next to each button, the current setting is displayed in textual form. If a button is pressed using the *left* mouse button, a textual popup menu appears offering a selection of currently defined elements for selection. By clicking a button with the *right* mouse button, a dialog window appears allowing to edit the currently selected element.

The display of the window and element buttons can be turned on and off using the command Options>Windows. The buttons can be displayed even if the ruler is not displayed.

Status Line (Screen Elements)

The status window can be placed anywhere on the screen. It has two functions: to display coordinates and other details for the current operation, and the direct entry of coordinates.

Its appearance depends upon where the mouse pointer is, and at what stage a command is.

The currently active command's icon is always shown at the left of the status window. The expected input is shown in the title bar.

Clicking the mouse button in the status window calls direct coordinate entry (see Coordinate Entry (F8)).

The way in which the status window functions depends on the position of the mouse pointer or crosshair and the stage of command input.

Inside a Drawing Window

Point Entry without Reference Point

"x" and "y"	Current crosshair position
"Used"	Used portion of currently reserved memory in KB (1024 bytes)
"Command"	Description of the current command
"Input"	Description of the currently expected input

Point Entry with Reference Point

"x" and "y"	Current crosshair position
"dx" and "dy"	Difference between the current crosshair position and the position of the reference point
"l"	Distance of the current crosshair position to the reference point
"a"	Angle of the current crosshair position relative to the reference point
"Input"	Description of the currently expected input

Above a button

"Button"	Description of the button under the mouse pointer
"Input"	Description of the currently expected input

During Coordinate Entry

For more information on direct coordinate input, see Coordinate Entry (F8).

System Menu

If moveable, the status line has a system menu from which certain settings relating to it can be changed. Apart from the usual commands for moving and closing the window the following commands are available:

Rearrange

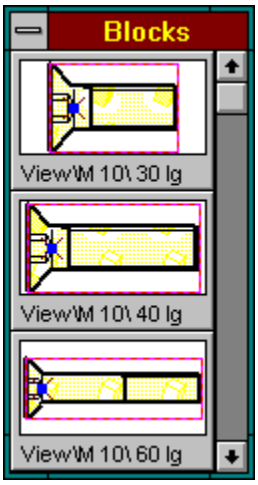
Pushes the window into one of the four corners of the workspace. The status line is normally in the lower area of the drawing window.

Save Position

If this menu entry has a tick against it, then the status line position is saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Block List (Screen Elements)

The block list contains 100 buttons which can have blocks allocated to them. This makes it easy to access frequently used blocks rapidly.



Left-clicking on a button allows the block associated with that button to be placed in the drawing. It hangs from the crosshair and drops into the drawing at the position of the crosshair when the left mouse button is clicked again. This operation can be repeated until the *right* mouse button is clicked, ending the command sequence.

Right-clicking allows the button allocation to be changed. It calls the Insert Block. dialog , in which the library and block name as well as other parameters like scaling and rotation can be specified. In addition, the button's block allocation can be cleared.

The values specified here are saved separately for each button, so the same block may be saved several times with different parameters (for example, rotated by 0° - 90° - 180° - 270°).

The content and form of the buttons in the block list be changed using the command Settings>Block List. The block list settings can be saved together with the other settings using the command Settings>Save As.

The command Save Settings on Exit can also be used to specify whether or not the settings (including the block list assignments) should be saved when the program is exited and loaded when it is restarted. The default settings can also be saved with the command Settings>Save As Default.

System Menu

The block list has a system menu from which certain settings relating to it can be changed. Apart from the usual commands for moving and closing the window the following commands are available:

Rearrange

Pushes the window into one of the four corners of the workspace. The block list is normally in the upper

left corner of the drawing window.

Save Position

If this menu entry has a tick against it, then the block list position is saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Toolbox (Screen Elements)

The toolbox window contains up to 100 buttons which can have program command assigned to them. This enables frequently used commands to be accessed easily.

Clicking briefly on one of the buttons with the *left* mouse button chooses the command associated with that button.

Clicking and holding down (for 1/2 second or more) the left mouse button on a tool window button calls a sub menu. This submenu contains all the commands from the same submenu as the one the button's command is located in. Another command can be chosen from this submenu; this command will be allocated to the button and immediately carried out.
This only works, naturally, enough, for commands which appear in a submenu. If the command appears in a menu, holding down the mouse button has no effect and the command is carried out immediately.

Clicking the *right* mouse button on a tool window button calls the Popup Menu with which the command allocated to the button can be changed.

The tool window allocations can be saved together with the other settings using the command Settings>Save As.

The command Save Settings on Exit can also be used to specify whether or not the settings (including the tool window assignments) should be saved when the program is exited and loaded when it is restarted.
The default settings can also be saved with the command Settings>Save As Default.

System Menu

If moveable, the toolbox has a system menu from which certain settings relating to it can be changed. Apart from the usual commands for moving and closing the window the following commands are available:

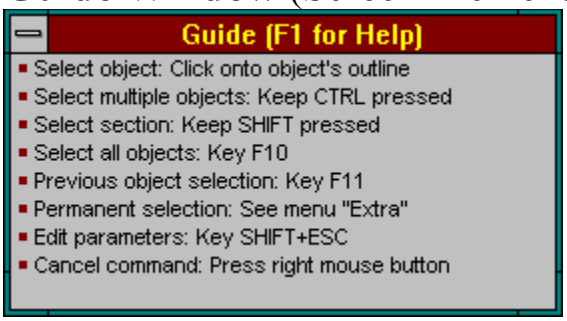
Rearrange

Pushes the window into one of the four corners of the workspace. The toolbox is normally in the upper right corner of the drawing window.

Save Position

If this menu entry has a tick against it, then the toolbox position is saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Guide Window (Screen Elements)



The guide window displays some hints based on the currently active command and the expected point entry. They show you all available options, how you can alter settings that influence the current command (especially the key sequences), and what the application is expecting from you.

System Menu

The guide window has a system menu from which certain settings relating to it can be changed. Apart from the usual commands for moving and closing the window the following commands are available:

Rearrange

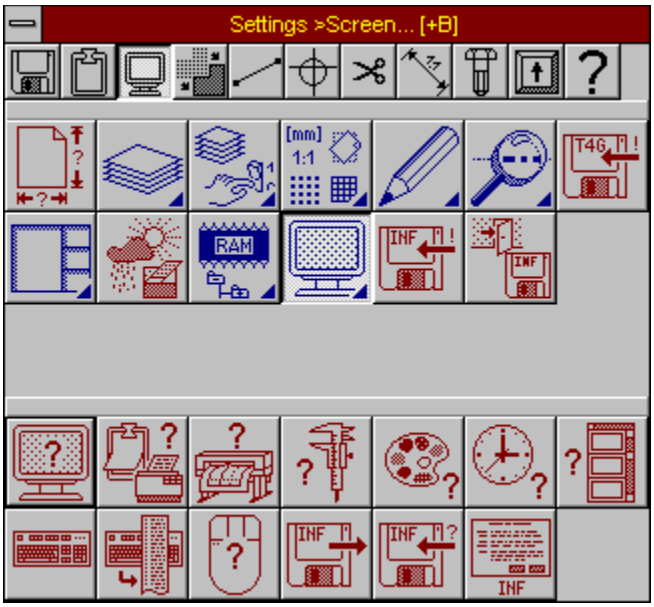
Pushes the window into one of the four corners of the workspace. The guide window is normally in the lower left corner of the drawing window.

Save Position

If this menu entry has a tick against it, then the guide window position is saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Popup Menu (Screen Elements)

The popup menu is a graphical menu which uses icons to represent commands instead of text descriptions. It always appears at the position of the mouse pointer. The popup menu is structured in the same way as the normal menu; it has the same menus, menu entries, and submenus as the normal menu.



Choosing a command can follow the same procedure as using the text menus. Firstly, the menu is chosen from the upper area; the menu entries and submenus from that menu appear in the central area. Choosing a menu entry carries out that command, or in the case of a submenu entry, the commands from the submenu appear in the lower arrea.

Commands which can be chosen directly are shown in red. Submenus are shown in blue. In addition there is a small solid triangle in the lower right corner of submenu icons to help distinguish them. Submenu icons always show the "most important" command from that submenu.

A command, menu or submenu is chosen by clicking on its icon with the left mose button. The choice of command can be cancelled by pressing ESC or clicking the right mouse button in the Popup Menu.

The title bar of the Popup Menu shows the description of the command or menu ander the mouse pointer. Help on this command can be called by pressing the Key F1.

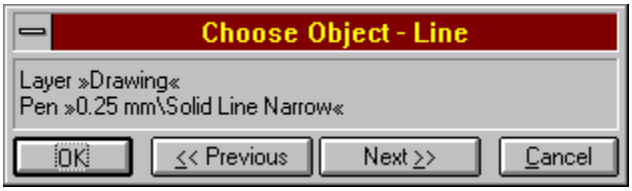
The Popup Menu is a little smarter than the normal menu bar. It remembers which menu or submenu the last command was chosen from and this information can be recalled by double-clicking on the menu or submenu icon.

Line>Standard was the last command chosen from the **Draw** menu. Double-clicking on the **Draw** menu's icon chooses **Line>Standard** again. The same happens if the **Line>** submenu icon is clicked on.

The icons used in the Popup Menu are the same as those used in the Toolbox and to display the current command in the Status Line.

Selection Window (Screen Elements)

How can I access information on this dialog?



The dialog appears whenver an identification was not conclusive or when a calculation produces more than one result and the user must choose which one to use.

The title bar of the dialog always shows the object type of the currently highlighted object, which makes the process of choosing easier.

System Menu

The selection window has a system menu from which certain settings relating to it can be changed. Apart from the usual commands for moving and closing the window the following commands are available:

Rearrange

The window is moved to one of three possible positions in the workspace. It is usually at the bottom of the drawing window.

Save Position

If this menu entry has a tick against it, then the window position is saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Keyboard Assignments (General)

TommySoftware® CAD/DRAW is delivered with predefined keyboard assignments which can be partly changed by the user.

After installing the program, you can open and print out the drawing `KEYBOARD.T4G` from the DRAWING directory. The only font used in the drawing is ARIAL® which is part of every Windows 95 and Windows NT installation.

The following tables show the keyboard assignments, arranged in command groups.

Clicking on the "Standard" button in the Settings>Display Key Assignments dialog restores the default keyboard assignment. This contains a few more key definitions than described above.

In general, only the permanent key assignments are referred to in this help file and the tutorial, in order to avoid possible errors caused by using different key assignments.

Keyboard Assignments for View and Zoom (General)

Command	Possible key combinations	
Move view area one step to the left	←	
Move view area one step to the right	→	
Move view area one step upward	↑	
Move view area one step downward	↓	
Move view area one page to the left	HOME	
Move view area one page to the right	END	
Move view area one page upward	PAGE ↑	
Move view area one page downward	PAGE ↓	
Move center of view area to current mouse position	NUM ×	
Zoom Out (factor 0.5)	NUM -	
Zoom In (factor 2.0)	NUM +	
Zoom Undo	NUM ÷	
Zoom Section		A
Zoom Overview		B
Zoom Page		S
Enlarge Window 2		SHIFT + 2
Enlarge Window 3		SHIFT + 3
Enlarge Window 4		SHIFT + 4
Window refresh	SPACE	

The keys in the right column are the default installation. They can be changed by the user at any time using the command Settings>Change Key Assignment. The defaults can be restored using the "Defaults" button in the dialog called by the command Settings>Display Key Assignments.

Keyboard Assignments for Selection and Identification (General)

Command	Possible key combinations	
Identify all objects	F10	SHIFT + Q
Use previous identification	F11	Q
Identify selected objects	F12	

The keys in the right column are the default installation. They can be changed by the user at any time using the command Settings>Change Key Assignment. The defaults can be restored using the "Defaults" button in the dialog called by the command Settings>Display Key Assignments.

Keyboard Assignments for Clipboard (General)

Command	Possible key combinations	
Cut	SHIFT + DEL STRG + X	
Copy	CTRL + INS CTRL + C	
Paste	SHIFT + DEL CTRL + V	

The keys in the right column are the default installation. They can be changed by the user at any time using the command Settings>Change Key Assignment. The defaults can be restored using the "Defaults" button in the dialog called by the command Settings>Display Key Assignments.

Keyboard Assignments for Settings (General)

Command	Possible key combinations	
Switch Arc Direction	F2	
Toggle Arc Mode	SHIFT + F2	
Display Grid on/off	F3	
Edit Display Grid	SHIFT + F3	
Position Grid on/off	F4	
Edit Position Grid	SHIFT + F4	
Orthogonal Mode on/off	F5	
Edit Orthogonal Mode	SHIFT + F5	
Toggle Area Mode	SHIFT + F6	
Snap on/off (permanently)	F6	
Short-time (de-)activation of Snap	SHIFT	
Duplicate/Multiline on/off (permanently)	F7	
Short-time (de-)activation of Duplicate/Multiline	CONTROL	
Geometry on/off	F9	
Geometry freeze/melt	SHIFT + F9	

The keys in the right column are the default installation. They can be changed by the user at any time using the command Settings>Change Key Assignment. The defaults can be restored using the "Defaults" button in the dialog called by the command Settings>Display Key Assignments.

Keyboard Assignments for Point Entry (General)

Command	Possible key combinations	
Enter point via mouse	LEFT (mouse)	
Enter point via coordinates	F8	ENTER
Undo last point entry	ESC	
Cancel command	RIGHT (mouse)	
Change parameters	SHIFT + ESC	N
Short-time (de-)activation of Snap	SHIFT	
Short-time (de-)activation of Duplicate/Multiline	CONTROL	

The keys in the right column are the default installation. They can be changed by the user at any time using the command Settings>Change Key Assignment. The defaults can be restored using the "Defaults" button in the dialog called by the command Settings>Display Key Assignments.

Keyboard Assignments for Object Identification (Allgemeines)

Command	Possible key combinations	
Identify object by clicking with mouse	LEFT (mouse)	
Identify object via coordinates	F8	ENTER
Undo current identification	ESC	
Cancel command	RIGHT (mouse)	
Change parameters	SHIFT+ ESC	N
Enter area instead of clicking onto single objects	SHIFT	
Identify multiple objects or areas, respectively	CONTROL	

The keys in the right column are the default installation. They can be changed by the user at any time using the command Settings>Change Key Assignment. The defaults can be restored using the "Defaults" button in the dialog called by the command Settings>Display Key Assignments.

Default Assignments (General)

Normal keys, when pressed withhout the SHIFT key, are always used to call active commands or the submenus which contain these commands.

1	Snap Mode "Center"	
2	Snap Mode "Quadrant"	
3	Snap Mode "Edge"	
4	Snap Mode "Corner / End-Point"	
5	Snap Mode "Intersection"	
6	Snap Mode "Geometry"	
7	Snap Mode "Marking"	
8	Snap Mode "Other Point"	
9	Snap Mode "Relative"	
A	Zoom Section	
B	Zoom Overview	
C	Drawing List	
D	Rotate Objects	(Submenu)
E	Edit Text	
F	Trim Area	(Submenu)
G	Tangent	(Submenu)
H	Command Queue	
I	Geometry Line	(Submenu)
J	Edit Properties	
K	Circle	(Submenu)
L	Line	(Submenu)
M	Dimension	(Submenu)
N	Change Parameters	
O	Circular Arc	(Submenu)
P	Polygon	(Submenu)
Q	Previous Identification	
R	Hatching	(Submenu)
S	Zoom Page	
T	Trim Objects	(Submenu)
U	Move Points	(Submenu)
V	Layer Management	(Submenu)
W	Move Objects	(Submenu)
X	Text	(Submenu)
Y	Block	(Submenu)
Z	Zoom	(Submenu)

Bear in mind that complete submenus are assigned to some keys. They allow quick access to a whole range of commands without having to use a key sequence for each individual command.

Keys pressed together with SHIFT are used to alter the system onfiguration and to choose setting-related commands.

SHIFT + 2	Enlarge Window 2
SHIFT + 3	Enlarge Window 3
SHIFT + 4	Enlarge Window 4
SHIFT + A	Settings Output
SHIFT + B	Settings Screen
SHIFT + E	Layer List on/off
SHIFT + F	Settings Windows
SHIFT + G	Settings Accuracy
SHIFT + I	Pen List on/off
SHIFT + J	Property Window on/off
SHIFT + L	Panel on/off
SHIFT + M	Mouse Button Usage

SHIFT + O	Arc Mode (Submenu)
SHIFT + P	Settings Plotter
SHIFT + Q	Identify All Objects
SHIFT + R	Guide on/off
SHIFT + S	Status Line on/off
SHIFT + T	Change Key Assignment
SHIFT + U	Set Origin
SHIFT + V	Overview on/off
SHIFT + W	Toolbox on/off
SHIFT + X	Settings Color
SHIFT + Y	Block List on/off

Command Identifiers (General)

Every command available from the TommySoftware® CAD/DRAW menus has an internal reference number. The commands can also be called using these numbers. To do this, the TAB key is used.

For example, the command Line>Standard has the reference number 501. The command can be called using this key sequence:

```
TAB  5  0  1  TAB
```

This has the same effect as choosing the command from a menu or with a defined key.

This can be used to automate certain operations using Window's Recorder or similar utilities. In addition if you are using a graphics tablet, you may be able to allocate key sequences to certain parts of the tablet. Custom commands are easy to create using this function.

Menu Start Numbers

The command numbers can be worked out from their positions in the menus. Each menu begins on a multiple of 100.

File Menu	100
Edit Menu	200
Configuration Menu	300
Shape Menu	400
Draw Menu	500
Geometry Menu	600
Trim Menu	700
Text Menu	800
Library Menu	900
Extra Menu	1000
Help Menu	1100

Numbering Within a Menu

Each menu entry, submenu and submenu entry has a number. The numbering order corresponds to the order in which the commands and submenus appear. For example, this is a section of the **Text** menu:

Text (Submenu)	800
Standard	801
Frame	802
Reference	803
Resolve	804
Dimension Line (Submenu)	805
Straight	806
Curved	807
Dimension (Submenu)	808
Length, Point - Point	809
...	...
Edit Dimension (Submenu)	830
Rotate	831
Position	832

Update	833
Fonts	834

If a submenu's command number is entered, then it appears on the screen and the required submenu entry can be chosen using the mouse or key combination.

Additional Commands

For command identifiers of additional commands, please refer to the file `TOSO40.H`, which is shipped with the TOSO Interface 4.0 documentation. This files includes a complete list of definitions of all command identifiers.

All the command numbers shown as examples may be changed without warning when the program is updated! They will, however continue to use the same format, and so will be easily amendable.

Registry (General)

TommySoftware® CAD/DRAW saves numerous settings in the registry. Normally, there will be no need to edit these settings manually, since all of them can also be edited from within TommySoftware® CAD/DRAW. Anyway, situations may occur, where the direct inspection of those setting is required. For this reason, the most important entries are explained below.

In order to edit the registry, you will have to start the registration editor. Normally, this editor will not be installed as an icon, so you will have to locate it using the File Manager or the Explorer. The registration editor is located in the `SYSTEM32` subdirectory of the operating system's home directory (e.g. `C:\WINNT\`) and its file name is `REGEDT32.EXE`. Start this file by double-clicking, or create an icon linked to that file if you plan to edit the registry frequently.

Non-User-Dependent Settings (HKEY_LOCAL_MACHINE)

In the registry key HKEY_LOCAL_MACHINE, TommySoftware® CAD/DRAW stores settings are not user-dependent like the EXE's location, hardware information, etc.

HKEY_LOCAL_MACHINE \Software \TommySoftware® \CAD/DRAW 4 \English

`Debugging` — This entry can have values between 0 and 2^31. If the value is greater than 0, the program starts in test mode. In test mode, numerous tests are carried out and detailed error messages appear.

 Only use test mode if asked to do so by our service staff. Its only function is to help locate problems. In test mode, many operations are a lot slower than normal!

`Client` — This entry can be set to 0 or 1. 1 indicates a client version of TommySoftware® CAD/DRAW. This information is needed by the installation program if the uninstall option is used. It has no effect on the functioning of the program.

`PathServer` — This path shows the location of the program and all associated files. On a network installation, this path points to a global directory on the server. TommySoftware® CAD/DRAW looks for all the global files like `RELEASE4.EXE` and `RELEASE4.BLD` in this directory.
This path is set by the installation program. A path name has to end with "\".

`PathLocal` — This entry shows the path to the program's own system files. On a network installation this entry points to a local directory and *not* to the server. TommySoftware® CAD/DRAW looks here for local files such as `RELEASE4.INF`, `RELEASE4.T4G`, etc.
This path is set by the installation program. A path name has to end with "\".

`MonitorWidth`
`MonitorHeight` — Size of the useable area of the monitor in millimeters. Values between 100 and 1000 millimeters are allowed. See Options>General, "Height" and "Width" input fields.

`Version` — This entry shows the version of the currently installed edition of TommySoftware® CAD/DRAW. If you are no longer able to start

TommySoftware® CAD/DRAW, this is the location where you can find the version information required for technical requests.

HKEY_LOCAL_MACHINE \Software \TommySoftware® \CAD/DRAW 4 \English \Modules

This key contains settings of some of the modules that are loaded when TommySoftware® CAD/DRAW is started. This is *not* a complete list, and a module does *not* have to be listed here to be loaded!

User-Dependent Settings (HKEY_CURRENT_USER)

In the registry key HKEY_CURRENT_USER, TommySoftware® CAD/DRAW stores settings are user-dependent like window layout, recent file list, libraries and fonts to load on start-up, etc. In this location, you will also find your personal user ID that is used to create user-dependent setting and element files.

HKEY_CURRENT_USER \Software \TommySoftware® \CAD/DRAW 4 \English

`Init(x)` These entries contain all settings that may not be edited directly. Please do not modify these data blocks, else the application may crash! Anyhow, you may delete all these entries to return the application to its initial state.

`Digitizer(x)` These entries contain the current digitizer settings. They do only exist if a WINTAB driver was detected by the application. Please do not modify these data blocks, else the application may crash! Anyhow, you may delete all these entries to return the application to its initial state.

`UserCodeLevel1` This entry shows the user code of the currently installed Level 1 edition of TommySoftware® CAD/DRAW. If you are no longer able to start TommySoftware® CAD/DRAW, this is the location where you can find the user code required for technical requests.

`UserCodeLevel2` This entry shows the user code of the currently installed Level 2 edition of TommySoftware® CAD/DRAW. If you are no longer able to start TommySoftware® CAD/DRAW, this is the location where you can find the user code required for technical requests.

`UserName` This entry shows the user name for whom the currently installed editions of TommySoftware® CAD/DRAW are registered. If you are no longer able to start TommySoftware® CAD/DRAW, this is the location where you can find the user name required for technical requests.

HKEY_CURRENT_USER \Software \TommySoftware® \CAD/DRAW 4 \English \General

`PathINF 1..8`
`PathT4G 1..8`
`PathT4L 1..8`
`PathEMF 1..8`
`PathWMF 1..8`
`PathBMP 1..8` These entries give the default paths for the various file types with which TommySoftware® CAD/DRAW works. They can be changed with the command Options>Paths.

`MemorySize` Size (in KB) of the memory reserved for internal calculations. Values between

256 and 1024 KB are allowed. See Options>General.

`UndoSteps` Maximum number of undo levels, i.e. the number of commands which can be "taken back". Values between 1 and 100 are allowed. See Options>General.

`PersonalID` This entry states the personal user ID used to identify user-dependent setting and element files. The personal user ID is a six-letter sequence.

`SaveINF` This entry can be set to 0 or 1. 1 indicates that the application's settings shall be saved automatically when exiting the application.

`PersonalINF` This entry can be set to 0 or 1. 1 indicates that the application's settings shall be saved user-dependent by default. The setting file will then be named based on the personal user ID stored in the `PersonalID` entry.

`PersonalT4G` This entry can be set to 0 or 1. 1 indicates that the application's elements shall be saved user-dependent by default. The element file will then be named based on the personal user ID stored in the `PersonalID` entry.

`CoordRangeDRV` This entry determines the maximum coordinate range that TommySoftware® CAD/DRAW will use when sending data to the printer.

`CoordRangeEMF` This entry determines the maximum coordinate range that TommySoftware® CAD/DRAW will use when exporting data to Enhanced Metafiles.

HKEY_CURRENT_USER \Software \TommySoftware® \CAD/DRAW 4 \English \Drawing *X*

`FileName` File name of drawing number *X*. These drawings appear in the Drawing List.

HKEY_CURRENT_USER \Software \TommySoftware® \CAD/DRAW 4 \English \Library *X*

`FileName` File name of library number *X*. This library appears in the library list. See Libraries.

`Caching` If this entry is set to 1, the library numbered *X* from the cache is used. In this case, no blocks are loaded from the library when the program is started, but only details of the library.

`Load` If this entry is set to 1, blocks from the library numbered *X* are loaded when required. If the `Caching` entry is set to 0, then this setting has no effect.

`Save` If this entry is set to 0, then the library numbered *X* cannot be altered.

 The library file itself will *not* be write-protected!

HKEY_CURRENT_USER \Software \TommySoftware® \CAD/DRAW 4 \English \Font *X*

`FileName` File name of font number *X*. This font appears in the font list. See also Fonts.

HKEY_CURRENT_USER \Software \TommySoftware® \CAD/DRAW 4 \English \Modules

This key contains settings of some of the modules that are loaded when TommySoftware® CAD/DRAW is

started. This is *not* a complete list, and a module does *not* have to be listed here to be loaded!

Important File Extensions (General)

When working with TommySoftware® CAD/DRAW the extensions of file names usually have a particular meaning. They also show the file's type. They should be used carefully to avoid confusion by using the same extension for several file types.

At present, TommySoftware® CAD/DRAW and its associated file converters recognize and use the following extensions:

BMP The BMP format is used by Windows bitmap graphics, which can be either black and white or colored.
BMP files can be loaded using the command Import>Windows Bitmap (*.BMP), saved using the commands Export>Windows Bitmap (*.BMP) and Export>Windows Bitmap, Area (*.BMP).

DXF The DXF format was first used by the AutoCAD® CAD system (registered trademark of AutoDesk). It has become a quasi-standard and can be read and/or written by most CAD and graphics programs.
DXF files cannot contain filled surfaces more complicated than rectangles! For this reason, most programs cannot export fillings, or have to laboriosly convert them to rectangles and triangles first.
The DXF-format changes with each new version of AutoCAD®. These changes can be significant. On one hand, they are important for the further development of the program, but on the other makes it almost impossible to keep up to date. However, we always try to deliver the most up to date import filters.
DXF files can loaded the command **File>Import>DXF Drawings (*.DXF)** and saved with the command **File>Export>DXF Drawings (*.DXF)** if the DXF filters are installed.

EMF The EMF format is used to display enhanced Windows vector graphics (enhanced metafiles).
EMF files can be loaded with the command Import>Enhanced Metafile (*.EMF), saved with the command Export>Enhanced Metafile (*.EMF).

MPG MPG 1.0 format is used by TommySoftware® graphics programs. It is relatively compact and easy to read. Complex surfaces including lines and Bézier curves can be modelled. The only restriction is that block definitions cannot be executed.
[Available converters: MPG_T2G + T2G_T3G + T3G_T4G]

T2G TVG 2.0 is used by TommySoftware® CAD/DRAW Professional 2. It is the succesor to the TVG format and includes block definitions and external references. In addition surfaces can now be boanded by circular arcs as well as Bézier curves and lines. In additio T2G files are more compact than TVG files (using about 70% of the space to store the same information).
[Available converters: T2G_T3G + T3G_T4G]

T2L The TVL 2.0 format is also used by TommySoftware® CAD/DRAW Professional 2. Libraries and Fonts are saved in this format. The internal structure and capabilities are the same as T2G.
[Available converters: T2L_T3L + T3L_T4L]

T3G The TVG 3.0 format is used by TommySoftware® Tek Illustrator. It is the successor to the TVG 2.0 format and has been extended by erasers and groups. T3G files are also somewhat smaller than T2G files.
[Available converters: T3G_T4G]

T3L The TVL 3.0 format is also used by TommySoftware® Tek Illustrator. Libraries and Fonts are saved in this format. The internal structure and capabilities are the same as T3G.
[Available converters: T3L_T4L]

T4G The TVG 4.0 format is used by TommySoftware® CAD/DRAW 4. It is the successor to the TVG 3.0 format and has been extended by bitmap objects and clipping surface. TVG 4.0 files are text files and can directly be edited within a text editor.
T4G files can be loaded using the commands Open Drawing and Import>TVG 4.0 (*.T4G), saved with the commands Save Drawing, Save Drawing As and Export>TVG 4.0 (*.T4G).

T4L The TVL 4.0 format is also used by TommySoftware® CAD/DRAW 4. Libraries and Fonts are saved in this format. The internal structure and capabilities are the same as TVG 4.0.
T4L files can be loaded using the commands Libraries or Fonts.

TVL The TVL 1.0 format is used by TommySoftware® graphics programs to store libraries and fonts. Internally, it is the same as MPG.
[Available converters: TVL_T2L + T2L_T3L + T3L_T4L]

VEK The VEK 1.0 format is used by MegaPaint® ST 2.3 and later on the Atari ST. Like MPG format, it can contain complex surfaces and graphics.
[Available converters: VEK_MPG + MPG_T2G + T2G_T3G + T3G_T4G]

WMF The WMF format is used to display Windows vector graphics (metafiles).
They can be loaded with the command Import>Windows Metafile (*.WMF), saved with the command Export>Windows Metafile (*.WMF).

Not all these converters are included with TommySoftware® CAD/DRAW. If you require a converter not included in the basic package visit our CompuServe library (GO TOSOENG) or our world wide web site (http://www.tommysoftware.com). There you can find a collection of our current converters and import/export filters.

Layers, Pens, and Transmission (In-Depth View)

Each object, e.g. a circle, has its own property set consisting of line width, line pattern, line color, fill color, and fill mode. These properties determine the appearance of the object on the screen and later on the paper.

In addition, each object contains a reference to a layer and a pen, i.e. it lies in a certain layer and uses a certain pen. When the program determines how to draw an object, this not only depends on the object's own properties, but also on the properties transmitted by the assigned layer and the properties specified for the assigned pen.

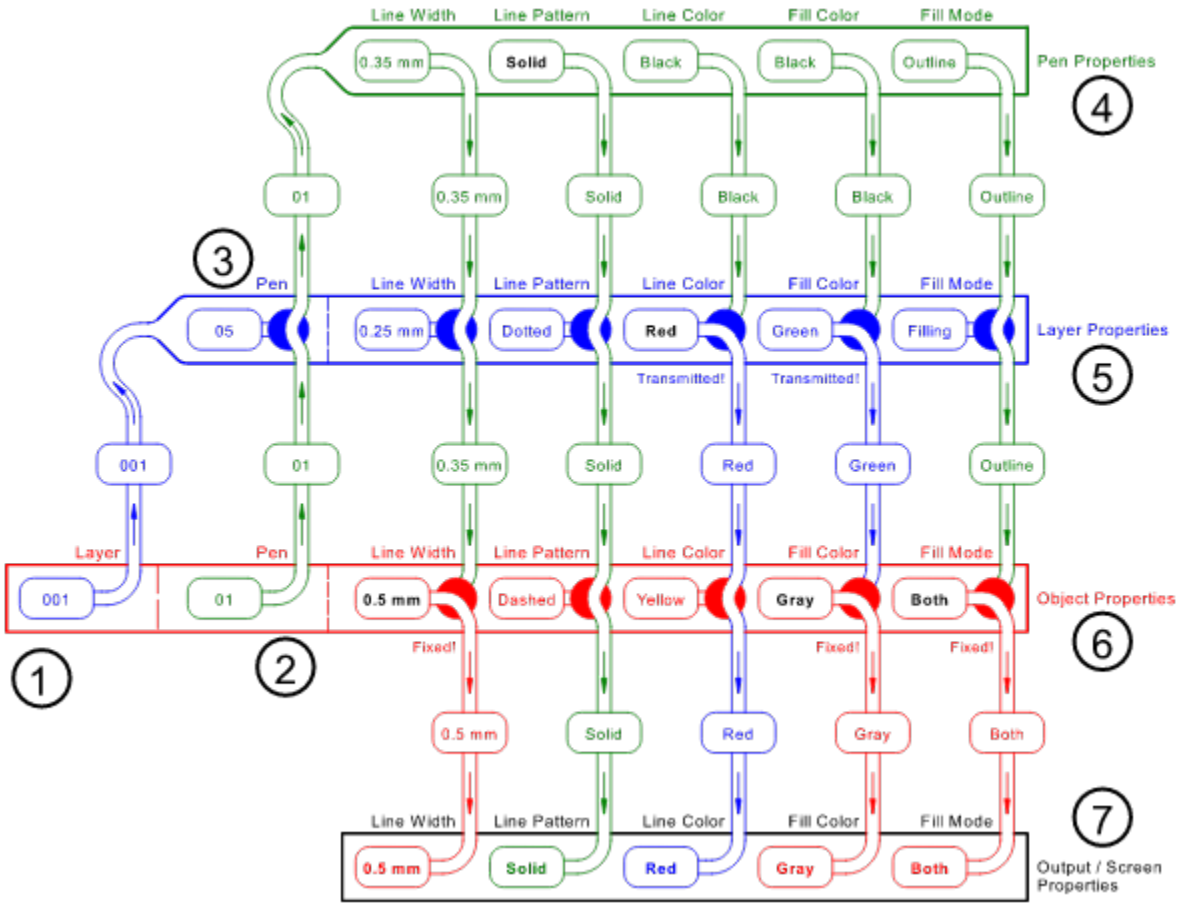
Both the property set of the layer and the one of the pen do not permanently alter the properties of the object itself, but do only superpose them temporarily. As soon as the layer or pen reference is changed, the object's appearance will change accordingly - without neccessarily changing its properties.

Usually, layers do not tramsmit any properties, so they do not influence an object's appearance. In this case, the object's appearence is only determined by the pen it uses, i.e. by the pen's properties (see Configure>Pens>Edit). Those pen properties may be different for screen display and printer output.

If a layer property is transmitted (see Configure>Layers>Edit), it superposes the corresponding pen property. It is therefore decisive for the appearance of the object. Layer properties may also be different for screen display and printer output.

Both types of property transmission (from a layer or a pen) can be refused by the object by fixing one or multiple of its properties. These fixed properties cannot be superposed by any transmission and will always remain unchanged. Fixing a property will always effect both screen display and printer output.

To change the properties of a single or multiple objects use the command Shape>Edit Properties.



This graphic illustrates the process used to determine the properties of an object. Each step is shown in the graphic by the relevant number in a circle:

1) First of all, the program determines which layer the object belongs to.

2) Next, the program gets the information about which pen is used by the object.

3) Then the program checks whether or not layer 1 transmits a pen. In such a case the transmitted pen would become the current pen replacing the pen determined in step 2.

4) All pen properties are initially taken from the pen determined in step 2.

5) All properties transmitted by the layer override the corresponding pen properties, as here the line color and the fill color.

6) The program checks each of the object's own properties to see if it is fixed. Fixed object properties are always passed on directly to the output / screen properties. In the diagram line width, fill color, and fill mode are replaced by fixed object properties.

7) Finally the properties determined in step 1 to 6 are used for output to plotter / display on screen.

When drawing an object, it is always assigned to the current layer and the current pen. You can change the current layer by clicking the corresponding button in the Panel or by selecting the command

Configure>Layers>List. You can also select a new current pen from the Panel or by choosing the command Configure>Pens>List.

For some object types, e.g. texts or dimensions, you can optionally specify a default layer (Configure>Layers>Defaults) and a default pen (Configure>Pens>Defaults) which then will be used instead of the current layer or pen respectively when creating such an object.

If you want to assign an object to a different layer or pen later, you can do so by menas of the Shape>Edit Properties command.

As mentioned above, layers are usually not used to change the appearance of objects by property transmission, but to structure a drawing logically. The Edit Layers dialog allows you to show or hide layers, to enable or disable them for snapping or modification, etc. If you create layers with meaningful names and consequently assign all newly created objects to these layers, you will find that this feature will greatly simplify your work.

The current pen and layer settings are part of the drawing and are saved with it. So that altered settings are available when the program is restarted, save them using the command Configuration>Save Elements as Standard. In addition to all pens and layers the current page format, pen and layer defaults, the coordinate systems, and the line patterns will be stored in the file created.

Coordinate Systems, Scales, Grids, etc. (In-Depth View)

Coordinate Systems

The coordinate systems determine the parameters which the program uses to process drawing data. The coordinate system determines whether you wish to work in a cartesian, isometric or dimetric system. By entering an angle of rotation the orientation of the coordinate system can be displayed. In addition, you can specify measurement units for lengths and angles as well as the type on number, floating point decimals and fractions. The scale is saved with the coordinate system and the position and display grids can be set here. You can set the coordinate system parameters via a dialog called with the command Configure>Coordinate Systems>Edit.

Each drawing can have several coordinate systems defined within it. This can be useful if you have several drawing windows and wish to use a different coordinate system with each one, perform dimensioning in a different coordinate system to that you are drawing in, or use objects with different scales in the same drawing.

To change the current coordinate system apply the corresponding button in the Ruler or select the command Configure>Coordinate System>List.

Scales and Units

The most important components of a coordinate system are the scale and the units. The scale determines the "real" size of an object, i.e. how large a drawn object is supposed to be in the model world. Using a scale of 1:20 means that an object whose length is 10 cm on the paper is supposed to have a length of 200 cm in reality.

Above all, scales are influencing dimensions, i.e. dimension statements within a drawing. Dimension do always show the "real" size of an object based on the scale, and not its size on the paper. This is why each dimension is directly linked to a coordinate system, out if which is determines the scaling information. In addition, it uses the desired length and angle units stored in the coordinate system for the dimension.

If dimensions are to be linked to a specific coordinate system, this can either be done in advance by means of the Lettering > Dimension Parameters command, or afterwards by means of the Shape > Edit Properties command.

Anyway, the scale and the units do not only influence dimensions placed in the drawing, but all types of measurement, both during user entry (e.g. numerical input of a length) and during screen output (e.g. the coordinate display in the Status Line).

If you do not like the default measurement units of a coordinate system during input, you can choose different ones (e.g. cm instead of mm) by typing the abbreviation for that unit after the figure. You can use this procedure in all dialogs. You can combine different units in the same calculation (see Extra > Coordinate Input (F8)).

Grids

Another important component of coordinate systems are grids, which are divided into position grid and display grid.
The *position grid* is an invisible, regular series of points which the crosshair moves along. The points are

always the same distance in one direction from another. The horizontal and vertical divisions can be different. The crosshair 'snaps' from point to point. This ensures that you can only move the crosshair in multiples of the specified grid interval.
The *display grid* shows the grid as small dots on the screen. It can be set separately from the position grid. You can choose a larger display grid than position grid. This speeds up screen redraws considerably and makes for a better overview as not so many points obstruct your view of the objects. If the zoom level is too small, that is if the display grid is too small and the number of points to be shown too large, the display grid is first automatically enlarged and then turned off. As soon as you revert to a level at which the grid can be displayed, it is turned on again.
Both grids are valid for the currently active coordinate system and can be defined as part of it.

The first point of each grid is at the origin. In order to be able to work with a common starting point for the grid in different windows using different coordinate systems, the origin should be moved so that all the grids can be used effectively. To do this, choose the command Configuration>Coordinate Systems>Set Origin.

Page Formats

Before starting a new drawing, it is often necessary to specify the page format for the drawing. To do this, choose Configuration>Page Formats. This calls a dialog where you can either choose a standard page format or a custom page format. The page orientation is also specified in this dialog.

Snapping, Duplicate, and other Options (In-Depth View)

Snapping

In technical drawing it is not enough to determine the position of points approximately. Most points have to be specified very precisely, which is not possible by hand. TommySoftware® CAD/DRAW offers mathematical aids for positioning objects. During drawing it is usual to orient yourself to preexisting construction points, such as edges, corners and intersections. To make use of the points within these objects, there are several snapping modes.

If the snap function is active during point entry, then the position of the point will be calculated automatically if it lies within the snap radius. The snap radius is an area around the center of the crosshair. It can be specified in a dialog called by choosing Extra>Snap Modes>Snap Radius.

If several points lie within the snap radius determined by the current snap radius settings, the crosshair will always be placed on the nearest one.

The snap function is permanently turned on or off with the F6 key. In addition, snap mode can be briefly turned on with the SHIFT key. Using buttons in the Panel, single snapping modes can be toggled.

Duplicate

Every alteration destroys the original object information. The original object information can only be restored by using the UNDO function. To avoid these problems, TommySoftware® CAD/DRAW has a duplicate function. It is permently activated or deactivated by pressing the F7 key. The function can be temporarily turned on or off during a command by pressing and holding down the CTRL key. The current setting is shown in the panel and is reversed by holding down the CTRL key.

The duplicate function is a good substitute for the "Copy" command, because it can be combined with every command. The classic "Copy" command can be called by choosing Shape>Move Object>Standard and keeping CTRL pressed while placing the objects (i.e. entering the destination point). The duplicate function works similarily with most commands that alter objects.

Multiline 

When creating new objects (e.g. drawing a line or circle), the "multiline function" can be used. It determines whether several parallel or concentric objects are created at once (depending on the multiline parameters) or not.

The multiline function is permently activated or deactivated by pressing the F7 key. The function can be temporarily turned on or off during a command by pressing and holding down the CTRL key. The current setting is shown in the panel and is reversed by holding down the CTRL key.

To edit the multiline parameters, use the Change Parameters (+ESC) command during a command's execution.

General Options

The general program options and settings can be edited by selecting the command Configure>Options>Windows or Configure>Settings>Screen respectively.

Libraries, Blocks, and Instances (In-Depth View)

Libraries are organized in a hierarchical structure similar to the file system. A library can contain any number of folders and a folder can hold any number of blocks. Folders are used to group related blocks together.

A block is a collection of objects, like for example lines, rectangles, circles, or texts. Blocks can be used to show screws, furniture, components for local area networks (LANs) etc.

There are a lot of predefined libraries for various areas of application available, e.g. mechanical engineering, electrical engineering, interior design, and others. And of course you can also create your own libraries.

The library / block concept not only allows you to efficiently organize and access frequently used graphical data it also saves a lot of memory. When inserting a block in the drawing you actually don't insert the block's object data but a reference to that block, the so-called instance. I.e. the program inserts only the block name, the library name, and some display parameters (e.g. position, scaling, rotation). Whereas the actual graphical data continues to be stored only in the block definition in the library.

When drawing a block the program uses the block and library name stored in the instance to locate the block definition in the library. The program then uses the graphical data in the block definition and the display parameters stored in the instance to draw the block. Because normally an instance requires much less storage than the actual block data, this concept dramatically reduces memory consumption.

A library is an external file (`*.T4L`) which is independent from the drawing. To use a block (Library>Block>Insert) of a specific library in a drawing you first have to load that library into CAD/DRAW (Library>Libraries). Blocks located in such a library are called *external blocks*. Blocks can also be located in a so-called pseudo-library named "* Internal Blocks". The pseudo-library "* Internal Blocks" is not an external file but it is located in the drawing. Consequently the blocks of the pseudo-library are called *internal blocks*.

When creating your own libraries (Library>Block>Create (Insertion Point)) you should consider the advantages and disadvantages of both block types. External libraries can easily be used by several people in different drawings. If for example all people in company use the same libraries located on a central server this helps to standardize drawings and also makes it easy to update them. Because if you replace a library with an updated version (of course the block names must remain unchanged) this update automatically will have an effect on each drawing which contains instances of blocks of that updated library simply by reloading the drawing.

Of course if you want to pass on your drawing files to another company or a client you either also have to pass on the used library files or you have to convert the external blocks into internal blocks using the command Library>Convert External Blocks. Alternatively you could also use the command Library>Resolve Instances to replace each instance with the objects that make up the respective external block. But if the drawing contains a lot of instances this will significantly increase the drawing's size.

Generally you should always use external blocks if you plan to use the blocks not only in the current project or if the blocks must be accessed also by others. On the other hand if you know that a certain block makes only sense in a specific drawing then you should make him an internal block.

There is a special kind of internal blocks the so-called groups. To create a group use the command Shape>Group>Create Group. This command automatically creates an internal block stored in the "#G"

folder of the pseudo-library "* Internal Blocks" from all currently selected objects.

Blocks can also contain attributes. An attribute can be a text or a number. For example, if you have a furniture library you can use the command Library>Block>Edit to add a price attribute to every piece of furniture. After finishing the drawing you then can use the command Library>Generate Parts List to generate a list of all pieces and their prices. Use the command Shape>Edit Text to edit the attributes of an instance.

File Format (TVG 4.0) (In-Depth View)

The following list shows all file elements available in a TVG 4.0 file of the type "drawing":


Toolbox


Block List


Key Assignment


Windows


Defaults


General Settings


Module Settings


Page Format


Custom Colors


Hatch Types


Line Sequences


Coordinate Systems


Pens


Line Patterns


Layers


Internal Blocks, Groups, Position Numbers


Objects and Instances

TVG 4.0 files of the type "library" or "font", respectively, do only contain some of the elements listed above.

When opening or saving a file the current File Options determine which of these elements are actually loaded or saved respectively. If you want to change the file elements to be loaded or saved only temporarily for the current file operation you can also apply the "Options" button in the File Selection dialog.

If you want to import certain file elements into the current drawing from another drawing then choose the command File>Import>TVG 4.0 (*.T4G). By applying the "Options" button in the File Selection dialog you can specify which file elements are to be imported and whether the existing file elements are to be replaced or to be merged with the imported file elements. To export certain file elements use the command File>Export>TVG 4.0 (*.T4G).

To import or export only the file element "Layers" you can also apply the "Layer Status" button in the

Edit Layers dialog.

All TVG 4.0 drawings have the file extension `.T4G`, TVG 4.0 libraries and fonts have the file extension `.T4L`. These files are pure ANSI files, i.e. you can load them into any text editor, like for example Notepad. This can be very helpful if you want to search and replace some character strings in text objects, layer names, pen names, etc. If you want to know more about the TVG 4.0 file format refer to the TommySoftware® TVG 4.0 Documentation (TVG40.HLP).

Module Concept (Toso Interface 4.0) (In-Depth View)

All TommySoftware® 32bit applications include a powerful software interface (called Toso Interface - TommySoftware® Open Software Interface) that allows the integration of external modules. These modules can either be import filters, export filters or command extensions in any form. See TommySoftware® Toso 4.0 Documentation (TOSO40.HLP) for details.

The Toso Interface offers an easy way to create powerful extensions to TommySoftware® applications. The interface was designed to make it as easy as possible to create modules that fit perfectly into the application's environment. Usually, the user won't realize any substantial difference between using an internal command provided by the application or using an external command provided by a module or filter.

Import and export filters can be created with only basic knowledge of the external file format (see TommySoftware® TVG 4.0 Documentation (TVG40.HLP) for documentation on that file format) and some knowledge about geometrical calculations. You will find that especially creating export filters is very easy - if required, the application does all calculations for you and passes simple lines to the export filter.

There are three sample modules including source code and everything else you need to create a module that come with CAD/DRAW: `IMPORT_.DLL`, `EXPORT_.DLL`, and `TSAMPLE_.DLL`. You can use these samples as a basis for your own module development.

After the modules' installation, you can also select the corresponding menu items in CAD/DRAW to see how the modules work: **File>Import>Coordinates (*.TXT)**, **File>Export>Coordinates (*.TXT)**, and **Draw>Sample Module>Star**. For more information on these modules highlight the relevant menu item and press F1.