## Help index for TOMMYSOFTWARE® CAD/DRAW

The Help index lists all available help subjects for TOMMYSOFTWARE® CAD/DRAW. Use the scroll bar to view entries that are not visible.

To learn how to use **Help**, press F1 or choose **Using help** from the Help menu.

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## SHAREWARE

TOMMYSOFTWARE® CAD/DRAW is a Shareware program. Shareware needs confidence. So please, remember to get registered (see Register). Shareware is extraordinary inexpensive. Because of this it is extremely important for us that everybody who uses the program gets registered. The development of TOMMYSOFTWARE® CAD/DRAW was very expensive and time-consuming and who wants to work for nothing?

So you have an influence on how the future will look like: Expensive software from market dominating companies or (just as professional) software for a good price.

## Fixed keyboard functions and mouse

The function keys and some other keys have been permanently assigned to certain commands: pressing one of those keys carries out the command. In addition to that you can assign the numeric and alphabetic keys to any commands you like. For information see assigning commands to keys.

The mouse is the main input device in TOMMYSOFTWARE® CAD/DRAW. Using the mouse moves the cursor on the screen. While drawing, the cursor appears as a crosshair; while choosing a command, it looks like an arrow; and when the program is not ready for entry (e.g. while loading, saving or carring out intensive calculations), it has the shape of an hourglass. During redraw the cursor disappears for a moment.

### Fixed keyboard functions

| | |
|---|---|
| Alt+Bksp | Undo the last operation. |
| Ctrl+Bksp | Undo the undo (redo). |
| Ctrl+Ins | Copy all selected objects to the clipboard. |
| Shift+Ins | Paste all objects from the clipboard into the drawing. |
| Del | Delete all selected objects. |
| Esc | Reverse the last point entered.<br>If more than one point needs to be entered for an operation, the point last entered can be reversed with Esc and then entered again.The point to be entered is displayed in the status window. |
| F1 | Context-sensitive help function.<br>If F1 is pressed while a menu is open and a command is high-lighted, the help text for this command appears. Otherwise the help text for the current function appears - this however is not possible unless a dialog box is open or a command is being carried out. |
| F2<br>F3 | X-direction constraints on/off.<br>Y-direction constraints on/off.<br>Using the constraints has the effect that the crosshair can be moved only along straight lines in the unconstrained direction. If for example the X-direction is constrained then the cursor can only be moved vertically. |
| F4 | Grid on/off<br>Activating the grid has the effect, that the crosshair can only be moved over the drawing area on certain points. These points are determined by the current grid setting. The current state is displayed in the status window and in the panel. |
| F5/Space | Redraws the screen |
| F6 | Catch function on/off<br>If the Catch/Trapping function is on while choosing points, pressing the left mouse button means choosing the definition point nearest to current position of the cursor. If the Catch function is active, this is |

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| | displayed by four small lines through the cursor and a character in the status window. Additionally an icon in the panel displays the current state. |
| F7 | Duplicate on/off<br>If the duplicate is on, the original objects remain unchanged during manipulation. If duplicate is on, this is shown by a plus-sign on the cursor and a character in the status window. Additionally an icon in the panel displays the current state. |
| F8 | Direct point entry.<br>Depending on the currently active command, a dialog box appears which enables the entry of coordinates. See entering points and objects. |
| F9 | Activating/deactivating the popup menu. |
| F10 | Activating the menu bar.<br>See choosing commands. |
| F11/Bksp | Pre-selecting the nearest object. |
| Shift+F11/Bksp | Undoing the pre-selection. |
| F12/Return | Selecting the pre-selected object.<br>See selection. |

Numeric keypad

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|---|---|
| + | Zooming in on the cursor position. |
| - | Zooming out from the cursor position. |
| ÷ | Memorizing the current zoom. |
| × | Restoring the last zoom memorized.<br>See screen display. |
| Arrow keys<br>Side keys | Moving the window in the corresponding direction. |

Some of the commands mentioned above (grid, catch, duplicate) are also available through the panel.

**Mouse**

The mouse moves the cursor on the screen.

| | |
|---|---|
| Left button | Inserting points, selection, choosing commands. |
| Right button | Canceling an entry or calling up the Popup menu.See procedures and commands. |

## Commands

This section explains how to handle commands and dialog boxes.

Choosing commands
The popup menu
Assigning commands to keys
Dialog boxes

## Menus

This section lists all TOMMYSOFTWARE® CAD/DRAW commands and explains them. The order corresponds to the order of the commands in the menu.

File
Edit
View
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## Procedures

This section contains general information on and explanations of working procedures.

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## Choosing commands

There are basically three ways of choosing a command: from the menu, from the popup menu or from the keyboard. Using the keyboard can either be done through a fixed function key, or through a key which the user has assigned a function to (see assigning commands to keys).

The currently active command is displayed in the status window. It remains active until a new command is chosen.

## Popup menu

The Popup menu is a window, consisting of icons and layer boxes. An icon is a graphical symbol whith a certain command assigned to it. You activate it either by pressing the right mouse button (provided the insertion of an object has not already been started, in which case the insertion will be cancelled), or by pressing F9.

If you move the cursor over one of the layer boxes, the icons of that layer are displayed and the frame around the previously active layer box moves to the currently active one. When you move the cursor over an icon, it will be inverted and at the same time the corresponding command text is displayed in the status window.

Pressing the left mouse button when the cursor is over an icon, means choosing the corresponding command and leaving the popup menu. Pressing the right mouse button means leaving the popup menu without choosing a command.

You can configurate the popup menu according to your own wishes. To do that, you first choose the change popup menu command and then the command you wish to add to the popup menu. In the then displayed popup menu you choose a position where you want that command icon to appear. This completes the procedure and now the command can also be chosen from the popup menu.

The assignments to the popup menu can be saved or reloaded together with assignments to the keyboard. This enables you to create special assignments to the popup menu and to the keyboard to be used for special applications.

## Assigning commands to keys

The keys "A"..."Z" or "1"..."0" can be assigned to commands. Pressing one of those keys will then choose the command it has been assigned to. Each of these keys can have up to three assignments (key, shift+key, ctrl+key).

In order to assign a key to a command, you first choose the change shortcuts command, then the command you want to be assigned to the keyboard. A dialog box then appears, showing which keys can be used. After selecting the key you want, it will be assigned to the command. By using the option buttons the command can be directly assigned to the key (normal), to a shift+key or to a command+key. If the "Info mode" check box is active, then pressing a key displays the current assignment to that key without any change to the assignment.

The keyboard assignments can be saved or reloaded together with the popup menu assignments. This enables you to create special assignments to the popup menu and to the keyboard, to be used for special applications.

## Dialog boxes

If you leave a dialog box by clicking the "Cancel" button, the activated settings will not be applied, nor will they be memorized. In every other case, clicking a button either means that a special function from the dialog box will be carried out, or it means leaving the dialog box, in which case all settings will be applied and memorized, that is to say, the next time the dialog box is called up the last settings will be displayed.

Clicking a text box with a frame of screen background color, calls up a dialog box, which can be used to change the contents of the text box (see e.g. edit search paths).

If you click a color box, a dialog box appears, which enables you to choose a color. There are three ways of choosing a color. One is to simply click on one of the sixteen standard colors. Another is to enter a color directly as a RGB- or CMY-value. The third is to choose a color from the listbox of user defined colors. Afterwards the chosen color and the RGB or CMY value will be displayed. When you then leave the dialog box by clicking "OK", this color will be applied. If non-standard colors are to be used frequently, then these can be remembered as user defined colors. For that purpose you enter the color you wish to use and the name of it. Then you click the "Apply" button, after that the color appears in the listbox for user defined colors.

## Program configuration

You have the possibility to adapt TOMMYSOFTWARE® CAD/DRAW to your own personal working environment. This includes e.g. memory control, screen adaption, color choice, standard pathes etc. For more information, see change settings and edit search paths.

The settings, which apply to the current drawing, e.g. paper size, scale, layer display, display modes etc., are of special importance. These settings are usually activated in the View menu.

Furthermore, you also can save **all** settings of the program at any time, as well as loading new ones.

TOMMYSOFTWARE® CAD/DRAW uses a numeric coprocessor, if existing. But the program also runs without a coprocessor.

You can choose between a white and a black background to work on. Shifting from one mode to the other is done in display modes.

## Status window

The status window is between the menu bar and the drawing window. In it all currently active program parameters are displayed. By clicking certain areas in the status window, a corresponding dialog box will be displayed, which enables you to change the parameters belonging to it.

In the first line, the currently active command and the next point to be entered are displayed. The next point depends on the currently active command and on those points already entered. Clicking the command line displays, if possible, a dialog box providing the possibilty to view and change the parameters for the current command.

In the second line, the current layer is displayed. By clicking the text the layer status will be displayed.

Then specifications on the line status follow (width, pattern, color). By clicking the line unit the unit dialog box will be displayed, and by clicking the displayed line the line status dialog box will be displayed.

Clicking the display of the currently active mode will call up the draw attribute box.

There may be different characters in the status display:

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|---|---|
| P | Primary selection is active |
| S | Secondary selection is active |
| X | X-constraints are active, i.e. the cursor can't be moved in an X-direction. |
| Y | Y-constraints are active, i.e. the cursor can't be moved in a Y-direction. |
| G | Grid is on. |
| C | Catch is active. |
| D | Duplicate is active. |

By clicking in this display, the show grid dialog box is called up.

In the third line, the current coordinates, as well as the direction vector, the lenght and the angle, will be displayed relative to the last point entered. By clicking the coordinate display, the scale measure dialog box appears. By clicking a unit display the unit dialog box appears.

## Panel

The panel is on the left side of the drawing area. It contains 17 icons. If an icon is inverted the corresponding setting is active. By clicking an icon with the left mouse button the icon will be displayed inverted or normal respectively. Clicking with the right mouse button causes a dialog box to be displayed, which allows you to edit the settings which belongs to the icon. If an icon has no dialog box, clicking with the right mouse button has the same effect as clicking with the left button.

The first icon indicates the status of the grid. Clicking this icon with the right mouse button causes the grid dialog box to be displayed.

The second icon displays the catch status. The third icon indicates whether duplicate is active or not.

Icons four and five display the current selection mode. "PS" means primary selection, "SS" stands for secondary selection.

The Numbers 1 to 12 in icon 6 to 17 represent the 12 parameter sets. A parameter set consists of drawing mode, filling mode, line style, and filling style. This corresponds in principle with the settings of drawing parameters. If one of these icons is clicked with the left mouse button, the settings associated with the parameter set are used to initialize the current drawing parameters. That means, from now on all objects will be created using the new parameters. Of cause the drawing parameters can be changed at any time. To do that, choose the command drawing parameters, or click with the mouse the corresponding area in the status window. After the current drawing parameters has been changed, the program checks automatically whether a parameter set with identical settings exists. If that is the case the corresponding icon will be inverted. This happens to show you that you also could have adjusted the current drawing settings by clicking that icon. You have to consider that changing the current drawing settings has no effect on the parameter sets.

In order to change a parameter set click with the right mouse button the corresponding icon. As a result a dialog box is displayed which allows you to edit the parameter set. If you have changed something, the program automatically checks whether parameter sets with identical settings exist. In such a case the concerned icons will be inverted, so you can see which parameter sets have identical settings.

You have the possibility to fade out the panel in order to get a larger drawing area. This is done with the command display modes.

## Screen display

A drawing can be displayed in various sizes on the screen. The size you want is to be specified in relation to the original size, i.e. the size of the drawing as it will be printed.

The factor specifying this relative size of the drawing is called the zoom. The zoom is specified as a percentage: 100% means the original size of the drawing, in which case the elements of the drawing appear on the screen in the same size as they would be printed. See zoom section.

If the zoom chosen is less than 100%, then the drawing will be displayed in a reduced size. This is for example necessary when a large drawing needs to be viewed as a whole on the screen.

In order to get an magnified display of the drawing, the zoom chosen should be larger than 100%. This can be useful in order to get a closer look at details of the drawing.

A few frequently used zooms can be adjusted directly by means of special commands. Furthermore, the current zoom can be memorized and restored. See fixed keyboard functions.

The dotted rectangle in the drawing window represents the edges of the paper. The position of the scroll boxes within the scroll bars shows the position of the visible part of the drawing on the paper.

## Point entry and objects

There are basically two ways of entering points: by clicking on the desired position in the drawing window or by entering the point from the keyboard.

When entering with the mouse, the deciding factor is the current position of the cursor on the drawing window, since that is where the point will be inserted to the drawing. The position of the cursor may however be affected by an activated grid or by an X- or Y-directed constraint, in which case the cursor can only be moved to certain positions in the drawing window. If the Catch function is on, then the nearest point will be chosen, independently of the current cursor position.

Entry by means of the keyboard is context-sensitive, which means that, depending on which command (line, circle etc.) is currently active, and independently of the point to be entered, a corresponding dialog box opens when you press F8 (see fixed keyboard functions). This permits the entry of absolute coordinates, relative coordinates (relative to the previous point) and polar coordinates.

The last point you have entered can always be reversed by pressing Esc, regardless of how it was entered.

Entries are always carried out in the current length unit or angle unit as displayed in the status window.

An object, such as a line or a circle (see the Draw menu), either consists of a certain number of points, or the points are used to calculate it. The points can be displayed by choosing the display mode command. The points will be then be displayed as small rectangles. Seeing the points can especially be useful when Catch is active. The points will also then be caught when they're not displayed. In order to make quite sure, however, that the desired point has been caught, the points should be displayed.

Objects are always drawn in the current draw modes with the current line or fill color, which can be changed at any time. Furthermore, an object usually belongs to the current layer.

## Layers

TOMMYSOFTWARE® CAD/DRAW has 256 different layers. Every object drawn is usually placed automatically to the current layer, which is displayed in the status window. With the layer status command, it is possible to determine the parameters of all layers.

A name can be given to each layer separately. You have to assign a parameter set number to each layer. A parameter set consists of drawing mode, filling mode, line style, and filling style. Parameter sets are defined by means of the panel. If you choose a number between 1 and 12 the current drawing parameters are initialized with the corresponding parameter set, if that plane is chosen as current plane. If 0 is entered as parameter set number the current drawing parameters are not changed, if that plane is chosen as current plane.

It is possible to show or hide the layers. If one layer is not shown, this has no effect on the objects of this layer, except that they're not visible and remain so as long as the layer is not displayed. Also it is not possible to select objects belonging to the layers not shown, i.e. you can not manipulate objects which are not displayed.

It is possible to freeze layers. This has the effect, that the layer remains visible but the objects on it can not be selected, and consequently not manipulated either. This function is especially useful for the display of forms text boxes, which should be visible but nevertheless remain unchanged during the creation of the drawing.

By giving a color to a layer, all objects belonging to that layer are displayed in the current pencil color. This setting has no effect on the objects themselves, only on how they are displayed.

By clicking once a line in the list box, where the layer parameters are displayed, the settings of the respective layer will be applied to the box above it, which you can now edit. If you have changed the settings of this layer and wish to apply them, then use the "Apply" button; the settings will then be applied and displayed in the list box. By double-clicking a line or by using the "OK" button you leave the box and the selected layer becomes the current layer. The current layer can not be displayed, nor can it be freezed. If you should try to choose either a displayed or a frozen layer for the current layer, a window opens stating this and the dialog box is not left.

When creating a drawing you should from the beginning keep in mind the possibility of using several layers at once. That makes the work easier and gives a better overview over the drawing.

## Reverse last operation (Undo)

TOMMYSOFTWARE® CAD/DRAW has an undo command. This means that you can reverse the last operation.

With the redo command an undo command can be reversed.

## Selection

Selection means choosing one or more objects. It is possible to select objects in various ways, see select menu. After selecting the objects, certain operations can be performed with them, such as rotate and scale.

Using the select object command, objects can be selected directly with the mouse by clicking. To that effect you must click in the rectangular area where the object is located (the selection area). The selection is displayed by a frame around the selected object (the selection frame).

If a large number of objects is to be selected, this can be done either from the whole drawing or within an area by specifying a rectangular area.

In many cases, objects with identical parameters, such as line width, object type or layer, need to be selected. In such cases the conditioned selection command should be used. The parameters, which the selected objects should have, are chosen from a dialog box.

Finally, objects can also be selected by means of the keyboard. For that purpose the Select an object command must be made active. By then pressing either F11 or BACKSPACE the object nearest to the cursor is preselected, i.e. marked with a frame. If the preselected object is not the desired object, re-pressing either F11 or BACKSPACE preselects the next-nearest object, and so forth. As soon as the desired object has been preselected, it can be selected by pressing either F12 or RETURN. The marking is cancelled by pressing the key combination SHIFT+F11 or SHIFT+BACKSPACE.

There are two different types of selection: the main selection and the secondary selection. They are displayed in different colors. In the Status window, one can see which of the two selection types is the active one. Normally the main selection is active. Basically, the secondary selection is only of importance for the hatch.

The objects remain selected until the selection has been explicitly reversed/cancelled. This is done by deselecting them. For the deselection you have all the same possibilities as for selection.

Various other operations also affect the selection. The Paste command, for instance, deletes the current selection before inserting objects from the clipboard to the drawing. Moreover, saving the drawing does not preserve the selection, which means that when you load a drawing all objects are unselected.

## Groups

A drawing consists of a number of individual objects. In some cases, it may be of use to assemble several objects into one unit so they can all be treated together.

The group is such a unit of objects. It can consist of any objects you wish. If then, for example, one object from the group is selected, all other objects of the group will also be automatically selected. All manipulations then performed will be automatically applied to all objects of the group.

Groups can be created and then dissolved again at any time. They remain unchanged through all operations. If, for instance, the objects of a group are copied, the original objects remain a group and the new objects automatically become one group as well.

The group definition has an absolute priority to all other settings. If, for instance, an object belonging to a group happens to be located in a frozen or a not displayed layer, it will nevertheless be selected and changed together with the other objects of the group.

## Manipulation

Manipulations (rotate, scale, copy etc.) affect only the selected objects. For information on the various manipulations available, see the Shape menu.

If duplicate is on, the original objects remain unchanged and the manipulated objects will be treated as new ones. For information on how to activate and deactivate duplicate, see fixed keyboard functions.

## Marks

Marks are small, point-shaped symbols, used to mark positions, e.g. during construction. They can also be created during the construction. Marks are displayed as small, crossed-through rectangles. They are visible only on the screen, not when printed and not when copied to the Windows clipboard.

There are two ways of creating marks. One is to place them into the drawing at any position with the mouse. The other is to construct them by using the construction commands and by choosing the division command.

Marks are also used in connection with the multiple copy command. This command enables you, for instance, to copy selected objects, in one operation only, to a number of different positions in the drawing, as determined by the marks.

## Construction

TOMMYSOFTWARE® CAD/DRAW is equipped with a number of construction possibilities, enabling you to handle all construction tasks needed. Commands are available for simple constructions, such as dropping a perpendicular, as well as for complex constructions, such as fitting a circle to two straight lines.

In almost all cases it is necessary to enter reference points or reference objects for the construction. If for instance a perpendicular is to be dropped, either the straight line or the circle, on which the perpendicular is to be dropped, must be determined. The determination of reference objects can happen in two ways.

One possibility is the direct entry of points defining the reference object. This is used for constructions of not-existing objects. If for instance the perpendicular is to be dropped onto a line, which as such does not exist, but is known only from two points, then these points can be entered directly.

If the reference object however is already existing, the second possibility will be more advisable. It enables you to determine an existing object directly. To do that the object should be selected by means of F11 or BACKSPACE and F12 or ENTER, as described under selection. This reference object will then be used for the construction.

This direct selection of a reference object is not in all cases possible. Therefore, when the possibility exists it will be specified in the status window by the display of the point to be entered. Otherwise, both types of entry are equally valid.

By simple constructions, such as the dropping of a perpendicular, the constructed object will be directly inserted as soon as the reference points have been entered. It can also be positioned by using the cursor.

By complex constructions there may be a number of correct possibilities to construct an object, depending on activated entries. In this case the program offers all the available selection variants, that is all variants are displayed as lines on the screen. In each case, one variant is marked with a color and enframed in a rectangle. The marking can be moved to the next variant by pressing either F11 or BACKSPACE. For each variant it can be decided, whether it should to be used or not. This is done by pressing either F12 or ENTER. If F12 or ENTER is pressed once, this means choosing the variant, whereby it becomes marked by a color and with a rectangle of the same color around it. Pressing F12 or ENTER the second time cancels/withdraws the choice/selection.

Any number of variants can be chosen and applied at the same time. By doing that the repeated entry of reference points will not be necessary.

It is possible to set two parameters for the construction. On the one hand, you can determine whether, instead of objects only marks should be inserted, corresponding to the definition points of the objects to be constructed. On the other hand it can be decided, during the construction of circles, whether full circles should always be created, or only arcs of a circle.

## Hatching

There are three different methods to hatch: union hatch, intersection hatch and difference hatch. Hatching can be carried out only if the objects to be hatched have been selected.

With the union hatch, all selected (primary or secondary selected) objects become hatched.

With the intersection hatch, the intersection of objects which are primary selected or secondary selected are hatched, covering exactly the same area as covered as well by the primary selected as by secondary selected objects.

The difference hatch refers to the difference between primary selected and secondary selected objects, i.e. the primary selected objects are hatched, with the exception of those areas, which are occupied by secondary selected objects.

By all hatch methods, the currently active hatch parameters are taken into consideration. The hatch parameters determine the width, the pattern and the color of the individual hatch lines, as well as the distance between them, their inclination angle and their display. There is a hatch 1 and a hatch 2, which can be turned on and off seperately. Both of them consist of 8 hatch lines. This makes it possible to create hatches with intersecting hatch lines.

Using the hatch types command, you can save up to 10 hatch parameters. In addition to that, you can create a hatch library. In this way any object (circles, triangles etc.) can be hatched, ignoring possible fillings of these objects.

The hatch layer command determines the layer, to which the hatch is to be inserted. You can choose between inserting the hatch to the current layer or to a fixed layer.

The hatching is an object like any other (as for instance a line or a circle), which means that it can at any time be selected and manipulated.

## Dimensioning

Dimensioning means giving a measure to the drawing. TOMMYSOFTWARE® CAD/DRAW is equipped with associative dimensioning . This means that, as desired, the dimension will be automatically adjusted to all changes made to the drawing.

A number of reference points need to be specified for dimensioning. The first thing is to determine points to define the position of the object to be dimensioned. When dimensioning a line, for instance, the endpoints of the line are first specified. Then the position of the dimension line is determined. TOMMYSOFTWARE® CAD/DRAW has several aids to offer for positioning the dimension line, which can be turned on and off seperately.

Then the dimension is calculated and can now also be positioned. The dimension number is automatically so rotated, that it stands correctly on the dimension line. It will also be automatically rotated, so that it can be read from below or from the right. For positioning, some aids are also available, for instance allowing to place the dimension number in the middle of the dimension line.

There are a number of different ways of changing the appearance of the dimension line. In addition to the color and the width of the dimension line, both of the dimension arrows can be turned on and off seperately. Also, a special dimension aid line can be displayed at each end.

To make the correct display of the dimension numbers possible, the font used in the display of dimension numbers must be loaded first by using the font command. Otherwise, the dimension numbers are indicated by a crossed rectangle.

The dimension font is not a part of the drawing itself. As soon as another font is loaded at its place, all dimensions are shown in the new font. This can be used to save time. During the process of drawing, the dimensions are either displayed in one simple font (e.g. DIN-DRAFT, see Font), or not at all. This is to make the screen setup very fast. Before printing this font will be replaced by a font of higher quality (e.g. DIN-LQ, see Font). From now on all dimensions are displayed and printed in the new font.

## Trimming

Trimming means making changes to already existing objects. When you are creating a drawing, it may often be necessary to make changes to objects in order to adjust them to changed measures or new objects.

For trimming an object, it must first be selected. Then the desired trimming command should be chosen. As by the construction of objects, reference objects must be entered for trimming. The procedure of entering reference objects is the same as by the construction.

And as by the construction, there are also different correct ways of making changes to the selected object by trimming. If that's the intention, the variants are displayed and may be chosen seperately. You can choose more than one variant at a time, thus turning the original object into a number of objects. The new objects, as well as the original one, will be selected, in order to make direct further changes to them easier.

Since the purpose of trimming is to make changes to an existing object, the original object will be changed by this operation. If it nevertheless is to be retained, then duplicate should be activated before starting the operation.

## Outline

An outline is a free form area, consisting of a number of contours. Each contour is again made up of lines and Bézier curves. Contours can be "boxed", in order to make "holes".

It is possible to enter outlines directly, by putting each contour seperately together from lines and Bézier curves. The Bézier curves can be entered in different ways.

An outline can, however, also be created from existing objects. This is done, for example, when a complex plane, made up of individual lines, needs to be hatched (see hatching).

Existing outlines can easily be changed later. Also, seperate outlines can be assembled into one, and complex outlines dissolved into seperate ones.

## Libraries

A library may contain up to 900 symbols. A symbol may consist of any objects you wish, representing for instance a circuit element, a drawing component or a font character.

You can insert symbols to the drawing by choosing the insert symbol command. A list box will then appear where you can choose the desired symbol. Then a rectangle will be displayed on the cursor, representing the symbol, which can be inserted to the drawing at any place by clicking with the mouse. This procedure can be repeated as long as another command is not chosen.

Symbols can also be read and saved. To do that, at first all objects you want to belong to the symbol must be selected. It is advisable to build a group out of these selected objects, because this eases the further work with this symbol. Now, choose the read symbol command. The first thing is then to choose a point to determine a position in the symbol, later to be at the cursor position when inserted. When this has been done a dialog box opens, in which the symbol name should be entered. The symbol is then added to the library.

When entering a symbol, attention should be payed to the symbol parameters. There are two setting possibilities: "Point" and "Area". If "Point" is the active setting, this being the standard setting, then the procedure runs as described above. If "Area" is the active setting, then instead of the point a rectangular area (i.e. two points) is to be entered. This is necessary when the library contains a font, see read symbol.

## Input ranges and calculation accuracy

The valid coordinate range is -1.0e100 to 1.0e100 (this is a One with hundred Zeros) millimeters, no matter of the currently active unit. Input in dialog boxes is checked for these limits.

TOMMYSOFTWARE® CAD/DRAW internally works with an accuracy of 15 fraction digits. This is enough to ensure an accuracy of 0.001 µm. This is nearly inmeasurable. But still there is no total exactness, what can be recognized in some displayed values, if e.g. they show the value 0,9999999999 instead of the exact value 1.0. This insignificant inaccuracy will never have any harmful affect on the printed drawing and can be ignored.

## Error messages and diagnosis

**"Execution not possible! Not enough memory. Reserve more memory?"**

This message appears, when the currently available memory is not sufficient for the execution of the current command.

Click on "Yes" in order to increase the available memory by the amount specified with the command edit settings in the box "initial size". The current command will be cancelled in any case.

If this messages frequently appears after short work with the program, you should increase the "initial size" of the available memory by means of the command edit settings and save those settings as default settings by means of the command save settings, so that these settings will still be active after leaving and restarting the application.

**"Calculation not possible! Invalid parameters or not enough system memory."**

When this messages appears, first check the parameters relevant for the current command and the point entered, because normally this messages indicates, that the input doesn´t make any sence, so that the command couldn´t be executed. This might occur, e.g., if the intersection of two parallel lines is to be calculated.

Another reason for this message to appear can be that the system memory, which is used for several internal calculation, is not sufficient. This often occurs, e.g., if complex areas are to be hatched. In this case, increase the size of the system memory by means of the command edit settings.

**"Calculation not possible! Invalid parameters or calculation too complex."**

When this messages appears, first check the parameters relevant for the current command and the point entered, because normally this messages indicates, that the input doesn´t make any sence, so that the command couldn´t be executed. This might occur, e.g., if no or only one point was entered when drawing a polyline.

Another reason for this message to appear can be that the calculation was too complex, i.e. it produced too many results. This might happen, e.g., executing the command intersection all objects, when too many objects have been selected.

**"No object existing or no valid object selected."**

The currently active command expects at least one object to be selected, but no object is selected. Please select at least one object before executing the current command again.

If this message appears during the reading of a symbol via the command read symbol, the message box might offer the button "Ignore" in addition to the button "Cancel". This button allows to continue the execution of the command. This possibilty is needed to read an "empty" symbol, like the character "Space" in fonts, which normally consists of zero objects. This option is only available, if the symbol parameters are set to "Area".

**"Invalid selection. Please select exactly ... ."**

The currently active command expects a specific number of special objects to be selected, but this is not the case. Please select the required objects before executing the current command again.

**"No definition point selected."**

The currently active command expects at least one definition point to be selected, but no definition point is selected. Please select at least one object before executing the current command again.

## File menu

New drawing
- Load
- Choose
- Merge
- Save
- Save as
- Close

Save objects as
Convert drawing
Print drawing
Print objects
Printer setup
Control panel
Exit

## New drawing

Using this command erases the current drawing. If necessary, a security check will appear first. Then a dialog box appears, that can be used to set a new sheet size and a new sheet orientation.

Predefined formats for the sheet size according to DIN and ISO, as well as some common US-formats, are available. Clicking on one of the option buttons sets the corresponding format. The size of the current format is displayed in input boxes, either in millimeters or inches. You also can enter any desired value between 10 and 4000 millimeters, either in millimeters or inches.

You set the sheet orientation by clicking the desired option button.

By clicking the "Custom format" or "Undefined" option buttons the values in the input boxes will be applied, not the standard formats.

After leaving the dialog box with the "OK" button, the drawing will be erased and the new sheet opens. The dotted frame indicates the sheet size and sheet position.

Clicking the "Cancel" button retains the current sheet format, but nevertheless erases the drawing.

The name of the new drawing appears in the title bar as "(UNTITLED)". When saved, the chosen name appears at its place.

An erased drawing can be reloaded quickly. For information, see choose drawing.

## Load drawing

A new drawing will be loaded. If necessary, a security check will appear first. Then a file selector box appears, where you choose the name of the desired drawing.

If the check box "Load MPI" is active then the settings which have been saved under the same name will be loaded as well.

The old drawing can be reloaded quickly. For information, see choose drawing.

## Choose drawing

A dialog box appears, that displayes the eight last manipulated drawings (excluding the current one). The last manipulated drawing appears on top of the list.

Clicking on a name in the list loads the corresponding drawing - if necessary a security check will appear first. Then the name appears in the title bar.

A new drawing is always added to the list, when you e.g. open a new drawing or load a drawing. If there are already eight drawings on the list when a new one is added, the last drawing on the list will be removed.

Only drawings with a name are kept on the list. If you open a new drawing and make some changes to it, a security check appears, as usual, before loading the selected file. If you then do not save the drawing, by giving it a name, it won't be added to the list.

## Merge drawing

Merging a drawing adds the chosen drawing to the current one; in other words, the current drawing will not be erased. There are a number of possibilities to merge a drawing. Sometimes it may be necessary to enter an area. If needed, a security check will also appear. Then a file selector box appears, where the name of the drawing to be loaded is chosen.

### Original

The chosen drawing will be added to the current one without any previous changes, i.e. its position on the sheet corresponds to the position it had when it was saved.

### Centered

Select two points to determine an area. The chosen drawing will then be centered at the mid point of that area. The drawing to be merged will not be scaled during this procedure, i.e. neither enlarged nor reduced.

### Proportional

The drawing will be scaled in order to fit exactly into the selected area after being merged. Its height-width relation remains unchanged, i.e. the drawing will be enlarged or reduced by the same factor in height as in width.

### Any

The drawing will be scaled, in order to fit exactly into the selected area after being merged. Its height-width relation may however be changed, i.e. the drawing can be enlarged or reduced by different factors in height and width.

## Save drawing

Saves the current drawing.

If it has not a name yet, in which case "(UNTITLED)" appears in the title bar, a file selector box will appear, that can be used to choose a name for the drawing. If a file already exists with the name you choose, a dialog box enables you to either cancel or confirm the saving.

Afterwards the settings are saved under the same name (of cause with the extension *.MPI).

## Save drawing as

A file selector box appears, that can be used to choose a name for the file to be saved. If a file already exists with the name chosen, a dialog box enables you to either cancel or confirm the saving.

If you want to save the current settings under the same name (with the extension *.MPI), you have to activate the check box "Save MPI".

Two dialog boxes will then appear, they can be used to enter some drawing details and setting details.

Finally, it is possible to apply this name as a new name for the current drawing.

## Close drawing

Closes the current drawing. If necessary, a security check will appear first. The drawing will then be removed from the list of the last manipulated drawings and can consequently not be selected with the choose drawing command any more.

Then, the next drawing on the list of the last manipulated drawings will be loaded. If no more drawings are on the list, a new drawing will be opened.

## Save objects as

A file selector box appears, that can be used to choose the file name for the selected objects to be saved under. If a file already exists with the name chosen, a dialog box enables you to either cancel or confirm the saving.

For information on the selection of objects, see Selection menu.

## Convert drawing

This command enables you to call converters. These are programs that convert drawings from one file format to another file format. If, e.g., a drawing is available in the DXF format of AutoCAD, and you want to work on it with TOMMYSOFTWARE® CAD/DRAW, you have to convert this drawing to the MPG format of TOMMYSOFTWARE® CAD/DRAW.

A dialog box appears, in which all available conversion programs (*.EXE) are displayed.

After choosing one of the conversion programs, it will be executed. The further behaviour depends on the specific conversion program file to be converted. For more information, see the description of the specific conversion program.

All conversion programs must be located in the folder "CONVERT" in the system path. The name of the program informs you of the file formats the program converts. E.g., the program DXF_MPG.EXE converts DXF files to MPG files, the program MPG_DXF.EXE converts MPG files to DXF files.

## Print drawing

This command enables you to print the current drawing. The standard printer will be used, you can select another standard printer with the control panel command.

With the printer setup command you can adjust the settings of the current printer.

A dialog box appears, that can be used to set some printer parameters.

Header
Footer

You can print one header line and one footer line. Both lines may contain up to 256 characters. In addition to normal text, they may also contain control characters, which enable for example the printing of the current time. When printed, the control characters will be replaced by the information entered.

Control characters begin with the "%" symbol, followed by one of the following characters:

| | |
|---|---|
| "Z" or "z" | Title of drawing |
| "D" or "d" | Current date |
| "T" or "t" | Current time |
| "A" or "a" | Name of author |
| "N" or "n" | Name of change author (see drawing details) |
| "%" | Prints a "%" |

Activating the check box turns the printing of header or footer on or off. The option buttons enable you to determine the position of the header or footer.

Orientation

Here you can determine whether you want to print in portrait or landscape. The default setting corresponds to the current sheet format.

Scale factor

The scale factor specifies the desired scaling of the drawing in percentages. 100% means that the size of the drawing remains unchanged. 50% means that, before printing, the drawing will be reduced to the half of its size. Typing 200% has the effect, that the drawing will be printed in its double size. The value entered affects only the printing size, the drawing itself will not be changed. Values between 10% and 1000% are valid.

The size of printed drawings can be up to 4000 by 4000 millimeters. Should the scale factor be to large, the resulting drawing may turn out to be larger than allowed. If this is the case, printing will be cancelled and an error message appears.

Number of copies

The number of copies specifies how many copies of each page should be printed. Values between 1 and 250 are valid.

Draft mode

If this check button is active, the output will be done in draft mode. This normally increases the print speed, but decreases the quality.

Leaving the dialog box by clicking "OK" starts the printing. Another dialog box will then

appear, stating that printing is still in process. By clicking the "Cancel" button, you can cancel the printing process at any time.

## Print objects

This command enables you to print all selected objects of the current drawing. The standard printer will be used, you can select another standard printer with the control panel command.

With the printer setup command you can adjust the settings of the current printer.

A dialog box appears, that can be used to set some printer parameters.

Header
Footer

You can print one header line and one footer line. Both lines may contain up to 256 characters. In addition to normal text, they may also contain control characters, which enable for example the printing of the current time. When printed, the control characters will be replaced by the information entered.

Control characters begin with the "%" symbol, followed by one of the following characters:

| | |
|---|---|
| "Z" or "z" | Title of drawing |
| "D" or "d" | Current date |
| "T" or "t" | Current time |
| "A" or "a" | Name of author |
| "N" or "n" | Name of change author (see drawing details) |
| "%" | Prints a "%" |

Activating the check box turns the printing of header or footer on or off. The option buttons enable you to determine the position of the header or footer.

Orientation

Here you can determine whether you want to print in portrait or landscape. The default setting corresponds to the current sheet format.

Scale factor

The scale factor specifies the desired scaling of the drawing in percentages. 100% means that the size of the drawing remains unchanged. 50% means that, before printing, the drawing will be reduced to the half of its size. Typing 200% has the effect, that the drawing will be printed in its double size. The value entered affects only the printing size, the drawing itself will not be changed. Values between 10% and 1000% are valid.

The size of printed drawings can be up to 4000 by 4000 millimeters. Should the scale factor be to large, the resulting drawing may turn out to be larger than allowed. If this is the case, printing will be cancelled and an error message appears.

Number of copies

The number of copies specifies how many copies of each page should be printed. Values between 1 and 250 are valid.

Draft mode

If this check button is active, the output will be done in draft mode. This normally increases the print speed, but decreases the quality.

Leaving the dialog box by clicking "OK" starts the printing. Another dialog box will then appear, stating that printing is still in process. By clicking the "Cancel" button, you can cancel the printing process at any time.

## Printer setup

A dialog box appears, which enables you to adjust settings of the current printer. The settings depend on the current printer type. Here you can for example change the current printer resolution.

You can find more information in the description of the printer driver or the help system delivered with the printer driver.

## Control panel

This command calls up the control panel. For more information, see the Help system for Windows.

## Exit

With this command you quit TOMMYSOFTWARE® CAD/DRAW. If changes have been made to the current drawing or library, a security check will appear before quitting TOMMYSOFTWARE® CAD/DRAW. Thus, you don't have to worry about loosing data by leaving TOMMYSOFTWARE® CAD/DRAW by mistake.

## Edit menu

Undo
Redo
Copy
Paste
Buffer
Change popup menu
  - Update
Change shortcuts
  -Table
Assignments
Assignment details
Program status
Drawing details
Edit search paths
Settings
Setting details

## Undo

Every time you choose this command, the last operation carried out will be reversed. Among others, the following operations can be reversed:

Erase drawing
Cut
Object info
Delete objects
Scale objects
Draw line
Move points
Hatch objects

Some operations can not be reversed, such as:

Close drawing
Erase drawing
Zoom page
Select objects
Build group

Saving a drawing erases the information needed for this command to be able to reverse operations, it is not saved with the drawing. In other words, after saving no undo can be carried out; nor can this be done immediately after a drawing has been loaded!

## Redo

With the redo command, the last undo operation can be reversed.

Saving a drawing erases the information needed for this command to be able to reverse undo-operations, it is not saved with the drawing. In other words, after saving no redo can be carried out; nor can this be done immediately after a drawing has been loaded!

## Copy

With this command you can copy all selected objects to the Windows clipboard. This makes them immediately available for other programs.

This function, just like the print function, does not consider the display modes settings. Also, the marks won't be applied.

It is also possible to import objects from the Windows clipboard, this is done with the command Paste.

The buffer (see the edit menu), as opposed to the clipboard, is TOMMYSOFTWARE® CAD/DRAW internal and can not be used by other programs. But it has the advantage that no rounding errors occur and all object types are available.

If you only want to buffer objects for a short period in order to insert them later, you should really do that with the TOMMYSOFTWARE® CAD/DRAW internal buffer, because this makes sure that all objects remain unchanged.

## Paste

With this command you can paste all objects from the Windows clipboard to the drawing. To do that, you have to enter a point which defines the upper-left corner of the clipboard objects. Afterwards the objects are being imported from the clipboard to the drawing. If there are objetcs which can not be imported an appropriate message appears.

This command makes it possible to import grafic data from other programs. But you should consider that because of the exchange via clipboard the grafic data normally gets changed. This happens due to the fact that in the clipboard points can be stored only with a limited accuracy and only a restricted number of object types are available. The result of this is that the pasting of objects from the clipboard creates rounding errors and originally complex objects (e.g. splines and Bézier curves) are splitted into simple objects (mostly lines).

The buffer (see the edit menu), as opposed to the clipboard, is TOMMYSOFTWARE® CAD/DRAW internal and can not be used by other programs. But it has the advantage that no rounding errors occur and all object types are available.

If you only want to buffer objects for a short period in order to insert them later, you should really do that with the TOMMYSOFTWARE® CAD/DRAW internal buffer, because this makes sure that all objects remain unchanged.

## Buffer

### Cut

All selected objects are copied to the buffer and then erased.

When a large number of objects needs to be moved within the same drawing, this should not be done by using the clipboard and the cut and paste commands, since much more memory and calculation time will then be needed than by using the move objects command to move objects directly.

### Copy

All selected objects will be copied to the buffer.

When a large number of objects needs to be copied, this should not be done by using the buffer and the cut and paste commands, since much more memory and calculation time will then be needed than by using the copy objects command to copy objects directly.

### Paste

### Original

All objects in the drawing become deselected. Then all objects in the buffer will be inserted to their original position in the current drawing. These objects will then be selected.

This enables you to work immediately on the objects so inserted, for example with the move objects command.

### Upper left

You have to enter a point. If this is done, all objects in the drawing become deselected. Then all objects in the buffer will be inserted in the current drawing in such a way, that the upper left corner of the frame which surrounds all objects lies on the previously entered point. Afterwards the inserted objects will be selected.

### Upper right

You have to enter a point. If this is done, all objects in the drawing become deselected. Then all objects in the buffer will be inserted in the current drawing in such a way, that the upper right corner of the frame which surrounds all objects lies on the previously entered point. Afterwards the inserted objects will be selected.

### Lower left

You have to enter a point. If this is done, all objects in the drawing become deselected. Then all objects in the buffer will be inserted in the current drawing in such a way, that the lower left corner of the frame which surrounds all objects lies on the previously entered point. Afterwards the inserted objects will be selected.

### Lower right

You have to enter a point. If this is done, all objects in the drawing become deselected. Then all objects in the buffer will be inserted in the current drawing in such a way, that the lower right corner of the frame which surrounds all objects lies on the previously entered point.

Afterwards the inserted objects will be selected.

**Erase**

This command erases all objects in the buffer. This clears the memory occupied by the buffer.

**Display**

Displays the contents of the buffer in a dialog box.

## Change popup menu

For general information, see popup menu. After choosing this command, choose the command you want to add to the popup menu (see choosing commands). The popup menu then appears. Then you must click on the position in the popup menu where you want the command icon to appear. This completes the procedure and the command can now be chosen from the popup menu.

The icon will be loaded from the "TSCADRAW.BLD" file, which however only is possible if the file is located in the system path.

The changed popup menu can be saved with the save assignment as command.

## Update popup menu

This command updates the icons of the popup menu, i.e. it loads the icon, that has been assigned to each command in the popup menu, from the "TSCADRAW.BLD" file. But no changes will be made to the assignments as such.

Should you one sunny day get hold of a new "TSCADRAW.BLD" file, you can replace the old icons of your popup menu with the new ones by using this command.

## Change shortcuts

For general informations, see assigning keys to commands. After choosing this command, choose the command you want to assign a key combination to (see choosing commands). A dialog box will then appear, that displayes all available keys. Selecting the desired key assigns the command to it.

By using the option buttons, you can assign a key, a shift+key or a ctrl+key directly to the command. If the "Info mode" check box is active, clicking a key displays its current assignment without changing it.

You can save the new shortcuts with the save assignment as command.

## Shortcut table

In the appearing dialog box all shortcuts are displayed. Keys which are not assigned to a command are not displayed. All keys and the command description are displayed in a list boxe. Shift is displayed as '+' and Control as '^'. Through applying of the "Delete" button the currently selected key assignment is deleted.

## Assignments

Assignments means the assignments of the keyboard and the popup menu with commands. You can save or reload the assignments at any time. That enables you to create a number of assignments, each shaped for its own field of application.

### Load assignments

Enables you to load a new assignment by using the file selector box.

### Save assignments as

Select a name in the file selector box for the new assignment. Should a file with that name already exist, a dialog box enables you to cancel or confirm the saving.

Note: On starting the program, TOMMYSOFTWARE® CAD/DRAW loads the assignments with the name TSCADRAW.MPB from the program path (see edit search paths). If you want to change the standard assignments permanently, you should therefore save them under this name.

## Assignment details

A dialog box appears, in which the assignment details can be viewed and changed.

| | |
|---|---|
| Title | This text should contain the name of the assignments. |
| Topic | This text should contain the topic of the assignments, i.e. the intended field of application. |
| Author | This text contains the name of the person who made the last changes to the assignments. As a rule, the entry here is the name of the person TOMMYSOFTWARE® CAD/DRAW has been installed on. But you may change it at will. |
| Last change | This entry shows the time of the last change made to the assignments, i.e. the time at which they were last saved. |
| Comment | This text is intended for any remarks and comments desired. |

## Program status

A dialog box appears, in which the various informations on the current program status are displayed:

**Program data**

| | |
|---|---|
| Memory | This entry shows how much memory has already been reserved for TOMMYSOFTWARE® CAD/DRAW, and has remained unused until now. Should less than 50% of the memory be usable, the memory should be installed again in change settings. |
| Drawing | This entry shows how many objects there are in the drawing and how much memory they occupy. Here, the data needed for the undo and redo also flow in. In other words, after the drawing has been erased, these values aren't necessarily zero! |
| Buffer | This entry show how many objects the buffer contains, and how much memory they occupy. If these objects are no longer needed, they can be erased from the buffer with the erase command. |
| Library | This entry shows how many symbols the current library contains, and how much memory they occupy. |
| Font | This entry shows how many characters the current font contains, and how much memory they occupy. |
| Hatch | This entry shows how many symbols the current hatching library contains (see hatch types), and how much memory they occupy. |

**Screen size**

These entries show the current screen resolution in pixels and the screen size in millimeters. If they deviate from reality more than 10 millimeters, you should set a new screen diagonal with the change settings command.

## Drawing details

A dialog box appears, in which some information on the current drawing can be viewed and changed.

| | |
|---|---|
| Title | This text should contain the name of the drawing. |
| Topic | This text should contain the topic of the drawing, i.e. the project name or the task name. |
| Author | This text should contain the name of the person who created the drawing. Default entry is the name of the person TOMMYSOFTWARE® CAD/DRAW has been installed on. |
| Creation time | This entry shows the time at which the drawing was created, i.e. the time when the drawing was opened with the new drawing command. |
| Change author | This text contains the name of the person that made the last change to the drawing. Default entry is the name of the person TOMMYSOFTWARE® CAD/DRAW has been installed on. It can however be changed as desired. |
| Last change | This entry shows the time at which the drawing was changed the last time, i.e. the time when it was last saved. |
| Comment | This text is intended for any comments and remarks you wish to add. |

## Edit search pathes

The dialog box shows all standard search paths in TOMMYSOFTWARE® CAD/DRAW.

| | |
|---|---|
| System path | This path points to the SYSTEM directory. The IMPORT and EXPORT directories and the TSCADRAW.BLD file must be located in the SYSTEM directory. |
| Info files | This path points to a directory that should contain all MPI files (see save settings as). |
| Assignment files | This path points to a directory that should contain all MPB files (see save assignments as). |
| Drawings | This path points to a directory that should contain all drawings (its default name is DRAWING). |
| Libraries | This path points to a directory that should contain all libraries (its default name is LIBRARY). |
| Images | This path points to a directory that should contain all raster images. |

Clicking inside one of these text boxes, that are enframed in the color of the screen-background, calls up a file selector box, that can be used to choose a new path. If you leave the file selector box via the "OK" button, the new path will be displayed in the corresponding text box (a file name that might have been selected will be ignored).

All paths, except for the system path, are to be thought of as default values only. In the file selector boxes they are offered as default settings, but can however be changed in any way desired.

The "Program path" stands for the path that points to the program itself. You don't have to set it, since it will be automatically determined each time the program is started.

## Settings

Settings refers on the one hand to those settings that can be affected with the change settings command, and on the other hand to almost all other settings that can be activated inside TOMMYSOFTWARE® CAD/DRAW, such as printing details, search paths, drawing parameters, layer status, sheet format, units, scale, grid, display modes, display thresholds, zoom area, mark layer and mark parameters, symbol parameters, selection mode, hatch parameters, hatch types and hatch layers, dimensioning parameters, dimensioning layer and dimensioning font, as well as outlines parameters and others more.

If TOMMYSOFTWARE® CAD/DRAW is to be used for various different purposes, it is recommended to save the proper settings for each purpose seperately. Then you can easily switch between different settings.

**Change settings**

**Standard system colors**

After clicking on any color box, a dialog box appears, allowing the choice of one out of 16 standard colors.

**System**

Memory — This value determines the amount of system memory in KBytes. The value may lie between 100 and 1500 KB.

**Object memory**

Initial size — This value determines the amount of memory that TOMMYSOFTWARE® CAD/DRAW reserves for objects; the value may lie between 100 and 16000 KB. Of course, TOMMYSOFTWARE® CAD/DRAW can't reserve more memory than available in the system. If the existing memory doesn't do to reserve the desired amount of memory, a message appears and you will have to decrease the value.

Step size — The step size is the amount of memory that will be added to the currently reserved memory each time neccessary. This value may lie between 100 and 16000 KB, too.

**Display**

Screen diagonal — Enter here the size of the screen diagonal of your monitor. The value is entered in inch, because normally, screen diagonals are documented in this unit. Common diagonal sizes are 14, 16, 19, 21 and 24 inch.

This specification is needed to display the drawing correctly in zoom original size.

Font size status text — This value is entered in pixels, i.e. screen points. It determines the size of the font used in the status window. Allowed are values between 8 and 30 pixels.

If the check box "Initialize memory" is active while leaving the box by clicking on "OK", the memory will be initialized using the current memory settings. If it is not active, the memory settings entered will be memorized, but not applied. All other settings will be memorized in

any case.

Leaving the box by clicking on "Cancel" cancels all changes done and restores the settings that were active before.

**Load settings**

All settings will be loaded, a file selector box appears. The current drawing will not be changed in any way.

**Save settings as**

All settings will be saved, a file selector box appears. Select the file name for the settings. If a file already exists with the name chosen, a dialog box enables you to either cancel or confirm the saving.

Note: On starting the program, TOMMYSOFTWARE® CAD/DRAW loads the settings with the name TSCADRAW.MPI from the program path (see edit search paths). If you want to change the standard settings permanently, you should therefore save them under this name.

## Setting details

A dialog box appears, in which the setting details can be viewed and changed.

| | |
|---|---|
| Title | This text should contain the name of the setting. |
| Topic | This text should contain the setting topic, i.e. the intended task field for this setting. |
| Author | This text contains the name of the person who made the last changes. Default entry is the name of the person TOMMYSOFTWARE® CAD/DRAW has been installed on. It can however be changed as desired. |
| Last change | This entry shows the time at which the settings were last changed, i.e. the time when these settings were last saved. |
| Comment | This text is intended for any remarks and comments you like. |

## View menu

Drawing parameters
Layer status
Sheet format
Units
Scale
Set origin
Adjust grid
Display modes
Zoom in percent
  - Area
  - Overview
  - Page
  - Original size
Zoom areas

## Drawing parameters

A dialog box appears, in which all drawing parameters can be changed. The drawing parameters are considered by all drawing functions. The displayed image of the objects will however also be affected by the display mode settings.

The current drawing parameters can also be completely new initialized by clicking a parameter set icon in the panel.

Changing the parameters does not affect those objects that are already existing (not even selected objects). Only those objects drawn from now on will become affected by the new settings.

For information on how to change the drawing parameters of already existing objects, see object info.

**Drawing mode**

| | |
|---|---|
| Opaque | When this option button is active, the objects will first, provided the "Filling" or "Both" filling mode is active, be drawn opaque with the current background-color and then the current filling pattern will be drawn with the foreground-color. This means, that objects which are overlapped by other objects will be hidden by these. |
| Transparent | Draws the filled objects as transparent. That is, the objects will not be filled with the background-color, the background will however be determined by the underlying objects, or by the color of the drawing area (see display mode). Only the current filling pattern in the foreground-color will be drawn. |

**Filling mode**

| | |
|---|---|
| Frame | All objects will be drawn with a frame only, that is without a filling. The frame will be drawn with the current line style. |
| Filling | Draws the objects with a filling only, i.e. without a frame. The drawing mode and the filling style will be considered for the filling. Objects that can't be filled, such as lines, would in other words not be visible, since they consist of a frame only. |
| Both | All objects will be drawn with a filling and a frame. The drawing mode and the line style and filling style will be considered. |

**Line style**

| | |
|---|---|
| Pattern | When you click the text box a change line style dialog box appears, in which you can choose a line pattern. A "hairline" is a full line of a minimal width. All other line denotations determined by DIN 15. If you leave the dialog box via the "OK" button, the chosen line-width appears in the text box. |
| Width | The change line style box appears, in which the line width can be set. Allowed (but not necessarily wise) values range from 0 to 100 millimeters, where 0 stands for a line of minimal width (1 pixel on the screen). The entry is always made in the current line unit (see unit). By |

| | |
|---|---|
| | clicking one of the buttons under the text box, the corresponding value appears in the text box. If you leave the dialog box via the "OK" button, the chosen line width appears. |
| Color | If you click inside this color box a dialog box appears, in which you can choose a color. For information, see dialog boxes. The color you choose will then be shown in the color box. |

**Filling style**

| | |
|---|---|
| Pattern | A filling pattern dialog box appears, that enables you to choose a filling pattern. |
| Foreground color | A dialog box enables you to choose another foreground color. |
| Background color | A dialog box enables you to choose another background color. |

## Layer status

For general information, see layers. By using the layer status, you can on the one hand change the current layer, and on the other hand you can set various layer parameters.

**Layer settings**

All settings here presented will only be applied to the chosen layer if you activate the "Apply" button (the chosen layer appears inverted in the list box). If you choose a new layer from the list box, its settings will be shown here and can be edited.

Name — The name of the layer. It may contain up to 15 characters.

Display — When "Yes" is active the layer will be displayed, i.e. its objects can be seen in the drawing area and it is possible to operate on them. Otherwise, the objects of that layer are not visible and can consequently not be changed or edited in any way.

Freeze — If "Yes" is active, all objects of this layer will be frozen, which means that they can be displayed, but not selected.

Dye — If "Yes" is active, all objects of this layer will, regardless of their color, be shown in the current dye. This does not affect the objects themselves. The dyeing color can be set by clicking inside the corresponding color box.

Parameter set number

If you enter a number between 1 and 12 the current drawing parameters will be initialized with the corresponding parameter set if this layer is chosen as current layer. If 0 is entered as parameter set number then the current drawing parameters will not be changed if this layer is chosen as current layer.

**General**

Dye — This check box enables or disables the dyeing of all planes at once. So the drawing can easily be displayed in original colors temporarily without having to undye all layers one by one. This setting has no effect on the "Dye"-setting of each single layer.

Sets — If this check box is active the drawing parameters will not be changed, even if the parameter set number of the chosen plane is between 1 and 12.

By clicking once on a layer line in the list box, its settings will be displayed (see above) and the layer number appears in the text box to the upper right of the list box. Double-clicking on the line means, that this layer will become the current layer and you leave the dialog box. Clicking inside the "OK" button means, that the layer that has its number in the corresponding text box will become the current layer.

The current layer can neither be not displayed nor frozen. Should you try to choose a displayed or a frozen layer for the current layer, a window appears stating this and you won't leave the dialog box.

## Sheet format

A dialog box appears, in which you can set a new sheet size and a new sheet orientation.

Predefined formats according to DIN and ISO, as well as some common US-formats, are available. Clicking one of the option buttons sets the corresponding format. The size of the current format is displayd in text boxes, either in millimeters or inches. You also have the possibility to type any value between 10 and 4000 millimeters as sheet size, either in millimeters or inches.

You set the sheet position by activating the desired option button.

By activating the "Custom format" or "Undefined" option buttons the values in the text boxes will be applied, not the standard formats.

After leaving the dialog box with the "OK" button, the new sheet will appear. The dotted frame symbolizes the sheet size and sheet position.

Clicking the "Cancel" button means retaining the current sheet format.

Changes made to the sheet format do not affect the current drawing.

## Units

All units here set affect almost all areas of the program: the display in the status window, scale, entry of coordinates (see fixed keyboard functions), the specifications in the dialog boxes, etc.

You can set the length units, line units and angle units by activating the disired option button.

Behind the actual value the current unit is displayed, enframed by "[" and "]". All displays and all entries use the current unit.

## Scale

This command enables you to change the scale measure.

One possibility is to choose predefined standard values by using the buttons in the "Enlargement scales" or "Reduction scales" boxes. The other possibility is to type in any scales you want in the text boxes. The area of valid values ranges from 1e-12 and 1e12.

The new scale measure affects the scale and (possibly) the adjusted grid (see adjust grid).

## Set origin

Having choosen this command, you must select a point on the drawing area that specifies the new position of the origin. The origin will then be placed at that position.

The point can be entered by clicking with the mouse on the desired position, or by pressing F8 on the keyboard (see fixed keyboard functions)

Pressing F8 calls up a dialog box, in which you can either directly type in the coordinates or choose positions on the sheet.

## Adjust grid

When the grid is activated (see also fixed keyboard functions) the cursor can be moved only along the raster lines, as determined by the X-grid or the Y-grid.

**X-grid**

| | |
|---|---|
| Buttons | Activating one of the buttons displays the corresponding value in the text box. |
| X-grid on | Activating the check box turns the X-grid on, regardless of whether the Y-grid is on or not. |
| Text box | Any value between 1e-12 and 1e12 may be written here. |

**Y-grid**

| | |
|---|---|
| Buttons | Activating one of the buttons displays the corresponding value in the text box. |
| Y-grid on | Activating the check box turns the Y-grid on, regardless of whether the X-grid is on or not. |
| Text box | Any value between 1e-12 and 1e12 may be written here. |

**Grid mode**

| | |
|---|---|
| Cartesian | The rasterlines, as determined by the X- and Y-grids, intersect perpendicularily. |
| Isometric | The rasterlines, as determined by the X- and Y-grids, intersect at an angle of -30° or +30°. The angles are shown by an additional dotted crosshair. |
| Dimetric1 (-7°) | The rasterlines, as determined by the X- and Y-grids, intersect at an angle of -7° or +42°. The angles are shown by an additional dotted crosshair. |
| Dimetric2 (+7°) | The rasterlines, as determined by the X- and Y-grids, intersect at an angle of +7° or -42°. The angles are shown by an additional dotted crosshair. |
| | All grids here used conform to DIN 5. |

**Grid status**

| | |
|---|---|
| Grid active | When this check box is marked, the grid is active, i.e. the cursor can only be moved along certain points (which points depends on the grid mode). |
| Display grid | Displayes the grid points by converting the corresponding points on the drawing area. Should the distance between two grid points fall short of a given value, the grid automatically won't be diplayed any more - it remains, however, active. The same happens if only one axis is active. |

## Display modes

The display modes determine the appearance of objects or of drawing areas, in no case however does changing the display modes change the object themselves.

**Display points**

The definition points of the objects can be displayed as small squares, if desired, in order to have a better survey when moving defintion points (see e.g. move point). Or in order to ease the catching of points (see fixed keyboard functions).

Three options are available:

| | |
|---|---|
| No | No definition points will be displayed. |
| Selected | Only definition points of selected objects will be displayed. This allows a limitation of points to be displayed, what might increase survey in complex drawings. |
| All | All definition points will be displayed. |

If only the definition points of selected objects are to be displayed, they will not be erased immediately after deselecting an object, due to time saving. They will only be erased after a redraw of the screen or if the screen redraw is set to "Automatic" (see below).

**Filling**

If "Display" is active, all objects are displayed with their respective filling. If not, they are displayed without any filling. Not displaying the filling has the effect that the drawing is redrawn more quickly. On the other hand, it may become more difficult to get an overview over it.

**Line width**

This setting determines the display of lines on the screen.

| | |
|---|---|
| Original | The actual line width will be displayed, depending on the current zoom; the line style will be displayed exact. |
| Minimal | All lines will be displayed with a line width of one pixel on the screen, no matter of their atual width. The line styles will be displayed simplified, so that they don't correspond to DIN exactly. |

In general: The setting "Minimal" is used for a fast screen refresh. This speed is to the debit of accuracy.

The setting "Original" is used for exact screen display. This needs more calculation time and decreases the screen refresh speed.

On the screen, a line may have a width of up to 1000 pixels - that is to say, even when the screen is reduced more (see zoom area), the line will not be displayed any broader.

**Drawing area**

The color of the drawing area may be either black or white.

**Screen redraw**

If the screen redraw is manual, the screen will not be redrawn after deselecting objects, e.g., but the selection frames will be erased directly. If the screen redraw is automatic, the affected screen area will

| | |
|---|---|
| | always be redrawn completely after a change. |
| Selection frame | The selection frame can be displayed either as an simple frame or, if large enough, with eight additional boxes. |
| Menu names | You can choose between long or short menu names. |
| Display panel | If "Yes" is active the panel is displayed. |

## Zoom in percent

A dialog box appears, in which the current zoom factor is displayed in percentages, and in which the new zoom can be entered.

Values under 100% cause the drawing to be displayed smaller than real.

Values over 100% cause the drawing to be displayed larger than real.

Allowed values range from about 1 (depending on the selected sheet size and the screen size) to 1e12.

## Zoom area

By selecting two points you specify the drawing area to be displayed. In this way, you can select a part of the drawing to work on.

If you select the same point twice, that position will be displayed as large as possible.

## Zoom overview

Displays all objects in the current drawing.

## Zoom page

The current sheet is displayed, as being visible on the screen in its entire size. This zoom factor offers a good overview over the drawing, and is at the same time a good starting point for using the zoom area command to display details of the drawing.

## Zoom original size

Choosing this zoom displays the drawing in such a way, that all objects appear in their actual size. A circle with a diameter of 50 millimeters would have a diameter of 50 millimeters on the screen.

For an exact display, it is important to set the correct size on your monitor with the change settings command.

## Zoom areas

With this command you can memorize and use again up to 20 zoom areas, i.e. to mark and insert again certain areas from the drawing.

A dialog box appears, in which you can select a zoom area by clicking once on an entry in the list box. Its name will be displayed in the text box at its right. You can now either enter a new name or change the present one. If you click the "Apply" button, this name will be displayed in the corresponding line in the list box, and the current zoom area will at the same time be assigned to it. After leaving the dialog box via the "OK" button, it will then be memorized and can be chosen again at a later time.

Double-clicking on an entry in the list box means leaving the dialog box and the corresponding zoom area will be applied. Clicking "OK" causes the inverted entry in the list box to be applied.

You can also move quickly between various areas of the drawing by choosing zoom areas that contain e.g. some details of the drawing.

Also see fixed keyboard functions.

## Shape menu

Erase drawing
  - Center
Object info
Delete objects
  - Copy
  - Move
  - Scale
  - Rotate
  - Reflect
  - Sheer
  - Center
  - Lock
  - Release
  - Foreground
  - Background

## Erase drawing

All objects of the current drawing (selected or not) will be erased. This happens without a security check appearing first. You can, however, reverse the erase by using the undo command.

## Center drawing

Depending on the selected mode, all objects of the drawing will be centered.

### Horizontal

The objects are moved in such a way, that they will be at an equal distance to the left and the right edge of the sheet. Their position relative to the upper and lower edges will not be affected.

### Vertical

The objects are moved in such a way, that they will be at an equal distance to the upper and the lower edge of the sheet. Their position relative to the left and right edges will not be affected.

### Both

The objects are moved in such a way, that their position will be, both vertically and horizontally, equally far from the edges of the sheet.

## Object info

Using the object info, you can change the drawing parameters of already existing, selected objects. This happens without affecting the current drawing parameters.

Most of the entries in the dialog box corresponds to the type and function of the entries in the Drawing parameters dialog box. In addition, there is an entry where you can type in a layer number. Moreover, there is a check box for each setting. If the check box is active, the corresponding setting will be applied to all selected objects. Should you make changes to a setting, its check box will automatically be activated.

If only one object is selected, its drawing parameters are be displayed as the default settings of the dialog box. The parameters remain unchanged if you leave the dialog box by clicking the "OK" button, and can consequently be applied to other objects.

If you leave the dialog box by clicking the "OK" button, all settings whose check box are active will be applied to all selected objects.

## Delete objects

Erases all selected objects (see also fixed keyboard functions). You can reverse the erase operation by using the undo command.

## Copy objects

First, you must select a point. All selected objects will later be copied in an orientation to that point. A crossed rectangle appears, that moves with the cursor and indicates the objects to be copied. Then, another point must be selected where the objects are to be inserted to the drawing.

The points can be entered either with the cursor or from the keyboard (see fixed keyboard functions). More information on the point entry will be displayed in the status window.

After the insertion of the objects, the originally selected objects will be deselected and the copied objects will be selected. This enables you to start working on them right away.

## Move objects

First, you must select a point. All selected objects will later be moved in an orientation to that point. A crossed rectangle appears, that moves with the cursor and indicates the objects to be moved. Then, another point must be selected where the objects are inserted to the drawing.

The points can be entered either with the cursor or from the keyboard (see fixed keyboard functions). More information on the point entry will be displayed in the status window.

After the insertion of the objects, the moved objects remain selected.

## Scale objects

All selected objects will be scaled, i.e. enlarged or reduced.

### Percent

A dialog box appears, that can be used to enter a percentage. A value of more than 100% will enlarge, a value of less than 100% reduce the objects.

### Proportional

Select two points to determine an area. The selected objects will then be moved and scaled (the height-width relation may change) to fit exactly into this area.

### Any

Select two points to determine an area. The selected objects will then be moved and scaled (the height-width relation may change) to fit exactly into this area.

The points can be entered either with the cursor or from the keyboard (see fixed keyboard functions). Additional information on the point entry will be displayed in the status window.

Should duplicate (see fixed keyboard functions) be active, the originally selected objects will be retained. They will be deselected and the inserted duplicates become selected. If duplicate is not active, the objects will be directly manipulated on and then remain selected.

## Rotate objects

### Midpoint

Rotates the selected objects around their common midpoint. A dialog box appears, in which the rotation angle is to be entered.

### Any

First, select a point around which the selected objects are to be turned. A crossed rectangle then appears, representing the selected objects. Also, a line appears, that is drawn from the already selected rotation point to the current cursor position. Now, choose a point to determine the rotation angle. The angle can also be directly entered from the keyboard (see fixed keyboard functions), using the current angle unit. Additional information on the point entry will be displayed in the status window.

Should duplicate (see fixed keyboard functions) be active, the originally selected objects will be retained. They will be deselected and the inserted duplicates become selected. If duplicate is not active, the objects will be directly manipulated on and then remain selected.

## Reflect objects

### X-axis

All selected objects will become reflected around a line parallel to the x-axis, that lies through the common midpoint of the selected objects.

### Y-axis

All selected objects will become reflected around a line parallel to the y-axis, that lies through the common midpoint of the selected objects.

### Line

Enter two points to determine the reflection line of the objects.

The points can be entered either with the cursor or from the keyboard (see fixed keyboard functions). Additional information on the point entry will be displayed in the status window.

Should duplicate (see fixed keyboard functions) be active, the originally selected objects will be retained. They will be deselected and the inserted duplicates become selected. If duplicate is not active, the objects will be directly manipulated on and then remain selected.

## Sheer objects

### Horizontal

First, choose a point to use as a reference point for the second point entered. A crossed rectangle then appears, representing the selected objects. Also, a line appears, going from the already selected reference point to the current cursor position. The point now selected determines the angle, by which the objects should be sheered. The objects will be sheered along the horizontal line. The angle can also be entered directly from the keyboard (see fixed keyboard functions), using the current angle unit. Additional information on the point entry will be displayed in the status window.

### Vertical

First, choose a point to use as a reference point for the second point entered. A crossed rectangle then appears, representing the selected objects. Also, a line appears, going from the already selected reference point to the current cursor position. The point now selected determines the angle, by which the objects should be sheered. The objects will be sheered along the vertical line. The angle can also be entered directly from the keyboard (see fixed keyboard functions), using the current angle unit. Additional information on the point entry will be displayed in the status window.

Should duplicate (see fixed keyboard functions) be active, the originally selected objects will be retained. They will be deselected and the inserted duplicates become selected. If duplicate is not active, the objects will be directly manipulated on and then remain selected.

## Center objects

### Sheet, horizontal

The selected objects are moved in such a way, that they will be at an equal distance to the left and the right edge of the sheet. Their position relative to the upper and lower edges will not be affected.

### Sheet, vertical

The selected objects are moved in such a way, that they will be at an equal distance to the upper and the lower edge of the sheet. Their position relative to the left and right edges will not be affected.

### Sheet, both

The selected objects are moved in such a way, that they will be, both vertically and horizontally, at an equal distance to the edges of the sheet.

### Frame, horizontal

Select two points to determine an area. The selected objects are moved in such a way, that they will be at an equal distance to the left and the right edge of the sheet. Their position relative to the upper and lower edges will not be affected.

### Frame, vertical

Select two points to determine an area. The selected objects are moved in such a way, that they will be at an equal distance to the upper and the lower edge of the sheet. Their position relative to the left and right edges will not be affected.

### Frame, both

Select two points to determine an area. The selected objects are moved in such a way, that they will be, both vertically and horizontally, at an equal distance to the edges of the sheet.

The points can be entered either with the cursor or from the keyboard (see fixed keyboard functions). Additional information on the point entry will be displayed in the status window.

Should duplicate (see fixed keyboard functions) be active, the originally selected objects will be retained. They will be deselected and the inserted duplicates become selected. If duplicate is not active, the objects will be directly manipulated on and then remain selected.

## Lock objects

Locks all selected objects, which is to say that they can't be selected any more and consequently not manipulated on.

## Release objects

Releases all objects that are locked at that moment. They can then be selected and manipulated on.

## Objects to foreground

Brings the selected objects to the foreground. This means changing the redraw order of all objects in the drawing in such a way, that the selected objects will be the last ones redrawn, and thus overlapping all other objects.

The appearance of the objects is also affected by the drawing parameters.

## Objects to background

Brings the selected objects to the background. This means changing the redraw order of all objects in the drawing in such a way, that the selected objects will be the first ones redrawn, and thus overlapped by all other objects.

The appearance of the objects is also affected by the drawing parameters.

## Draw menu

Line
Polyline
Triangle
Quadrangle
Rectangle
Trapezium
Parallelogram
Polyeder
Circle
Ellipse
Parabola
Zigzag line
  - Parameters
Circle arc chain
Bézier chain
Spline

## Line

A line is defined by two definition points:

| | |
|---|---|
| Start-point | Specifies the start-point of the line |
| End-point | Specifies the end-point of the line |

By pressing the ESC key, you can take back the last point entered. Pressing the right mouse button cancels the entry (for information, see point entry and objects).

## Polyline

A polyline is defined by a number of definition points. When the start-point has been entered, each additional entry of an end-point adds a line to the polyline.

By pressing the ESC key, you can take back the last point entered (see also point entry and objects).

Pressing the right mouse button cancels the entry. A dialog box then appears, in which you can decide how to go on:

| | |
|---|---|
| Yes | Applies the polyline in the current form and ends the entry. |
| No | Cancels the polyline entry. Already entered lines will be lost. |
| Return | Returns to the polyline entry. Additional points can be entered to the polyline. |

A polyline may consist of up to 2000 definition points.

## Triangle

A triangle is defined by three definition points:

| | |
|---|---|
| Corner 1 | Specifies the first corner-point of the triangle. |
| Corner 2 | Specifies the second corner-point of the triangle. |
| Corner 3 | Specifies the third corner-point of the triangle. |

By pressing the ESC key, you can take back the last point entered. Pressing the right mouse button cancels the entry (for information, see point entry and objects).

## Quadrangle

A quadrangle is defined by four definition points:

| | |
|---|---|
| Corner 1 | Specifies the first corner-point of the quadrangle. |
| Corner 2 | Specifies the second corner-point of the quadrangle. |
| Corner 3 | Specifies the third corner-point of the quadrangle. |
| Corner 4 | Specifies the fourth corner-point of the quadrangle. Two of the edges of the quadrangle may intersect. |

By pressing the ESC key, you can take back the last point entered. Pressing the right mouse button cancels the entry (for information, see point entry and objects).

## Rectangle

A rectangle is defined by two definition points:

| | |
|---|---|
| Corner 1 | Specifies the first corner-point of the rectangle. |
| Corner 2 | Specifies the second corner-point of the rectangle. It lies diagonally opposite to the first corner-point. |

By pressing the ESC key, you can take back the last point entered. Pressing the right mouse button cancels the entry (for information, see point entry and objects).

The rectangle is saved as a quadrangle. This has the effect, that each of the four corners can later be changed individually and indepedently of the other corners (see e.g. move point).

## Trapezium

A trapezium is defined by four definition points:

| | |
|---|---|
| Corner 1 | Specifies the first corner-point of the trapezium. |
| Corner 2 | Specifies the second corner-point of the trapezium. |
| Corner 3 | Specifies the third corner-point of the trapezium. |
| Corner 4 | Specifies the fourth corner-point of the trapezium. This point can be moved only along a line parallel to the line between the first and the second corner-point. |

By pressing the ESC key, you can take back the last point entered. Pressing the right mouse button cancels the entry (for information, see point entry and objects).

The trapezium is saved as a quadrangle. This has the effect, that each of the four corners can later be changed individually and independently of the other corners (see e.g. move point).

## Parallelogram

A parallelogram is defined by three definition points:

| | |
|---|---|
| Corner 1 | Specifies the first corner-point of the parallelogram. |
| Corner 2 | Specifies the second corner-point of the parallelogram. |
| Corner 3 | Specifies the third corner-point of the parallelogram. |

By pressing the ESC key, you can take back the last point entered. Pressing the right mouse button cancels the entry (for information, see point entry and objects).

The parallelogram is saved as a quadrangle. This has the effect, that each of the four corners can later be changed individually and independently of the other corners (see e.g. move point).

## Polyeder

A polyeder is a regularily shaped polygon. First, determine the number of the sides. A dialog box appears, in which the number of sides should be entered. It has to be between 3 and 99.

### Standard

A standard polyeder is defined by three definition points:

| | |
|---|---|
| Mid-point | Specifies the mid-point of the surrounding circle. |
| Radius | Specifies the radius of the surrounding circle. |
| Rotation angle | Specifies the position of the first corner. It will lie on the radius that runs from the mid-point through this point. |

### 3 points

A 3 point polyeder is defined by four definition points:

| | |
|---|---|
| Point 1 on circle | Specifies the first point on the surrounding circle. |
| Point 2 on circle | Specifies the second point on the surrounding circle. |
| Point 3 on circle | Specifies the third point on the surrounding circle. |
| Rotation angle | Specifies the position of the first corner. It will lie on the radius that runs from the mid-point through this point. |

### Side length

A side length polyeder is defined by two definition points:

| | |
|---|---|
| Mid-point | Specifies the mid-point of the surrounding circle. A dialog box then appears, where the side length should be entered. |
| Rotation angle | Specifies the position of the first corner. It will lie on the radius that runs from the mid-point through this point. |

By pressing the ESC key, you can take back the last point entered. Pressing the right mouse button cancels the entry (for information, see point entry and objects).

## Circle

### Complete, standard

A standard complete circle is defined by two definition points:

| | |
|---|---|
| Mid-point | Specifies the mid-point of the circle. |
| Radius | Specifies the radius of the circle. |

### Complete, 3 points

A 3 point complete circle is defined by three definition points:

| | |
|---|---|
| Point 1 on circle | Specifies the first point on the circle. |
| Point 2 on circle | Specifies the second point on the circle. |
| Point 3 on circle | Specifies the third point on the circle. The mid-point will be calculated to make the circle run through all three points. |

### Complete, diameter

A diameter complete circle is defined by two definition points:

| | |
|---|---|
| Start-point diameter | Specifies the start-point of the diameter. |
| End-point diameter | Specifies the end-point of the diameter. |

The diameter complete circle will be saved as a standard complete circle. This has the effect, that only the mid-point and the radius can be changed later (see e.g. move point).

### Arc, standard

A standard circle arc is defined by four definition points:

| | |
|---|---|
| Mid-point | Specifies the midpoint of the circle. |
| Radius | Specifies the radius of the circle. |
| Start-angle | Specifies the start-angle. The radius, that runs from the mid-point through this point sets a limit to the arc. |
| End-angle | Specifies the end-angle. The radius, that runs from the mid-point through this point sets a limit to the arc. The arc will then be drawn anticlockwise from the start-angle to the end-angle. |

The end-points of the standard circle arc will be calculated as definition points and can be used for the catch function.

### Arc, 3 points

A 3 point circle arc is defined by three definition points:

| | |
|---|---|
| Start-point | Specifies the start-point of the circle arc. |

| | |
|---|---|
| End-point | Specifies the end-point of the circle arc. |
| Point on circle arc | Specifies a point through which the circle arc should run. |

**Sector**

A circle sector is defined by four definition points:

| | |
|---|---|
| Mid-point | Specifies the mid-point of the circle. |
| Radius | Specifies the radius of the circle. |
| Start-angle | Specifies the start-angle. The radius, that runs from the mid-point through this circle sets a limit to the sector. |
| End-angle | Specifies the end-angle. The radius, that runs from the mid-point through this circle sets a limit to the sector. The sector will then be drawn anticlockwise from the start-angle to the end-angle. |

The corner-points of the circle sector will be calculated as definition points and can be used for the catch function.

**Ring**

A circle ring is defined by three definition points:

| | |
|---|---|
| Mid-point | Specifies the common mid-point of both circles. |
| Radius 1 | Specifies the radius of the first circle. |
| Radius 2 | Specifies the radius of the second circle. |

**Ring sector**

A circle ring sector is defined by five definition points:

| | |
|---|---|
| Mid-point | Specifies the common mid-point for both circles. |
| Radius 1 | Specifies the radius of the first circle. |
| Radius 2 | Specifies the radius of the second circle. |
| Start-angle | Specifies the start-angle. The radius, that runs from the mid-point through this circle sets a limit to the ring sector. |
| End-angle | Specifies the end-angle. The radius, that runs from the mid-point through this circle sets a limit to the ring sector. The ring sector will then be drawn anticlockwise from the start-angle to the end-angle. |

The corner-points of the circle ring sector will be calculated as definition points and can be used for the catch function.

By pressing the ESC key, you can take back the last point entered. Pressing the right mouse button cancels the entry (for information, see point entry and objects).

## Ellipse

Ellipses are defined by a mid-point and two vectors, which can be entered either directly (complete, any) or indirectly (complete, standard).

### Complete, standard

A standard complete ellipse is defined by two definition points:

| | |
|---|---|
| Mid-point | Specifies the mid-point of the ellipse. |
| Point on frame | Specifies a point on the surrounding rectangular frame of the ellipse. The vectors of the resulting ellipse lie perpendicularily to each other. |

The standard complete ellipse will be saved as a complete ellipse, any. Therefore, only the mid-point and the end-points of the two vectors can later be changed (see e.g. move point).

### Complete, rotated

A rotated complete ellipse is defined by three definition points:

| | |
|---|---|
| Mid-point | Specifies the mid-point of the ellipse. |
| Point on frame | Specifies a point on the surrounding rectangular frame of the ellipse. The vectors of the resulting ellipse lie perpendicularily to each other. |
| Rotation angle | Specifies the rotation angle, for the already entered ellipse to be rotated around. The rotation angle is calculated, by using the radius from the mid-point through this point. |

The rotated complete ellipse will be saved as a complete ellipse, any. Therefore, only the midpoint and the end-points of the two vectors can later be changed (see e.g. move point).

### Complete, any

A complete ellipse, any, is defined by three definition points:

| | |
|---|---|
| Mid-point | Specifies the mid-point of the ellipse. |
| Point 1 on ellipse | Specifies the end-point of the first of the surrounding vectors. |
| Point 2 on ellipse | Specifies the end-point of the second of the surrounding vectors. |

### Arc

An ellipse arc is defined by four definition points:

| | |
|---|---|
| Mid-point | Specifies the mid-point of the ellipse. |
| Point on frame | Specifies a point on the surrounding rectangular frame of the ellipse. The vectors of the resulting ellipse lie perpendicularily to each other. |
| Start-angle | Specifies the start-angle. The radius from the mid-point through this point sets a limit to the arc. |

| | |
|---|---|
| End-angle | Specifies the end-angle. The radius from the mid-point through this point sets a limit to the arc. The arc is drawn anticlockwise from the start-angle to the end-angle. |

The end-points of the ellipse arc will be calculated as definition points and can be caught.

**Sector**

An ellipse sector is defined by four definition points:

| | |
|---|---|
| Mid-point | Specifies the mid-point of the ellipse. |
| Point on frame | Specifies a point on the surrounding rectangular frame of the ellipse. The vectors of the resulting ellipse lie perpendicularly to each other. |
| Start-angle | Specifies the start-angle. The radius from the mid-point through this point sets a limit to the sector. |
| End-angle | Specifies the end-angle. The radius from the midpoint through this point sets a limit to the sector The sector is drawn anticlockwise from the start-angle to the end-angle. |

The corner-points of the ellipse sector will be calculated as definition points and can be caught.

**Ring**

The ellipse ring is defined by three definition points:

| | |
|---|---|
| Mid-point | Specifies the mid-point of the two ellipses. |
| Point on frame 1 | Specifies a point on the surrounding rectangular frame of the first ellipse. The vectors of the resulting ellipse lie perpendicularly to each other. |
| Point on frame 2 | Specifies a point on the surrounding rectangular frame of the second ellipse. The vectors of the resulting ellipse lie perpendicularly to each other. |

**Ring sector**

An elliptical ring sector is defined by five definition points:

| | |
|---|---|
| Mid-point | Specifies the common mid-point of the two ellipses. |
| Point on frame 1 | Specifies a point on the surrounding rectangular frame of the first ellipse. The vectors of the resulting ellipse lie perpendicularly to each other. |
| Point on frame 2 | Specifies a point on the surrounding rectangular frame of the second ellipse. The vectors of the resulting ellipse lie perpendicularly to each other. The two ellipses may overlap each other. |
| Start-angle | Specifies the start-angle. The radius from the mid-point through this point sets a limit to the ring sector. |

| | |
|---|---|
| End-angle | Specifies the end-angle. The radius from the mid-point through this point sets a limit to the ring sector. The ring sector is drawn anticlockwise from the start-angle to the end-angle. |

The corner-points of the elliptical ring sector will be calculated as definition points and can be caught.

By pressing the ESC key, you can take back the last point entered. Pressing the right mouse button cancels the entry (for information, see point entry and objects).

## Parabola

With this command, you can draw standard parabolas.

### Standard

A standard parabola is defined by three definition points:

| | |
|---|---|
| Vertex | Specifies the vertex point of the parabola. |
| Rotation angle | Specifies the rotation angle of the parabola. The line from the vertex to this point is the symmetry axis of the parabola. |
| End-point | Specifies the end-point of one of the parabola arms. The end-point of the other arm is determined by mirroring this point by the symmetry axis. |

### 3 point

A 3 point parabola is defined by three definition points:

| | |
|---|---|
| Start-point | Specifies the start-point of the parabola, i.e. the end-point of the first parabola arm. |
| End-point | Specifies the end-point of the parabola, i.e. the end-point of the second parabola arm. |
| Point on arc | Specifies a point, through which the parabola should at least theoretically run through. Only that part of the parabola will however be drawn, that lies between the end-points and the vertex. This point is mainly intended to determine the parabola coefficients. |

By pressing the ESC key, you can take back the last point entered. Pressing the right mouse button cancels the entry (for information, see point entry and objects).

## Zigzag line

A zigzag line is defined by two definition points.

| | |
|---|---|
| Start-point | Specifies the start-point of the zigzag line. |
| End-point | Specifies the end-point of the zigzag line. |

By pressing the ESC key, you can take back the last point entered. Pressing the right mouse button cancels the entry (for information, see point entry and objects).

By using the zigzag line parameters command, you can set the indent distance. The indent form and height will be derived from the line width.

## Zigzag line parameters

Using this command, you can adjust the indent distance of the zigzag line. A dialog box will appear, where the desired indent distance can be entered.

Changing the indent distance only affects zigzag lines drawn after the adjustment is made. Already existing zigzag lines will not be changed.

## Circle arc chain

A circle arc chain is a sequence of individual circle arcs, that are linked together at the ends. The program saves it as seperate standard circle arcs, which means that each circle arc can later be changed independently (see e.g. move point).

A circle arc chain is defined by a number of definition points. After the first arc has been entered as a 3 point circle arc, each new entry of an end-point adds a new circle arc to the chain.

By pressing the ESC key, you can take back the last point entered (see also point entry and objects).

Pressing the right mouse button cancels the entry. A dialog box then appears, in which you decide how to carry on:

| | |
|---|---|
| Yes | Applies the circle arc chain in the current form and ends the entry. |
| No | Cancels the entry of the circle arc chain. Already entered points will be lost. |
| Return | Returns to the entry of the circle arc chain. Additional points can be entered. |

A circle arc chain may consist of up to 2000 definition points.

## Bézier chain

A Bézier arc chain is a sequence of individual, linked Bézier curves. Each Bézier curve is defined by four points: start-point, two pivots, end-point. Since the Bézier curves are linked, the end-point of one Bézier curve is the start-point of the next Bézier curve, so this point does not have to be entered twice.

A Bézier chain is defined by a number of definition points. After the entry of a start-point, each entry of two pivots and an end-point cause one Bézier curve to be linked to the chain.

By pressing the ESC key, you can take back the last point entered (see also point entry and objects).

Pressing the right mouse button cancels the entry. A dialog box then appears, in which you decide how to carry on:

| | |
|---|---|
| Yes | Applies the circle arc chain in the current form and ends the entry. If the last Bézier curve wasn´t finished by entering all three points, lines will be drawn through the existing points. |
| No | Cancels the entry of the circle arc chain. Already entered points will be lost. |
| Return | Returns to the entry of the circle arc chain. Additional points can be entered. |

A Bézier chain may consist of up to 2000 definition points.

## Spline

A spline curve is defined by a number of definition points. After the start-point has been entered, each entry of an endpoint adds a section to the curve.

By pressing the ESC key, you can take back the last point entered (see also point entry and objects).

Pressing the right mouse button cancels the entry. A dialog box then appears, in which you decide how to carry on:

| | |
|---|---|
| Yes | Applies the spline curve in the current form and ends the entry. |
| No | Cancels the entry of the spline curve. Already entered points will be lost. |
| Return | Returns to the entry of the spline curve. Additional points can be entered. |

A spline curve may consist of up to 2000 definition points.

## Construction menu

Mark
  - Layer
  - Parameters
Intersections
Construction line
Construction tangent
Construction circle
Division
Multiple copy
Trim line
Trim circle
Chamfer edge
Round edge
Resolve objects

## Mark

Using this command, you can insert a mark directly with the mouse. The mark is entered as a definition point and can be caught as a definition point.

Marks are always drawn with the color set with the change settings command. This color is not influenced by dyeing layers.

## Mark layer

Marks will usually be assigned to the current layer. This command can however be used to choose a layer that all new marks will be assigned to. In this way, all marks can easily be hidden and frozen.

Marks are always drawn with the color set with the change settings command. This color is not influenced by dyeing layers.

## Mark parameters

The mark parameters affect the construction and multiple copy commands.

For the construction, you can choose whether only marks or objects are to be created. In the first case, marks will be inserted instead of the definition points needed to construct objects. This is wise when the object itself is only needed as a reference object for further construction.
In the second case, the objects to be constructed will directly be inserted. When constructing circles, you also have the possibility to choose whether always to construct complete circles or circle arcs. This decides whether the variant selection should offer the choice of full circles only or a number of circle arcs instead.

As to multiple copy, you can choose whether the copied objects are also to be rotated. This affects for example the multiple copy, circle command. If the circle for the multiple copy command is divided into 8 equal parts, then the objects inserted at the first division point will become rotated by 0 degrees, the objects at the second division point by 45 degrees, the objects at the third division point by 90 degrees, etc.

## Intersections

This command enables you to calculate intersections of standard objects. The resulting intersections will be displayed as marks.

### Line - line

Calculates the intersection of two lines. To do that, both lines must be identified. If the lines run parrallel to each other, no marks will be placed.

### Line - circle

Calculates the intersections of a line and a circle. To do that, the line and the circle must both be identified. If the line and the circle don't intersect, no marks will be placed.

### Circle - circle

Calculates the intersections of two circles. To do that, both circles must be identified. If the circles don't intersect, no marks will be placed.

## Construction line

Using this command, lines can be constructed, depending on already existing objects. The results will be inserted, according to the mark parameters, either directly as a line or as two marks at the both ends of the line.

Line constructions are always unequivocal, i.e. there is always exactly one line that meets the requirements. A variant selection will therefore not be needed.

### Perpendicular on line

Drops a perpendicular on a line. To do that, you first identify the reference line onto which the perpendicular is to be dropped. Having done that, you choose the end point of the perpendicular with the mouse.

### Perpendicular on circle

Drops a perpendicular on a circle. To do that, you first identify the reference circle onto which the perpendicular is to be dropped. Having done that, you choose the end point of the perpendicular with the mouse.

### Parallel, variable

Constructs a parallel with any desired distance to another line. To do that, you first identify the reference line for the parallel. Having done that, you enter the start point and the end point of the parallel. The distance to the reference line is defined by the start point.

### Parallel, fixed

Constructs a parallel with a fixed distance to another line. To do that, you first identify the reference line for the parallel. A dialog box then appears, in which you enter the distance. Finally, enter the start point and the end point of the parallel with the mouse.

## Construction tangent

This command enables you to construct a tangent. The result will, depending on the mark parameters, be inserted either directly as a line or as two marks at both ends of the line.

The construction of tangents is not always unequivocal, i.e there may be a number of possible lines that meet the requirements. If that is the case, a variant selection must be made. For some commands, depending on the mark parameters, it is also possible to choose reference circles or circle arcs in addition to the tangents.

If no variant exists that meets the requirements, a message appears and the command will not be carried out.

### Circle - point

Places a tangent on a circle, starting from a fixed point. To do that, you first identify the reference circle on which the tangent is to be placed. Then enter the point where the tangent is to start.

In addition to the two tangents, the variant selection also offers the choice between a reference circle and its circle arc.

### Circle - circle outside

Constructs the outside tangent common to two circles. To do that, you need to identify the two circles on which the tangent is to be placed.

In addition to the two tangents, the variant selection also offers the choice between the reference circles and their circle arcs.

### Circle - circle inside

Constructs an inside tangent common to two circles. To do that, you need to identify the two circles on which the tangent is to be placed. If the circles overlap so that no inside tangent exists, the outside tangent will be drawn.

In addition to the two tangents, the variant selection also offers the choice between the reference circles and their circle arcs.

## Construction circle

Depending on already existing objects, this command enables you to construct circles. The results will, according to the mark parameters, either be entered directly as a circle or a circle arc, or as marks.

The construction of circles is not always unequivocal, there may in other words exist a number of possible circles that meet the requirements. If this is the case, a variant selection must be made. Most commands allow you the choice between a full circle and a circle arc, depending however on the mark parameters.

If no variant exists that meets the requirements, a message appears stating this and the command will not be carried out.

### Circumcircle

Constructs the circumcircle of a triangle. To do that, enter the three corner points of the triangle with the mouse. If an object is to be constructed at all, a full circle will always be inserted, regardless of the mark parameters.

Note: The midpoint of the circumcircle is the intersection of the mid-perpendiculars of the triangle. The command can therefore also be used to construct the intersection of two mid-perpendiculars.

### Incircle

Constructs the incircle of a triangle. To do that, enter the corner points of the triangle with the mouse. If an object is at all to be constructed, a full circle will always be inserted, regardless of the mark parameters.

Note: The midpoint of the incircle is the intersection of the bisectors of the angles of the triangle. The command can therefore also be used to construct the intersection of two bisectors.

### Perpendicular on line

Constructs a circle that is tangent to a line. To do that, you need to identify the reference line the circle should be tangent to. Then enter the midpoint of the circle with the mouse. If an object is to be constructed in the first place, a full circle will always be inserted, regardless of the mark parameters.

### Perpendicular on circle

Constructs a circle that is tangent to another circle. To do that, you first identify the reference circle the circle should be tangent to. Then enter the midpoint of the circle to be constructed with the mouse. If an object is to be constructed in the first place, a full circle will always be inserted, regardless of the mark parameters.

## Division

This command enables you to carry out various divisions. It creates marks. All divisions are based on the division of a line. It needs not to be a straight line, but may e.g. run along a circle arc.

There are six different types of dividing the line. A dialog box appears, in which various parameters should be entered, depending on the desired division type.

| | |
|---|---|
| Linear | Divides the line into a given number of equal segments. To do that, you must specify the number of the segments. The number should to lie between 1 and 999. |
| Logarithmic | Divides the line logarithmically into a given number of decades. To do that, you must specify the number of the decades. The number should to lie between 1 and 99. |
| Golden section | Divides the line a number of times according to the golden section by dividing each time the larger part again according to the golden section. To do that, you need to specify the number of divisions, which should lie between 1 and 999. |
| Geometric | Divides the line by a geometrical succession. To do that, you specify a start length that determines the length of the first section. Having done that, you either specify the factor that is used to multiply this start length with for the next section, or you specify the number of sections the line should be divided into. This number should lie between 1 and 999.<br><br>Note: By a given number of sections, an infinite geometric succession will be assumed, since the calculations would otherwise become too complicated. This has the effect that the line may not be completely used, especially when the number of sections is a small one. |
| Arithmetic | Divides the line by an arithmetic succession. To do that, you need to specify a start length to determine the length of the first section. Then, you either specify the summand to be added to this start length for each additional section, or you specify the number of sections the line is to be divided into. This number should lie between 1 and 999.<br><br>Note: By an arithmetic succession it may happen, that dividing the line into a fixed number of sections is only possible if the last section length becomes negative. If that is the case, only those sections will be inserted which lie on the given line. The number of sections may be significantly lower than the disired number. |
| Relation | Divides the line proportionally as specified by a number of numeric values. Up to 20 numeric values may be specified. If you enter for example the values 40, 100 and 60, the line will be divided into three sections. The length of the first section will be 20% of the length of the line, the lenght of the second section will be 50% and the length of the third section 30%.<br><br>Four buttons are available for editing the list of number values:<br><br>Delete — Deletes the selected entry from the list.<br>Insert — Inserts the value in the input box right before the selected entry. |

| | |
|---|---|
| Replace | Replaces the selected entry with the value in the input box. |
| Append | Adds the value at the end of the list. |

You select the disired division type by choosing one of the "Active" option buttons. All other values will be retained to make possible to shift quickly between different division types.

**Line**

Divides a line. The line is defined by a reference line that has to be identified.

**Circle**

Divides a full circle. The circle is defined by a reference circle that has to be identified.

**Array**

Partitions an array. You enter the array with the mouse like a parallelogram. Two dialog boxes will then appear, one after the other, in which you should specify the division of each of the two edges of the array, one at a time.

## Multiple copy

With this command you can copy the selected objects to a number of positions at the same time. Most of the commands used for this purpose are based on the division commands.

When copied, the objects may be rotated. This depends on the mark parameters.

**Rotation**

The selected objects are rotated several times around a point to be entered. The number of copies can be determined by entering the number of circle sections. This number may lie between 2 and 99.

This command offers a fast and easy way to rotate objects that already lie in the right position. The selected objects will be considered, so only five new copies will have to be generated when six copies are needed.

The objects will be rotated in any case, regardless of the mark parameters.

**Distance**

Copies the selected objects to all positions as determined by the division of a line. In addition to the input needed for the division, distance command, you must choose a reference point to determine the position within the selected objects where the new position within the copied objects is to be.

The objects will not be rotated, regardless of the mark parameters.

**Circle**

Copies the selected objects to all positions as determined by the division of a circle. In addition to the input needed for the division, circle command, you must choose a reference point to determine the position within the selected objects where the new position within the copied objects is to be.

**Array**

Copies the selected objects to all positions as determined by the partition of an array. In addition to the input needed for the division, array command, you must choose a reference point to determine the position within the selected objects where the new position within the copied objects is to be.

The objects will not be rotated, regardless of the mark parameters.

## Trim line

These commands enable you to make changes to existing lines. If you want to use them on standard objects that are composed of lines, these objects must first be divided up into lines with the resolve objects command.

For all trim line commands, the line to be trimmed must be selected. Only one line may be selected at a time. Other objects may be selected as well, since they will be ignored.

There are a number of possibillities to change lines by trimming. By an intersection with two lines, for example, three seperate lines will be created. Each one can then be selected as variants. If two connected line parts are selected together, they will always be inserted as two seperate lines and not as one continous line.

If the object to be trimmed is member of a group, this group will be released first!

As a rule, trimming causes the original object to be deleted. This can be prevented by activating the duplicate.

### Cut with perpendicular

Divides up the selected line at the foot of a perpendicular. To do that, enter with the mouse the point from which the perpendicular is to be dropped on the line.

### Cut with 2 perpendiculars

Divides up the selected line at two perpendicular foot points. To do that, enter with the mouse the two points from which the perpendiculars are to be dropped on the line.

### Intersection with line

Divides up the selected line at the intersection with another line. To do that, identify the reference line for the cut.

If the selected line and the reference line are parallel to each other and consequently don't intersect, a message will state this and the command won't be carried out.

### Intersection with 2 lines

Divides up the selected line at the intersections with two other lines. To do that, identify the two reference lines for the cut.

If the selected line and the reference lines are parallel to each other and consequently don't intersect, a message will state this and the command won't be carried out.

### Intersection with circle

Divides up the selected line at the intersections with a circle. To do that, identify the reference circle for the cut.

If the selected line doesn't cross the circle and consequently no intersection exists, a message will state this and the command won't be carried out.

## Trim circle

These commands enable you to make changes to existing circles, circle arcs and circle sectors. If you want to use them on other standard objects, that are composed of circles and circle arcs, you must first divide these objects up into circles and lines with the resolve objects command.

For all trim circle commands, the circle, circle arc or circle sector to be trimmed must be selected. Only one circle, circle arc or circle sector may be selected at a time. Other objects may be selected as well, since they will be ignored.

There are a number of possibillities to change circles, circle arcs or circle sectors by trimming. By an intersection with a line, for example, two seperate arcs will be created. Each one can then be selected as variants. If two connected arcs are selected together, they will in every case be inserted as two seperate arcs and not as a full circle.

If the object to be trimmed is member of a group, this group will be released first!

As a rule, trimming causes the original object to be deleted. This can be prevented by activating the duplicate.

### Change radius

Gives the circle, circle arc or circle sector a new radius. A dialog box first appears, in which you enter the new radius. When the dialog box first opens, the current radius is displayed in it.

### Change angles

Changes the circle into a circle arc, or gives the circle arc a new opening angle. A dialog box appears, in which you enter the new opening angle. When the dialog box first opens, the current opening angles are displayed in it (0 if the object is a circle).

### Extent to point

Gives the circle, circle arc or circle sector a new radius. The new radius will be calculated from a reference point. Enter the reference point with the mouse. The circle will then be drawn so as to run through this reference point.

### Extent to line

Gives the circle, circle arc or circle sector a new radius. The new radius will be calculated from a reference line, which first must be identified. The circle will then be drawn so as to be tangent to that reference line.

### Extent to circle

Gives the circle, circle arc or circle sector a new radius. The new radius will be calculated from a reference circle, which first must be identified. The circle will then be drawn so as to be tangent to that reference circle.

### Cut with 2 perpendiculars

The circle, circle arc or circle sector will be divided into two seperate arcs. These arcs will be calculated from the intersections with two perpendiculars. Enter two points with the

mouse, where the perpendiculars are to be dropped on the circle. This entry corresponds to the angle definition in circle arcs, standard. However, both of the seperate arcs may be selected.

**Intersection with line**

Divides the circle, circle arc or circle sector into two seperate arcs. These arcs will be calculated from the intersections with a straight line. The reference line, with which the circle is to intersect, has to be identified.

If no intersection exists because the reference line does not cut the selected circle, a message appears stating this and the command will not be carried out.

**Intersection with circle**

Divides the circle, circle arc or circle sector into two seperate arcs. These arcs will be calculated from the intersections with a circle. The reference circle, with which the selected circle is to intersect, has to be identified.

If no intersection exists, because the reference circle does not cut the selected circle, a message appears stating this and the command will not be carried out.

## Chamfer edge

This command enables you to chamfer a corner that is made of two lines. A diagonal line will be inserted in the corner.

To chamfer a corner, the two lines that constitute the corner must be selected. Exactly two lines must be selected. Other objects may be selected as well, since they will be ignored.

If you want to use this command on other standard objects that are made up of lines, you must first divide up those objects into lins with the resolve objects command.

A dialog box appears, where you should enter the length and the inclination angle of the chamfer. This usually gives eight different possibilities. The variant selection then enables you to choose between them. You should only choose **one** variant at a time.

The selected chamfer will be inserted and the selected lines will be shortened or lenghtened until they touch the chamfer line.

If no possibility exists to place the chamfer, a message will state this and the command will not be carried out.

If the selected lines are members of groups, these groups will be released first!

The chamfer usually causes the two original lines to be deleted. This can be prevented by activating the duplicate.

## Round edge

This command enables you to round a corner that is made of two lines. This means inserting a circle arc in the corner.

To round a corner, the two lines that constitute the corner must be selected. Exactly two lines must be selected. Other objects may be selected as well, since they will be ignored.

If you want to use this command on other standard objects that are made up of lines, you must first divide up those objects into lines with the resolve objects command.

A dialog box appears, where you should enter the radius of the rounding. This usually gives eight different possibilities. The variant selection then enables you to choose between them. You should only choose **one** variant at a time.

The selected rounding will be inserted and the selected lines will be shortened or lenghtened until they reach exactly to the rounding.

If no possibility exists to place the rounding, a message will state this and the command will not be carried out.

If the selected lines are members of groups, these groups will be released first!

The rounding usually causes the two original lines to be deleted. This can be prevented by activating the duplicate.

## Resolve objects

This command enables you to disassemble standard objects and outlines into lines, circles, circle arcs and Bézier chains. This is e.g. necessary when you want to manipulate created objects with the trim commands.

All objects that are to be disassembled must be selected. Only standard objects and outlines will be affected.

Since the selected objects are divided into seperate objects that are in no way connected to each other, the fillings of the original objects can not be retained.

When disassembling, the memory needed usually increases enormously. Especially when disassembling hatches into seperate lines, the need for memory becomes up to four times more than usual. You should therefore only use this when absolutely necessary.

The initial objects will usually be erased. This can be prevented by activating the duplicate.

## Library menu

Insert symbol
  - Layer
Read symbol
  - Parameters
Delete library
  - Edit
  - Scale
Symbol info
New library
  - Load
  - Choose
  - Save
  - Save as
  - Close
Library details

## Insert symbol

A dialog box appears, in which you choose a symbol to insert.

| | |
|---|---|
| Number | You may enter here the number of the desired symbol. This text box always displays the number of the currently selected symbol. |
| Factor | This value determines the factor the symbol will be scaled with before inserting it into the drawing. |
| Angle | This value determines the rotation angle for the insertion of the symbol. |
| Display symbol | If this check box is active, the currently selected symbol will be displayed in the rectangular box within the dialog box. |
| Cancel | The symbol choice is cancelled, no symbol will be inserted. |
| OK | The currently selected symbol, whose number is displayed is the "Number" text box, is selected and kann be inserted. The choice can also be done by double clicking in the list box. |

If a symbol has been selected, a crossed rectangle appears at the position that was selected when the symbol was read in.

By clicking at the desired position in the drawing area, the symbol becomes inserted there. This procedure can be repeated until a new command has been chosen.

## Symbol plane

Symbols are usually placed to the current layer.  With this command you can choose one layer onto which all symbols are to be placed.  In this way, all symbols can easily be hidden or frozen.

## Read symbol

With this command, you can define all selected objects as a symbol and insert this symbol into the current library.

Reading a symbol depends on the settings in the symbol parameters. Usually, you first choose a point to determine a position in the symbol where the cursor position is to be when you later make the insertion. When this has been done, a dialog box appears where you enter the symbol name. The symbol, i.e. all selected objects, will then be added to the library.

There are different ways of adding a symbol to the library:

| | |
|---|---|
| Append | The new symbol will be appended to the library. |
| Insert | The new symbol will be inserted before the currently selected symbol, whose number is displayed in the "Number" text box. |
| Replace | The currently selected symbol, whose number is displayed in the "Number" text box, will be replaced by the new symbol. |
| Cancel | The reading process is cancelled, no symbol will be added to the library. |

If the "Area" option button in the symbol parameters is active, two points must be entered to read in a symbol. The first one again determines a position in the symbol where the cursor position is to be when you later make the insertion. The second point determines by its distance to the first one the width and height of the symbol. These specifications are important if the symbol is to represent a font character or if it is used as a hatching symbol.

In this case, the symbol may be empty, i.e. it contains no objects. A warning will appear when reading an empty symbol, but clicking on "OK" continues reading the symbol. This is especially important when reading the space character for a font. Simply enter the desired size of the space, without any object selected.

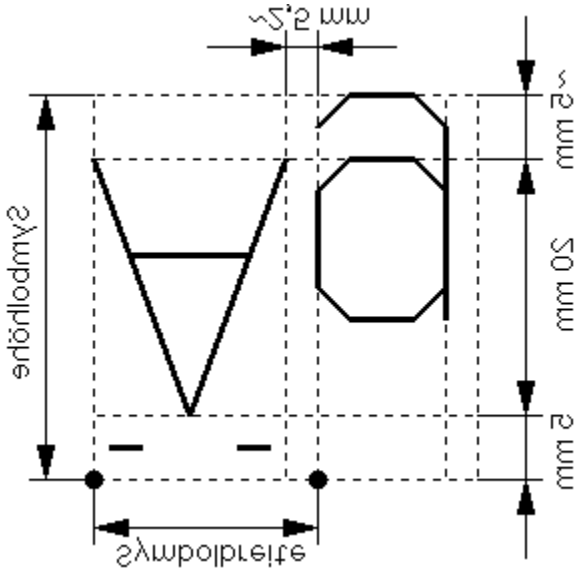
A font library (e.g. the enclosed DINDRAFT) is a library of symbols that represent characters. The intervals between characters are determined by the height and the width of the symbol.

Font libraries contain font characters that correspond to the ANSI character set. Usually, you insert a large number of them simultaneously by using the line or text editors. Or you use them as a font for dimensioning.

To make possible an automatic line-up of characters, in addition to the information about the size of the symbol, other information is needed as well, e.g. the character spacing. The origin inside a symbol determines the upper left hand corner of the character cell, the symbol width and symbol height determine the width and height of the character cell. The character cell is the rectangular region a character occupies within a text, i.e. the size of the character itself and the spacing to the next character.

For the characters used in TOMMYSOFTWARE® CAD/DRAW, the height of the character cells is predefined as 30 millimeters. The baseline should lie 25 millimeters from the upper edge of the cell. The upper edge of the cell should lie 5 millimeters above the highest characther (diacritical marks not included). This means: "A" is 20 millimeters high and its distance to the upper as well as to the lower edge of the cell is 5 millimeters. The width of the character cell should be equal to the width of the character plus 2.5 millimeters. These predefinitions are necessary in order to make sure that height specifications of the text made when

working with the program can also be correctly converted. The following graphic displays these values.



If you want to use a symbol as a character in a font, its symbol number must correspond to the number of the corresponding ANSI character (an "A" has for example the number 65). Not all numbers need to be available. If a character needed is not found in the library when a text is being converted, it will be ignored. A list of the ANSI characters can be found in the Windows User's Guide.

Font used for dimensioning have to include following special characters in addition to the standard characters ('A'..'Z', 'a'..'z', punctuation etc.):

| | |
|---|---|
| Number 128 | The character 128 is used as a symbol for area measurement. It should be a small square (height of a noncaptial letter). This character is not supported by Windows directly and will be displayed as a thick, vertical line in all edit boxes. |
| Number 176 | The character '°' is used as a symbol for the unit degree when dimensioning angles. |
| Number 216 | The character 'Ø' is used as a symbol for diameter measurements. |

## Symbol parameters

| | |
|---|---|
| Point | If this option button is active, you only need to enter one point to read in a symbol. This point determines a position in the symbol where the cursor position is to be when you later make the insertion. |
| Area | To read in a symbol, two points need to be entered. The first one again determines a position in the symbol where the cursor position is to be when you later make the insertion. The second point determines by its distance to the first one the width and height of the symbol. These specifications are important if the symbol is to represent a font character. A font library (e.g. the enclosed DINDRAFT) is a library of symbols that represent characters. The intervals between the characters are determined by the height and width of the symbol. |

For general information on libraries and symbols, see libraries.

## Delete library

If necessary, a security check appears before deleting the current library.

## Edit library

This command enables you to work on an already existing library. A dialog box appears:

| | |
|---|---|
| Display | If this check box is active, the currently selected symbol, whose number is displayed in the "Number" text box, will be displayed in the rectangular box within the dialog box. |
| Name | A dialog box appears, in which you can enter a new symbol name. The name may be up to 63 characters long. If you leave the box via the "OK" button, the new name appears in the corresponding line of the list box. |
| Erase | If necessary, a security check appears before erasing the current symbol. |
| Move | A dialog box appears, in which you enter a number for the current symbol. The list will then be sorted again. If the number already exists, or if it is smaller than 1 or larger than 900, an error message appears and the entry will be ignored. |
| Renumber | Renumbers the entire list, starting with the current symbol. A dialog box appears, in which a start-value should be entered. All following symbol numbers will be raised by one, starting from the value entered. |

## Scale library

A dialog box appears, in which the desired scale factor can be entered. All symbols in the current library will be scaled.

## Symbol info

The symbol info functions in almost the same way as the object info. The only difference is, that here the activated settings apply to the objects of the current library.

## New library

If necessary, a security check will appear before opening a new library with the name "(UNTITLED)".

It is possible to load the old library again in a quick and easy way. For information, see choose library.

## Load library

If necessary, a security check will appear before loading a new library.

It is possible to load the old library again in a quick and easy way. For information, see choose library.

## Choose library

A dialog box appears, in which the eight last edited libraries (including the current one) are displayed. The first library on the list is the current one, second on the list is the library edited before that, etc.

Clicking on a name loads the corresponding library. If necessary, a security check appears first.

A new library is added to the list whenever e.g. a new library is opened or a library is loaded. If there are already eight libraries on the list, the last one on the list will be removed.

Only libraries with a name are kept on the list. If a new library is opened and changed, a security checkIDH_TERM_SAVEMESSAGE appears, as always, before loading the selected data. If you don't save the library now, it won't be added to the list since it only gets a name if it is saved.

## Save library

Saves the current library.

If it doesn't have a name already, a file selector box appears, in which you choose a name for the new library. If a file already exists with the name chosen, a dialog box enables you to cancel or confirm the saving.

A dialog box will then appear, in which you can enter some library details.

## Save library as

A file selector box appears, in which you can choose a name for the file. If a file already exists with the name you have chosen, a dialog box enables you to cancel or confirm the saving.

A dialog box will then appear, where you can enter some library details.

Finally, you have possibility to apply this name as the new name to the current library.

## Close library

If necessary, a security check appears before closing the current library. It will then be removed from the list of the last worked on libraries, which means that then you can't choose it with the choose library command.

The next library on the list of the last worked on libraries will then be loaded. If there are no more libraries on the list, a new library opens.

## Library details

A dialog box appears, in which the details of the current library can be viewed and changed.

| | |
|---|---|
| Title | This text should contain the library name. |
| Topic | This text should contain the library topic, e.g. its application field. |
| Author | This text should contain the name of the person who created the library. Default is the name of the person TOMMYSOFTWARE® CAD/DRAW has been installed on. |
| Creation time | This entry shows the time at which the library was created, i.e. the time when it was opened with the new library command. |
| Change author | This text contains the name of the person who made the last changes to the file. Default is the name of the person TOMMYSOFTWARE® CAD/DRAW has been installed on. It can however be changed if desired. |
| Last change | This entry shows the time at which the library was last changed, i.e. the time when it was last saved. |
| Comment | This text is intended for any remarks and comments desired. |

## Selection menu

Selection mode
Select object
  - Deselect
Select area
  - Deselect
  - Invert
Conditional selection
Conditional deselection
Build group
Release groups
Move point
Select points
  - Deselect
  - Move
Discard point selection

## Selection mode

Decides whether the primary or secondary selection is active. The two selection modes differ in the color of the selection frame when they are displayed - see change settings under "Standard system colors". The active selection mode is displayed in the status window.

As a rule, the primary selection should be active. The secondary selection is needed almost only for the hatch objects command, whereas all commands in the object menu only work with primary selected objects. The secondary selection should only be used when this is expressly required.

For general information, see also selection.

On selection, see also layer status and lock objects.

## Select object

With this command, you can select objects directly by clicking on them with the mouse. To do that, you must click in the rectangular area where the object is located (the selection area). The selection is indicated by a frame around the selected object (the selection frame).

An object can also be selected with the keyboard, see fixed keyboard functions. Pressing either the F11 or the Backspace key preselects the object that is nearest to the crosshair, i.e. marks with a frame. Should the preselected object not be the desired object, pressing the F11 or Backspace key again preselects the nextnearest object, and so forth. When the desired object has been preselected, pressing F12 or the Insert key selects the preselected object. Pressing the Shift+F11 or Shift+Backspace key combination deletes the markings. Only not yet selected objects become preselected. Should the preselected object belong to a group, all other objects in the group will automatically be preselected as well.

Objects will be either primary selected or secondary selected, depending on which selection mode is active at the time.

Should the selected object belong to a group, all other objects in the group will automatically be selected as well.

## Deselect object

With this command, you can deselect objects directly by clicking on them with the mouse. To do that, you must click in the rectangular area where the object is located (the selection area).

An object can also be deselected with the keyboard, see fixed keyboard functions. By pressing either the F11 or the Backspace key, the object that is nearest to the crosshair will be preselected, i.e. marked with a frame. Should the preselected object not be the desired object, pressing again the F11 or Backspace key preselects the nextnearest object, and so forth. When the desired object has been preselected, pressing the F12 or Insert key deselects the preselected object. Pressing the Shift+F11 or Shift+Backspace key combination deletes the markings. Only not yet deselected objects will be preselected. Should the preselected object belong to a group, all other objects in the group will automatically be preselected as well.

Should the deselected object belong to a group, all other objects in the group will automatically be deselected as well.

## Select area

### All

Selects all objects in the drawing.

### Interior

Enter two points to determine an area. All objects that lie completely within that area will be selected. Should at least one member of a group lie completely within the chosen area, all other objects of the groups will also be selected, regardless of their position.

### Overlapping

Enter two points to determine an area. All objects that overlap into that area will be selected. If at least one object of a group lies partly inside the chosen area, all other members of that group will automatically be selected as well, regardless of their position.

The objects will either be primary or secondary selected, depending on which selection mode is active at the time.

## Deselect area

### All

Deselects all objects of the drawing.

### Interior

Enter two points to determine an area. All objects that lie completely within that area will be deselected. Should at least one member of a group lie completely within the chosen area, all other objects of the groups will also be deselected, regardless of their position.

### Overlapping

Enter two points to determine an area. All objects that overlap into that area will be deselected. If at least one object of a group lies partly inside the chosen area, all other members of that group will automatically be deselected as well, regardless of their position.

## Invert area

### All

Inverts the selection of all objects in the drawing, i.e. all objects that were selected before will subsequently be deselected and vice versa.

### Interior

Enter two points to determine an area. The selection of all objects that lie completely within that area will be inverted. If at least one object of a group lies completely within the chosen area, the selection of all other members of the group will automatically be inverted, regardless of their position.

### Overlapping

Enter two points to determine an area. The selection of all objects that lie partly within that area will be inverted. If at least one object of a group lies partly within the chosen area, the selection of all other members of the group will automatically be inverted, regardless of their position.

The invertion of the selection is always applied to the momentarily active selection mode only. It does not affect objects that are selected in the other selection mode.

## Conditional selection

See select area for the meaning of all, interior, and overlapping.

A dialog box appears, in which you can activate settings that represent the attributes of objects.

Each setting has a "Consider" check box; when it is active, the setting will be considered. If you make changes to one of the settings, its check box will automatically be activated. After leaving the dialog box by clicking the "OK" button, considering a setting has the effect that all objects will be selected that have all of the attributes to be considered.

**Line style, filling style**

The meaning and function of these settings correspond to the ones in the object info.

**Object**

In the list box you can choose the names of the objects that are to be selected. There is no check box for this setting. It will be considered for the selection if at least one name is chosen.

**Layer**

You can enter a layer number to be considered by the selection.

The objects will either be primary or secondary selected, depending on which selection mode is active at the time.

## Conditional deselection

See deselect area for the meaning of all, interior, and overlapping.

A dialog box appears, in which you can activate settings that represent attributes of objects.

Each setting has a "Consider" check box; when it is active, the setting will be considered. If you make changes to one of the settings, its check box will automatically be activated. After leaving the dialog box by clicking the "OK" button, considering a setting has the effect that all objects will be deselected that have all of the attributes to be considered.

**Line style, filling style**

The meaning and function of these settings correspond to the ones in the object info.

**Object**

In the list box you can choose the names of the objects that are to be deselected. There is no check box for this setting. It will be considered for the deselection if at least one name is chosen.

**Layer**

You can enter a layer number to be considered by the deselection.

## Build group

Assembles all selected together into one group.  If some of the objects should already be assembled into a group, that group will first be released and then assembled into a new group along with the other selected objects.

For general information, see groups.

## Release group

All selected groups will be released. Objects that don't belong to any group, but are nonetheless selected when the command is carried out, will not be affected.

For general information, see groups.

## Move point

With this command, an individual definition point can be moved.

Before doing that, you should set the point display with the display modes command. You set the color of a definition point, the frame color and filling color of a selected point with the change settings command.

First, click on the desired definition point. This point will automatically be caught (see fixed keyboard functions), so you only have to click near to the desired point. Next, click on a position in the drawing area where the definition point should be moved to.

The definition point will then be moved to that position and the object belonging to it will be redrawn.

If several definition points lie on the same position, they will all be moved, i.e. several objects might be changed. The position of other definition points within these objects will not be affected.

## Select points

Enter two points to determine an area.  All definition points within that area will become selected.  The selected definition points may belong to different objects.

You can set the frame color and the filling color of a selected point with the change settings command.

## Deselect points

Enter two points to determine an area.  All definition points within that area will become deselected.

## Move points

First, enter a point to serve as a reference point for moving the selected points. A crossed rectangle then appears that represents the selected definition points. By using the previously selected reference point, you can now move this rectangle. By clicking with the mouse, the definition points will be moved to the chosen position.

The definition points will then be moved to that position and the object or objects belonging to it will be redrawn. The definition points remain selected after being moved.

The position of not selected definition points won't change.

## Discard point selection

Deselects all selected points (see select points and move points).

## Special menu

Hatch objects
- Parameters
- Types
- Layer

Dimension line DIN 406
Measure DIN 406
Dimension DIN 406
- Parameters
- Layer

Dimensioning info
Edit additional texts
Update dimension

## Hatch objects

The hatching is affected by the following commands: Hatch objects parameters, hatch objects types and hatch objects layers.

All selected objects will be hatched. The selection mode is of crucial importance for the three hatch modes.

For more information, see hatching.

### Union

Hatches all selected objects. In this hatch mode, primary and secondary selected objects are treated as identical.

### Intersection

Hatches the intersection of primary and secondary selected objects, i.e. the area covered by both primary and secondary selected objects.

### Difference

Hatches the difference between primary and secondary selected objects, i.e. all primary selected objects will be hatched with the exception of areas covered by secondary selected objects.

The hatching is an object like any other (e.g. a line or a circle). It can therefore be selected and manipulated at any time.

## Hatch objects parameters

A dialog box appears, in which you can change the current standard hatching.

A standard hatching consists of two hatches: Hatch 1 and hatch 2. These in turn consist of 8 lines each. The display of each line can be turned on and off individually. Not displaying a line means that this line will not be displayed while hatching, but the line distance will remain.

In addition to that, a line style can be determined for each line (for information, see drawing parameters) by clicking in the box framed in screen background color. The distance and the inclination angle of the lines can also be edited.

Each hatch as a whole can be activated or deactivated at will. If a hatch is deactivated, its lines will not be considered by the hatching.

## Hatch objects types

This command enables you to set the current hatch type on "Standard" or "Library".

| | |
|---|---|
| Standard | If this option is active, the hatching will be carried out in the current standard hatching (see hatch objects parameters).<br><br>When this hatch type gets active, a possibly existing hatching library in memory will be erased. |
| Library | The hatching will be carried out using the symbol whose name is displayed in the corresponding text box. The library containing that symbol is also displayed in the text box. If you click on one of the text boxes, a dialog box appears that enables you to change the corresponding settings. Hatching is carried out with lines only, all objects in the symbol will therefore be converted to lines. In addition to standard objects, also ellipses, ellipse arcs, ellipse sectors, ellipse rings and ellipse ring sectors, as well as parabolas, Bézier chains, splines and outlines can be used. Fillings will be ignored. For information about the creation of hatching libraries see read symbol.<br><br>When this hatch type gets active, the desired hatching library will be loaded into memory. If this fails, the hatch type "Standard" becomes active. |

Using this command, you can also choose or define standard hatchings. If the "Apply" button is activated, the name from the input box will be applied to the selected line in the list box, and the current standard hatching will be assigned to it.

Doubleclicking on a line in the list box sets the corresponding standard hatching. Clicking on the "OK" button sets the standard hatching whose name is selected in the list box.

## Hatch objects layer

The hatch layer determines the layer onto which the hatch is to be inserted. The hatch can be inserted either to the current layer or to a fixed layer.

## Dimension line DIN 406

With this command dimension lines can be created without having to make a dimensioning first.

Together with the measure commands, you can create dimensionings that can't be created directly with the dimension commands.

Some settings in the dimension parameters affect the appearance of the dimension lines.

**Straight**

Draws a straight dimension line. The two end points of the dimension line must be entered with the mouse.

**Curved**

Draws a curved dimension line. Enter it as a circle arc, standard.

## Measure DIN 406

With this command, you can locate dimensionings to be inserted in the drawing as a text (see dimensioning). In contrast to the dimension commands, no dimension lines or extension lines are drawn.

The measuring is carried out in two steps. In the first step, you enter or identify the reference points or objects for the dimension to be calculated from. This entry is different for each command.

In the second step, you insert the dimension number to the drawing. A crossed rectangle then appears, indicating the dimension number. To insert the dimension number, you move this rectangle to the desired position and click with the left mouse button.

The dimension number will be affected by some of the settings in dimension parameters and the settings in dimension font.

### Distance

Measures a distance. To do that, you need to identify a reference line that specifies the distance to be measured. The additional texts (see edit additional texts) are empty.

### Radius

Measures a radius. To do that, you need to identify a reference circle or circle arc that specifies the radius to be measured. Then enter the desired direction of the dimension number. The pre-text (see edit additional texts) contains an 'R', the other additonal texts are empty.

### Diameter

Measures a distance. To do that, you need to identify a reference line that specifies the distance to be measured. A diameter symbol "Ø" will be inserted in the pre-text (see edit additional texts). The other additional texts are empty.

### Angle

Measures an angle, as specified by its two sides. To do that, you need to enter the common point of both sides and the two end points of the sides with the mouse. The first line can also be identified as a line. A degree symbol "°" will be inserted in the post-text (see edit additional texts). The other additional texts are empty.

### Arc length

Measures an arc length. To do that, you need to identify a reference arc that specifies the circle arc to be measured. The additional texts are empty (see edit additional texts). When drawn, the dimension number will be overlined.

### Coordinates

Specifies a coordinate. To do that, you need to enter a reference point with the mouse. The coordinates will be calculated according to the current scale origin. Characters that represent a pair of coordinates will be inserted to the additional texts (see edit additional texts).

**Area**

Calculates the area of standard objects and of outlines. The areas of all primary selected objects will be put together, and then the areas of all secondary selected objects will be subtracted from the result.

The area is calculated object by object, i.e. overlapping parts will be considered several times!

A small square will be inserted in the pre-text (see edit additional texts) as a symbol for the area. The other additional texts are empty.

In contrast to all other dimensionings, this dimensioning is not associative, i.e. in no case does changing one or more of the objects it has been derived from affect the dimensioning, not even an explicit update dimension.

## Dimension DIN 406

This command enables you to locate dimensionings that will then be inserted in the drawing as a text (see dimensioning). In contrast to the measure commands, dimension lines and, if needed, extension lines are drawn.

The measuring is carried out in three steps. In the first step, you enter or identify the reference points or objects which the dimension to be calculated from. This entry is different for each command.

In the second step, you position the dimension line. To do that, the line drawn must be moved to the desired position and inserted by clicking with the left mouse button. The positions the dimension line can be moved to may be restricted by using the dimension parameters command. The appearance of the dimension line is affected by dimension parameters.

In the third step, you insert the dimension number in the drawing. A crossed rectangle then appears, representing the dimension number. To insert the dimension number, you move this rectangle to the desired position and click with the left mouse button. The dimension number is always so rotated, that it lies parallel to the dimension line and can be read either from the right or from below (rotation angle between -90° and 90°). The positions to which the dimension number can be moved to, can be restricted by using the dimension parameters command. The appearance of the dimension number is affected by dimension font.

### Distance

Dimensions a distance. To do that, you need to identify a reference line that specifies the distance to be dimensioned. The dimension line will then be positioned.

Extension lines, if needed, are drawn parallel from the end points of the reference line to about 2 mm beyond the dimension line.

The additional texts (see edit additional texts) are empty.

### Radius

Dimensions a radius. To do that, you need to identify a reference circle or circle arc that specifies the radius to be dimensioned.

The dimension line will then be positioned. First enter the desired orientation, then the end-point of the dimension line. The dimension line will be drawn from the entered end-point to the circle resp. circle arc. A dimension arrow will only be drawn at the end on the circle resp. circle arc.

The pre-text (see edit additional texts) contains an 'R', the other additonal texts are empty by default.

### Diameter

Dimensions a diameter. To do that, you need to identify a reference line or circle that specifies the diameter to be dimensioned.

The dimension line will then be positioned. First enter the desired orientation, then position the dimension line.

Extension lines, if needed, are drawn parallel from the end points of the reference line to about 2 mm beyond the dimension line.

A diameter symbol "Ø" will be inserted in the pre-text (see edit additional texts). The other additional texts are empty.

**Angle**

Dimensions an angle, as specified by its two sides. To do that, you need to enter the common point of both sides and the two end points of the sides with the mouse. The first side can also be identified as a line. The dimension line will then be positioned.

Extension lines, if needed, are drawn parallel from the end points of the reference line to about 2 mm beyond the dimension line.

A degree symbol "°" will be inserted in the post-text (see edit additional texts). The other additional texts are empty.

**Arc length**

Dimensions an arc length. To do that, you need to identify a reference arc that specifies the circle arc to be dimensioned. The dimension line will then be positioned.

Extension lines, if needed, are drawn parallel from the end points of the reference line to about 2 mm beyond the dimension line

The additional texts are empty (see edit additional texts). When drawn, the dimension number will be overlined.

## Dimension parameters

With this command, you can set a number of parameters for dimensioning. For that purpose, a dialog box appears, in which you can view and change the settings.

The settings can be explained as follows:

**Dimension line**

Fixed distance to object

If this check box is active, whenever a dimension is entered the distance of the dimension line will be rounded off to a multiple of the nearest length. In this way, a uniform appearance of all dimensions is obtained.

Moved perpendicularly

If this check box is active, whenever you enter a dimension line it will be so moved, that the extension lines are drawn perpendicular to it. This is always the case when angles and arc lengths are to be dimensioned.

**Dimension line type**

Extension line - start If this check box is active, an extension line is drawn at the beginning of a dimension line. The extension line starts at the object edge and reaches about 2 mm beyond the dimension line.

Extension line - end Same as the last function, except the extension line is drawn at the end of a dimension line.

Arrow head - start When you click on the area where the current arrow head form for the start of the dimension line is displayed, a dialog box appears, that enables you to choose another arrow head form for that arrow head.

Arrow head - end Same as the last function, except it applies to the arrow head form at the end of the dimension line.

Line width Enter here the desired line width for the dimension line and the extension line.

**Dimension number**

Centered on dimension line

If this check box is active, whenever you enter a new dimensioning the dimension number will always be kept centered on the dimension line.

Close to dimension line

If this check box is active, whenever you enter a new dimensioning the dimension number will always be kept close to the dimension line, i.e. it will stand directly on the dimension line.

| | |
|---|---|
| Edit tolerance | If this check box is active, whenever you enter a new dimensioning a dialog box opens, that enables you to edit the additional texts and the the tolerance of that dimension (see edit additional texts). |
| Dimension Tolerances | Enter here the font sizes for the dimension number and the tolerances. The size of the tolerances should be considerably smaller than the size of the dimension itself (DIN recommends a ratio of dimension : tolerance = 1.41 : 1, e.g. 5.0 mm and 3.5 mm). |
| Fractional digits | Enter here the desired number of fractional digits of the dimension line. The numbers should lie between 0 and 9. The setting is a global one, i.e. it affects all existing dimensions, as soon as they are updated (see update dimension). |
| Color | The color box displays the current font color for dimensioning. If you click in the color box, a dialog box opens, that enables you to choose a new color for the dimension font. |

These settings generally only affect new dimension objects. Should you want to make changes to already existing dimensions, that can be done with the object info and dimension info commands.

In addition to the settings available here, the dimension font settings will also affect the dimensioning.

## Dimension layer

Dimensions are usually placed to the current layer. With this command you can choose one layer onto which all new dimensions are to be placed. In this way, all dimensions can easily be hidden or frozen.

## Dimension info

This command enables you to change those attributes of dimension objects, that cannot be changed by using the object info.

To do that, all objects whose attributes are to be changed must be selected. A dialog box appears, that enables you to set all attributes. Only those attributes will be applied, whose "Set" check box has been marked.

For information on individual attributes, see dimension parameters and dimension font.

## Edit additional texts

This command enables you to edit the texts of a dimension. To do that, exactly one dimension object has to be selected. Other objects may be selected also, since they will be ignored.

A dialog box appears, that enables you to edit all texts. To begin with, it contains the current texts of the selected object.

The texts here edited are used to represent the dimension. How the text is used depends on the dimension type. The pre-text contains for example the letter "R" when dimension radius is used. For the measure coordinates command the dimension is composed of pre-text, x-coordinates, middle-text, y-coordinates, post-text and tolerances. In this case, the pre-text contains an opening bracket "(", the middle-text a slash "/" and the post-text a closing bracket ")".

Each text may be up to 15 characters long.

For some types of dimensions, special characters are used, that might not be available directly on your keyboard. These are in detail:

| | |
|---|---|
| Area symbol | The symbol for an area measurement is a small square, it has the number 128. This character is not supported by Windows directly, so that it appears as a small vertical line within the edit boxes. This character is not available on any keyboard. To enter this character, do the following:<br><br>Press the key ALT and hold it pressed. Then type the following numbers on the numeric keypad: 0128. Finally release the ALT key, the area symbol appears. |
| Degree symbol | The degree symbol '°' has the number 176. On some keyboards, it is available directly, on some not. In this case, do the following to enter this character:<br><br>Press the key ALT and hold it pressed. Then type the following numbers on the numeric keypad: 0176. Finally release the ALT key, the degree symbol appears. |
| Diameter symbol | The diameter symbol 'Ø' has the number 216. On some keyboards, it is available directly, on some not. In this case, do the following to enter this character:<br><br>Press the key ALT and hold it pressed. Then type the following numbers on the numeric keypad: 0216. Finally release the ALT key, the diameter symbol appears. |

## Update dimension

This command enables you to update the dimension numbers, i.e. adjust them to changed conditions. You have the choice between updating automatically or only on instruction.

A dialog box with two check boxes appears:

Update automatically

If this check box is active, the dimension numbers will be automatically updated as soon as any changes made to the drawing affect the dimension object.

Update selected objects

If you activate this check box, the dimension lines of those dimension objects that are selected when you leave the dialog box via the "OK" button will be recalculated.

## Design menu

Input outline
- Modify
- Move point
- Link
- Separate
- Parameters

Freehand line
- Parameters

Line editor

Text editor

Font
- Layer

## Input outline

With this command, you enter an outline. To do that, you must enter all contours of that outline seperately, one after the other.

For each contour, you first enter the start point. A dialog box will then appear, where you must choose whether lines or Bézier curves are to be drawn first or whether to start a new contour. Further entry depends on the chosen entry possibilities.

| | |
|---|---|
| Polyline | To enter a polyline, simply enter its corner points one after the other.<br><br>When you want to finish the entry of the polyline, press the right mouse button. |
| Bézier chain | The entry of a Bézier chain depends on the outlines parameters settings. You will find a description of it there.<br><br>When you want to finish the entry of the current Bézier chain, press the right mouse button. |
| Cancel | Cancels the entry. |

After finishing an entry with the right mouse button, another dialog box will appear where you can make the following choices:

| | |
|---|---|
| New contour | Begins a new contour. The old contour will first be completed, i.e. the start and end points of the contour will, if necessary, be connected with a line to create a closed area. The start point of the new contour can now be entered with the mouse. |
| Polyline | To enter a polyline, simply enter its corner points one after the other.<br><br>When you want to finish the entry of the polyline, press the right mouse button. |
| Bézier chain | The entry of a Bézier chain depends on the outlines parameters settings. You will find a description of it there.<br><br>When you want to finish the entry of the current Bézier chain, press the right mouse button. |
| Return | Returns to the previous entry, e.g. if the right mouse button has been pressed by a mistake. |
| Cancel | Cancels the entry of an outline. No outline will be drawn. |
| Accept | Accepts the current outline. The last contour will first be completed, i.e. the start and end points of the contour will, if necessary, be connected with a line to create a closed area. |

Note: When entering an outline, the last point can always be taken back as usual by pressing ESC. When entering Bézier curves however, it may be unavoidable that pressing ESC once may take back a number of points.

## Edit outline

This command enables you to make changes to already existing outlines. Exactly one outline must be selected. Other objects may also be selected as well, since they will be ignored.

A dialog box appears, in which you can decide how to continue.

| | |
|---|---|
| Insert line | Enables you to insert a line. Choose the insertion point by clicking near it. A new line will be inserted there, which has a length of 0 to begin with. It can be extended by dragging one of its definition points (see outline move point).<br><br>It is not possible to insert a line into a Bézier curve. A message appears if you try to do that. |
| Delete line | Removes a line. To do that, choose an end point by clicking near it. The lines to the left and right of this point will be put together into one line. If a Bézier curve is on one side, the line will be removed and the Bézier curve will be lengthened to the end of the line. |
| Insert Bézier | Enables you to insert a Bézier curve. Choose the insertion point by clicking near it. A Bézier curve will then be inserted there. Initially, the Bézier curve has a minimun expansion, it can be extended by dragging one of its definition points (see outline move point).<br><br>It is not possible to insert a Bézier curve into a Bézier curve. A message appears if you try to do that. |
| Delete Bézier | Removes a Bézier curve. Choose a point on the Bézier curve by clicking near it. The Bézier curve will then be erased. If you choose the common end point of two Bézier curves, they will be put together into one Bézier curve. |
| Cancel | All changes that have been made will be discarded. Concludes the entry. |
| OK | All changes will be applied. Concludes the entry. |

After each change this dialog box appears again until you click inside the "Cancel" or "OK" button. If you want to break off the entry of a change, you can press the right mouse button. This dialog box will then also appear.

The initial outline will usually be erased. This can be prevented by activating the duplicate.

## Outline move point

This command enables you to move definition points of an outline. First click near the point to be moved. Then select the new position of the point.

Depending on the kind of point (start-point, end-point, pivot), some of the neighbouring points will be moved, too. Depending on the outline parameters, moving one end-point of a tangent might cause the other end-point of this tangent to be moved correspondingly.

The initial outline will usually be erased.  This can be prevented by activating the duplicate.

## Link outline

This command enables you to link a number of outlines into one. This is done by assembling the individual contours of all selected outlines together into one outline.

The initial outlines will usually be erased. This can be prevented by activating the duplicate.

## Separate outlines

This command divides up outlines into their individual contours. This is done by turning the individual contours of all selected outlines into seperate outlines.

This enables you for example to divide up outlines in order to remove individual contours or replace them by another contour. This makes it easier to work on complex outlines.

The initial outlines will usually be erased. This can be prevented by activating the duplicate.

## Outline parameters

When entering outlines, there are different ways of entering Bézier curves. A Bézier curve is usually defined by four points: start-point, two pivots, end-point. The type of entry determines how these points are entered. Use the dialog box to set the type. There are three possibilities:

| **Entry type** | |
|---|---|
| Angle points | The Bézier curves are entered as a succession of angle points. With this type of entry, for each Bézier curve enter the two pivots and the end-point directly in that order (you don't have to enter the start-point, since it is the same point as the end-point of the previously entered contour). For each curve, in other words, you enter three points. |
| Tangents, equal | To enter Bézier curves, tangents are used. The direction of the tangents specifies the gradient at the start- or end-point of the Bézier curve, the length of the tangents specifies the deviation of the curve. In this setting you only enter one end-point for each tangent, the other one will be mirrored at the start- or end-point so that both parts of the tangent will be equally long. This has the effect that the passage between the two neighbouring Bézier curves will be very smooth.<br><br>For the first Bézier curve, you first specify the tangent at the start-point by entering the end-point of the tangent with the mouse. Then, enter for each curve the end-point and the tangent at the end-point. This means that for each curve you must enter two points. |
| Tangents, any | As described above, tangents are used to enter the Bézier curves. The two parts of the tangent may however be of different lengths. This is an antvantage when very different curves need to be entered, but still with smooth transitions.<br><br>For each Bézier curve, you first specify the tangent at the start-point by entering the tangent's end-point with the mouse. Then, enter the end-point and the tangent at the end-point. This means that for each curve you must enter three points. |

Changes made to the entry type only take effect when you next choose the input outlines command, not at once. This means that you cannot change the entry type during an entry!

## Freehand line

This command enables you to draw a line as desired. To do that, move the mouse while keeping the left button pressed. A line will be drawn. No definition point can be caught when a mouse button is pressed!

By releasing the mouse button, seperate lines can be entered directly in the same way as by polyline. In this case, points can also be caught.

Pressing the right mouse button cancels the entry. A dialog box appears, in which you can decide how to carry on:

| | |
|---|---|
| Yes | Accepts the freehand line in the current form and ends the entry. |
| No | Cancels the entry of the freehand line. Already entered lines will be lost. |
| Return | Returns to the entry of the freehand line. More points can be entered. |

A freehand line may consist of up to 2000 individual lines.

The freehand line is, depending on the freehand line parameters, more or less exact. The closer the freehand line corresponds to the movements of the mouse, the more memory will be needed.

## Freehand line parameters

This command enables you to determine how closely the freehand line follows the movements of the mouse.

A dialog box appears, in which you should enter a threshold value. The threshold specifies the minimum length of each single line of the polyline. If the threshold is large and the mouse is moved in very small steps, not all movements of the mouse will turn into a new line. If however the mouse is moved very fast, the line may become considerably longer than the threshold value.

## Line editor

| | |
|---|---|
| Text | Here you can enter a text of up to 255 characters. |
| Height | This value determines the height of the font inserted. |
| Angle | This value determines the rotation angle of the text line. |
| Italic | If this check box is active, the text will be inserted in the italic font type. |
| Narrow | If this check box is active, the text will be inserted in the narrow font type. |
| Current font | Inserts the text in the current font, the same one as used for dimensioning. You set the current font by using the dimension font command. |
| Library | Uses the current library to insert the text. The current library should be intended as a font library. |

## Text editor

First, enter two points to determine the area where the text is later to be inserted. Then a dialog box appears:

| | |
|---|---|
| Text box | When the dialog appears, the contents of the clipboard will be copied to the text box. The text can then be edited. |
| Height | Determines the height of the font to be inserted. |
| Line distance | Detemines the distance between lines. |
| Tabulator width | Determines the distance between two tabulators. |
| Italic | If this check box is active, the text will be inserted in the italic font type. |
| Narrow | If this check box is active, the text will be inserted in the narrow font type. |
| Left | Inserts the text left aligned, as determined by the previously chosen rectangle. |
| Centered | Inserts the text centered, as determined by the previously chosen rectangle. |
| Right | Inserts the text right aligned, as determined by the previously chosen rectangle. |
| Current font | Inserts the text in the current font, the same one as used for dimensioning. You set the current font by using the dimension font command. |
| Library | Inserts the text by using the current library. The current library should be intended as a font library. |

The text editor offers all standard procedure like deleting, inserting, copying etc. It can use the clipboard, i.e. texts can be imported from other applications or can be exported to them.

For the text editor is working within a dialog box, the keys TAB and RETURN are not available for direkt input. A tabulator can be inserted by pressing CTRL+I, a new line can be started by pressing CTRL+J or CTRL+M.

The area entered at the beginning is only used to execute the formatting of the text (left, centered, right). The text will **not** be cut at the borders of the area, nor will he be fit into the area.

## Font

TOMMYSOFTWARE® CAD/DRAW gets the information on characters from the current font. There are four different settings:

| | |
|---|---|
| No font | No font will be used. If a font already exists in the memory, it will be removed. Dimension numbers will not be displayed as a text, only as crossed rectangles.<br><br>When this setting gets active, a possibly existing font in memory will be erased. |
| DIN-draft | A simple font will be used, that is modelled after the DIN 6776 font, but composed only of lines. This makes it fast to draw. This font should be used while the drawing is in a draft stage and the font quality is not important.<br><br>When this setting gets active, the corresponding font will be loaded into memory. If this fails, the setting "No font" becomes active. |
| DIN-LQ | A font will be used, that largely corresponds to the DIN 6776 font. It is composed of outlines and needs a lot more time to be drawn than the DIN-draft. This font should be used for printing on devices that can easily present filled areas (e.g. the PostScript laser printer).<br><br>When this setting gets active, the corresponding font will be loaded into memory. If this fails, the setting "No font" becomes active. |
| Custom font | In this setting, any library can be loaded as a font. To load a font library, click in the box where the current file name is displayed. A file selector box then appears, that enables you to select a file name for the library.<br><br>When this setting gets active, the desired font will be loaded into memory. If this fails, the setting "No font" becomes active. |

The text and number specifications used for dimensioning are not, as is the case with e.g. the line editor, inserted into the drawing as objects. Instead, they will be redrawn with each redraw as specified by the saved values. This means, that if you load a new dimension font, all text and number specifications used for dimensioning will be displayed in the new font.

This enables you to work with a simple font (DIN-draft) or no font at all when the drawing is still in a draft stage, thus making a fast redraw possible. When the drawing has been finished, you load a font of a better quality in order to acquire the best printing results.

Note: Plotters usually can not draw filled areas, or only with a great amount of difficulties. Therefore you should not use filled fonts like DIN-LQ, that consist not only of lines, for plotting!

In addition to the settings here available the appearance of dimension numbers will also be affected by the font color and font size settings in the dimension parameters.

## Font layer

Characters are usually placed to the current layer. With this command you can choose one layer onto which all characters are to be placed. In this way, all characters can easily be hidden or frozen.

## Help menu

Register
About TOMMYSOFTWARE® CAD/DRAW

## Register

Use this command to generate a registration form for TOMMYSOFTWARE® CAD/DRAW. This form you have to send in to your local TommySoftware Partner to get registered. After choosing this command Windows Write will be started with the registration form loaded. This form already contains the address of your local TommySoftware Partner. You only have to print the form. And afterwards fill in your name, address, and credit card information. If you want to buy several licenses simply print the form several times.

And please, don't forget to sign it!

## About TOMMYSOFTWARE® CAD/DRAW

The appearing dialog box displays some information about the program, e.g. the version number.

Further, it displays customer information (name and serial number) and the country,

And finally, the available amount of free Windows system memory is displayed.

A scroll bar appears at the right or bottom border of a window, whose content is not entirely visible. Each scroll bar contains two scroll arrows and a scroll box, allowing you to scroll within the window or the listbox.

A dialog box in used to set parameters, enter text or numbers or confirm resp. cancel the execution of a command. It often appears immediately after choosing a command from a menu (the corresponding menu item then ends with '...') or during the execution of a command, if additional information is needed.

The clipboard is a temporary storage location used to transfer data between drawings and between applications.

The cursor can have different shapes, depending on its position and the current application status. During drawing, it appears as a crosshair, during command choice as an arrow and during application activity (i.e. during loading, saving, intensive calculation etc.) as an hourglass. During the screen refresh, the cursor may disappear short-time.

The buffer is an application-internal storage location used to hold cut or copied WINCAD drawings. The contents of the buffer can only be accessed from WINCAD itself.

An icon is a graphical representation of a command. The icon describes the type of the command graphically as far as possible.

To click means to press the left mouse button, while the cursor is above a specific area (Button, window etc).

A security check always appears, if changes in the current drawing or the library might get lost due to the execution of a command.

A file selector box enables you to select a name (including the path) of a file on diskette or on hard disk. All files in the current path, having the selected extension (e.g. "*.MPG"), are displayed in a listbox. Clicking on "*.*" displays all files in the current path.

Clicking on "Info" shows information (length and creation time) on the currently selected file. Clicking on "Standard" switches to the standard path, clicking on "Current" switches back to the last path used. If no files are displayed, you should edit the corresponding path by means of the command edit search paths.

A double click on a file name selects this file and exits the dialog box. A single click on a file name displays this file´s name in the edit box. This file name can now be edited. The extension does not have to be entered, for it will be appended automatically. Clicking on "Load" resp. "Save" selects the file, whose file name is displayed in the edit box.

A definition point is an user-entered point defining an object. A line, e.g., is defined by two definition points (start-point and end-point). Definition points are displayed as small squares, if desired. They can be caught (see point entry and objects).

Standard objects are simple objects, consisting exclusively of lines and circles. In detail these are: line, polyline, triangle, quadrangle, rectangle, trapezium, parallelogram, polyeder, circle, circle arc, circle sector, circle ring, circle ring sector and hatching.

The identification of a reference object is either done by entering its definition points, if the object does not exist yet, or by choosing an existing object (see construction).
As a "circle" you can identify circles, circle arcs and circle sectors. As a "circle arc" you can identify circle arcs and circle sectors.

The variant choice allows a well-aimed, keyboard-driven choice of one or multiple results of a construction or trimming.