Exit

Topics

Quick Start
Data Exchange
Status Window
Panel
Drawing Window
Selecting Commands
Key Assignments
Dialog Boxes
Program Configuration
View/Zoom
Properties
Layers
Reference Systems
Points
Selection
Identification
Variants
Geometry
Marks
Dimensions
Hatching
Generate Surface
Fonts
Symbols and Blocks

This chapter should help you to get started straight after installing the program. Because of this, only essential initialization procedures are described here. It is highly recommended that you read the "Working with.." chapter of the handbook.

After you have started TommySoftware® CAD/DRAW Professional, its program window appears. This is made up of up to four drawing windows (consisting of rulers, drawing area and crosshair), the status window (above the drawing window), the Panel (to the left of the drawing window) and the menu bar.

Your first decision should be whether you want to work with a white or black drawing area. The default setting is for a white drawing area. If you prefer a black drawing area, choose **System>Settings>Load**. Choose the file BLACK.T2I from the filing dialog and click on OK. When you have done this, the drawing area appears in black. The other window elements change automatically to suitable colors. You can also change the color of the drawing area according to your own preferences by choosing **System>Settings>Colors**. See also "Working With TommySoftware® CAD/DRAW Professional" in the manual.

Next, you should choose **System>Settings>Screen...** to set the program window according to your requirements and personal preferences. Screen Size and the number of drawing windows are particularly important. If you cannot read all of the legends in the status area, try reducing the Font Size in pixels.

Finally, you should choose **System>Settings>Memory** to configure system memory. This is where you specify the amount of memory which is made available for TommySoftware® CAD/Draw Professional. Of course this will depend on how much RAM your PC has and how much of it can be used by Windows. The values in the following table assume that all of the PC's RAM is available to Windows.

All values in KB (kilobytes):

RAM	Initial Size	System	Undo/Redo
3072	512	128	3
4096	1536	256	3
8192	4096	512	5
12288	8096	512	5

Save the settings using **System>Settings>Save as...** under the name TSCADPRO.T2I in your TSCADPRO directory. This will save your settings which will be used automatically the next time you start the program.

Now you can start working with TommySoftware® CAD/DRAW Professional.

Our addresses:

TOMMYSOFTWARE®

Headquarter
Selchower Strasse 32
D-12049 Berlin
Germany

Support Center
Asternstrasse 23
D-30167 Hannover
Germany

Phone (0 30) 621 59 31
Fax (0 30) 621 40 64
CompuServe 100142,3665

Hotline (05 11) 71 67 45
Fax (05 11) 71 33 90
CompuServe 100273,1143

CAD/DRAW Professional Help - Version 2.00e - Copyright 1990-1994 TommySoftware®

Various Converters are supplied with TommySoftware® CAD/DRAW Professional, which make possible the import of drawings which are not in TommySoftware® CAD/DRAW Professional's TVG 2.0 Format, or make possible the export of TommySoftware® CAD/DRAW Professional drawings in other formats. Using these Converters, you can convert individual drawings or whole sub directories of drawings.

In order to load drawings made with earlier versions of the program into TommySoftware® CAD/DRAW Professional , they must first be converted using the program TVG_T2G.EXE (Drawing files saved with earlier versions of TommySoftware® CAD/DRAW have the ending TVG; TommySoftware® CAD/DRAW Professional usually have the ending T2G).

If you have TommySoftware® MEGAPAINT® for Windows you can convert this package's MPG format files using the program MPG_T2G.EXE.

The converters DXF_T2G.EXE and T2G_DXF.EXE convert files from and to DXF format.

Please read the README.WRI file in the TSCADPRO directory for the latest information on converters.

It is of course possible to exchange information with other Windows programs via the clipboard. **Edit>Paste** pastes the contents of the Clipboard into the current drawing; **Edit>Copy** copies all the currently selected objects from the drawing to the Clipboard.
It is also possible to export a file in PostScript® or HPGL® format by loading the appropriate printer driver and printing to a file rather than the actual device.

The Status Window shows important information about the status of the program and displays certain other information. The Status Window has the following fields:

Command

This field shows the description of the currently selected command. This command remains active until a new command is chosen (see Selecting Commands). The same command can be carried out repeatedly without reselecting it. If the mouse pointer is in the Panel, this field shows a short description of the function of the Icon which it is over. If the mouse pointer is over a Pen Icon, then the name of the Pen is also displayed. If the Popup Menu is displayed, this field shows the name of the command whose Icon the mouse is over.

Point

This field shows the type of input which the program is waiting for. If, for example, "Enter Starting Point" is shown, the program is waiting for the point from which an object is to be drawn to be entered. If "Identify Circle" is shown, then the program is waiting for a circle to be identified.

Layer

This field shows the name of the current Layer. This is normally the Layer in which newly drawn objects will be placed. By clicking on the Layer name you can carry out Layer-related commands.

Left Mouse Button

Clicking in the field with the left mouse button carries out the command **System>Layer**, i.e. you can change layers.

Right Mouse Button

Clicking in the field with the right mouse button carries out the command **System>Define layers**; i.e. you can view and change the Layer definition.

This field can only be chosen when the *Long Status Window* check box in the **System>Settings>Screen** dialog has been enabled.

x=, y=

These fields show the coordinates of the point above which the cross-hair is located. If the mouse pointer is outside the drawing window, nothing is displayed here.

dx=, dy=

These fields show, after the input of a point, the X and Y difference to the crosshair's previous position. If this is not possible, or the mouse pointer is outside the drawing window, nothing is displayed here.

l=

These fields show, after the input of a point, the distance between the current crosshair position and the starting point. If this is not possible, or the mouse pointer is outside the drawing window, nothing is displayed here.

[ ]

The current measurement unit is displayed inside the square brackets.

Left Mouse Button

Clicking with the Left Mouse Button in this field brings up a dialog where you can change the measurement unit.

ß=

This field shows the angle of the current crosshair position relative to the starting point. If this is not possible, or the mouse pointer is outside the drawing window, nothing is displayed here.

[ ]

The current angle measurement unit is displayed in square brackets after the field *ß*=.

Left Mouse Button

Clicking with the Left Mouse Button in this field brings up a dialog where you can change the angle measurement unit.

System

This field displays the name of the Reference System in use by the current drawing window. By clicking on the name of the Reference System, you can carry out commands related to the Reference System.

Left Mouse Button

Clicking with the Left Mouse Button in this field brings up a dialog where you can change the Reference System for the current window.

Right Mouse Button

Clicking with the Right Mouse Button in this field brings up a dialog where you can view and change the Reference System definition.

This field can only be chosen when the *Long Status Window* check box in the **System>Settings>Screen** dialog has been enabled.

The Panel displays various Icons. These Icons serve as a representation and a means of specifying settings and carrying out actual commands. An Icon can be either "inactive", "active" or "unusable". The Icons change in appearance according to their condition. If an Icon is "inactive", it appears to stand out. If it is active, it changes in color and appears to be pressed in.
If it is "unusable" it appears to stand out like an "inactive" Icon, but the text and graphics are shown in a different color.
The status of an Icon can usually be changed by clicking on it with the left mouse button. This also changes the setting which the Icon represents.
In many cases, clicking on an Icon with the right mouse button brings up a dialog which can be used to alter the settings associated with the Icon. If the mouse pointer is moved over an Icon, the *Command* field in the Status Window displays the command associated with the Icon.

The following listing describes all the Icons from left to right and top to bottom.

Catch mode "Global"

By activating this Icon, the catch function is turned on; the last active catch mode is also activated. Alternatively, press F6 (see Key Assignments).
The "Catch" Icon will also be activated, if the catch function is off and one of the catch mode Icons is activated. If this happens, the only catch mode to be activated will be the one specifically chosen.
If no catch mode is active, the catch function will be turned off and the Icon will be shown as "unusable".The catch function can be turned back on by clicking on one of the catch mode Icons.
If the catch function is turned off, the last active catch modes are remembered and when the catch function is reactivated, either by clicking on the catch Icon or by pressing F6, the last active catch modes are restored. This allows the catch function to be temporarily turned off and back on without having to activate each individual catch mode.
If the catch function is active, four extra lines are displayed on the crosshair. The distance between these lines shows the actual catching radius.
The catching radius is measured in screen pixels. The catching radius determines the maximum distance which a point can have from the cursor and still be captured. Clicking on one of the catch Icons with the right mouse button brings up a dialog box where you can edit the catching radius.
In the same dialog box you can also set or alter the tolerance; this is needed for Identification (see Identification and **View>Catch radius...**).
More than one catch mode can be active at the same time. However, you should be careful only to activate sensible combinations. In general, it is best to only activate the catch modes which you actually need to use at any one time.
Instead of clicking with the right button on one of the Icons in the Panel, you can alter the catch mode by choosing one of the options from the **View>Catch modes** sub-menu. Active catch modes are marked with a tick. Usually, it is best to change the catch modes by clicking on the relevant Icon in the Panel.
Changing the catch mode via the menus does however allow you to allocate the activation/deactivation of an individual catch mode to a shortcut key for even quicker catch mode switching (See also Selecting Commands).

Catch mode "Center"

If this catch mode is active, the centers of lines, circles and arcs can be captured. The object to be captured must be identified.
The point which is captured is the midpoint of lines, or the center (the point from which the radius is measured) of a circle/arc.

Catch mode "Intersection"

If this catch mode is active, the intersections of objects can be captured. This catch mode will be applied to all intersections of Lines, Circles and Arcs, but not to Ellipses, Bézier Curves, Splines or Zigzag lines.

Catch mode "Quadrant"

If this catch mode is active, quadrant points of Circles and Arcs can be captured. Quadrant points are the intersections of a circle with a cross radiating from its center; they are at 0°, 90°, 180° and 270° relative to the center of the circle.

Catch mode "Edge"

If this catch mode is active, the edges of objects can be captured. The object whose edge is to be captured must be selected.
A perpendicular is drawn from the ID-Point to the edge of the object. The tangent point is then captured. Ellipses, Bézier curves, Splines and zig-zag lines are ignored.

Catch mode "corner/end-point"

If this catch mode is active, corner and end points can be captured.
Amongst other things, endpoints of lines, arcs and curves as well as corner points within curves and areas will be captured.

Catch mode "Marking"

If this catch mode is active, markings can be captured.

Catch mode "Pivot"

If this catch mode is active, pivots of Bézier curves which are within curves and areas can be captured.

Catch mode "Geometry"

If this catch mode is active, intersections of objects within the Geometry can be captured.

Catch mode "Other Point"

If this catch mode is active, all points of types which cannot be captured in the other catch modes can be captured.
These include the centers of circles and arcs, which have to be clicked on directly, as well as the centers of ellipses and portions of ellipses.

Arc direction "Positive"

If this direction is active, angles are shown positively, that is, counting anticlockwise.
This mode can be switched by pressing F2.

Arc direction "Negative"

If this direction is active, angles are shown negatively, that is, counting clockwise.
This mode can be switched by pressing F2.

Duplicate on/off

If the Duplicate function is on, a copy of the selected object and not the original is manipulated when you change an object with "changing" commands from the **Transform** and **Geometry** menus. The Duplicate function can also be turned on or off by pressing F7.

Geometry on/off

If the Geometry indicator is active, the objects of the Geometry will be displayed.

Clicking on this Icon with the right mouse button brings up a dialog where you can change the Geometry settings (see **Geometry>Geometry**).

Selection mode "Primary"

This turns the Primary selection mode on. All subsequently selected objects will be selected in this mode.

Selection mode "Secondary"

This turns the Secondary selection mode on. All subsequently selected objects will be selected in this mode.

Display grid on/off

If this is turned on, the Display grid will, if it is practical, be shown.
By clicking on this Icon with the right mouse button, the Display grid settings can be altered.

Position grid on/off

If the Position grid is active it will, if sensible, be used.
By clicking on this Icon with the right mouse button, the Position grid settings can be altered.
Both the Display and Position grids are determined by the current Coordinate System.

Pen Switch

Up to 40 Pens can be defined. Depending on the screen resolution, a different number of Pens can be displayed as Icons in the Panel. Clicking on the Pen switch Icon cycles through all the available Pen Icons.

Pens 0 to 40

Clicking on one of these Icons with the left mouse button selects it. This is not possible when the Icon is displayed as "unusable", because the relevant Pen has not been defined. Clicking on a Pen Icon with the right mouse button brings up a dialog where you can define the Pen's characteristics.
In the upper corners of each Pen Icon, there are two colored triangles. The left triangle shows the line color used by the printer/plotter; the color of the right hand triangle shows the colors used on screen (see **System>Define pens**).

Up to four Drawing Windows can be active at the same time. The proportions of Drawing Windows are always the same. All Drawing Windows show the same drawing.

Each Drawing Window can be allocated to a different Reference System and Zoom Area. The Reference System of the Drawing Window which the cursor is over is displayed in the Status Window. This also changes the displayed measurement and angle units, and the display of the Position and Display grids in the Panel, because they are determined by the Reference System currently in use. (See Reference System).

In order to allocate a new Reference System to a drawing window, it must first be made active by clicking on the ruler which is then highlighted in color.

Next, the command **System>Reference system** is used to choose a new Reference System which is then immediately applied to the active drawing window.

If the status of the Position of Display grid is changed by clicking on the relevant Icon in the panel, this alteration will apply to the active window. The same thing applies if the measurement or angle units are altered by clicking on the relevant area in the Status Window or if you set a new origin with the command **System>Set origin**.

If a new zoom area is defined with the command **View>Zoom section**, this section is only set in the active Drawing Window.

If more than one Drawing Window is displayed, this procedure can be used to display a whole page overview in one of the small windows where zoom areas for the other windows can be chosen.
To do this, first make the small window active by clicking on its ruler. Next, choose **View>Zoom page** to transfer these zoom characteristics to the whole page. Finally make the window which will be the usual working window active.
A section of a drawing can be chosen using the **View>Zoom section** command, in a drawing overview window. The chosen area will appear in the main drawing window, but the whole drawing will still be shown in the drawing overview window.

If you alter the Zoom area with the help of the scroll bars or the arrow keys, or by pressing +, -, × or ÷ (see Key Assignments), the alteration applies to the Drawing Window over which the cursor is positioned. If the cursor is not in a drawing window, the keyboard command cannot be carried out.

For all other commands, it makes no difference whether they are carried out in an active or inactive window. If you wish, you can start drawing a line in one window and finish drawing it in another.

The rulers are always displayed according to the current measurement unit and zoom area. The dashed line represents the page border. The position of the scroll buttons within the scroll bars shows the position of the zoom area relative to the sheet.
The small page symbol in the lower left corner shows the rotation or of the current Reference Systems (see Reference System).

There are several ways of selecting commands: via menus, via the Popup Menu, via the keyboard or - if available - a graphics tablet.
In addition, clicking on certain areas in the Panel or the Status Window accesses dialog boxes or allows settings to be changed.

Popup Menu

The Popup Menu is a window which consists of Icons and fields for changing layers. An Icon is a graphic symbol which represents a particular command. The Popup menu is activated either by clicking in the drawing window with the right mouse button or by pressing F9.

The Popup Menu can only be activated by clicking with the right mouse button if the program is *not* waiting for input. If the program is waiting for input, the *first* click of the right mouse button cancels the function, and the *second* click with the right mouse button activates the Popup Menu.

If the cursor is moved over one of the Layer fields in the Popup Menu, the Icons belonging to this Layer are shown and the frame around the Layer field will move to the new field.
If the cursor is moved over an Icon, the Icon will be displayed in reversed colors. At the same time, the command which the Icon represents will be displayed in the *Command* field of the Status Window.

Clicking the left mouse button while the cursor is over an Icon carries out the command associated with that Icon and leaves the Popup Menu.
Clicking the right mouse button or pressing ESC while the Popup Menu is displayed leaves the Popup Menu without carrying out a command.

You can customize the Popup Menu. Choose the command **Edit>Change popup menu** and then choose the command which you want to place in the Popup Menu. In the Popup Menu which appears after this, click the cursor at the position where you want the command's Icon to be placed. The "new" Icon replaces the "old" Icon, and the chosen command can now be accessed via the Popup Menu.

The Popup Menu configuration, together with the keyboard configuration, can either be saved or reloaded. This makes it possible to save special Popup Menus and/or keyboard layouts for special applications. See **Edit>Save assignments as**.

Note: When TommySoftware® CAD/DRAW Professional is started, the configurations from the file TSCADPRO.T2B are loaded; this file is stored in the same directory as the program file TSCADPRO.EXE. To make new configurations the defaults, they should be saved in the file TSCADPRO.T2B.

Assigning commands to keys

You can assign commands to the keys A..Z and 1..0 so that a command can be carried out with a keystroke. Each key can be used for up to three commands (KEY, SHIFT+KEY, CTRL+KEY).
In order to assign a command to a key, choose **Edit>Change keyboard assignment,** choose the command which you wish to assign to that key. and then indicate the key you wish to use in the dialog box which appears. By using the check boxes, you can assign the command to the key in its normal state or to the SHIFT+Key or CTRL+Key combinations. If the *Info-Mode* check box is enabled, clicking on a letter or number in the dialog displays its current assignment in an information box.

The command **Edit>Display keyboard assignments** brings up a dialog where you can see the complete

list of assignments and delete assignments. Some keys are pre-assigned. You can use the **Edit>Display keyboard assignments** command to view these assignments, which you can of course add to or alter.

Keyboard and Popup Menu configurations can be saved or reloaded. This makes it possible to save custom Popup Menus and/or keyboard layouts for special applications. See **Edit>Save assignments as**.

Note: When TommySoftware® CAD/DRAW Professional is started, the settings from the file TSCADPRO.T2B are loaded; this file is stored in the same directory as the program file TSCADPRO.EXE. To make new settings the defaults, they should be saved in the file TSCADPRO.T2B.

Some commands are permanently allocated to the function keys and alphanumeric keys, which means that pressing the relevant key carries out a certain command. In addition, you can allocate commands to your own preferred keys. See Selecting Commands.

Alphanumeric keys

ALT+BACKSPACE

Undoes last operation (see **Edit>Undo**).
The number of operations which can be undone depends on the number of undo levels set with **Systems>Settings>Memory**.

CTRL+BACKSPACE

Undoes last Undo.(see **Edit>Redo**).
The Redo function undoes the last Undo, i.e. operations which have been Undone are carried out after all.
The number of operations which can be undone depends on the number of undo levels set with **Systems>Settings>Memory**.

CTRL+INS

Copies all selected objects to the Clipboard (see **Edit>Copy**).

SHIFT+INS

Pastes all objects from the Clipboard into the drawing (see **Edit>Paste**).

DEL

Deletes all currently selected objects (see **Edit>Delete objects>Selection**).

SPACEBAR

Redraws the active drawing window.

ESC

Cancels the last point entry: the operation can then be repeated.
The type of point to be entered or the type of object to be selected next is displayed in the Status Window's *Point* field.

SHIFT+ESC

Restarts the current command. This is the same as choosing the command from the menu again.
This is most useful in the case of commands which are dependent on parameters entered in a dialog box immediately after choosing the command.
For example after choosing **Drawing>Polyeder>Standard**.the number of sides which the Polyeder should have must be entered.
All new Polyeders will be drawn with this number of sides, until a new number of sides is specified.
This can be done, if you are still drawing polyeders, by pressing CTRL+ESC, which is quicker than choosing the command from the menu.

Function Keys:

F1

Starts the Help function (see **Help>Information**).

SHIFT+F1

Loads an existing drawing (see **File>Load drawing**).

F2

Changes the Arc direction (see Panel).

SHIFT+F2

Saves the current drawing under the current name (see **File>Save**).

F3

Turns Display grid on/off (see Panel).

SHIFT+F3

Saves the current drawing under a new name (see **File>Save As...**).

F4

Turns Position grid on/off (see Panel).

SHIFT+F4

Prints the current drawing on the currently selected output device

F5

Redraws all currently visible Drawing Windows.

SHIFT+F5

Select a new Zoom area in the active window by dragging out a rectangle (see **View>Zoom section**).

F6

Turns Catch Function on/off (see Panel).

SHIFT+F6

Alters the Zoom level in the current window so that all existing objects are visible (see **View>Zoom>Overview**).

F7

Turns Duplicate Function on/off (see Panel).

SHIFT+F7

Alters the Zoom level in the current window so that the whole page is visible (see **View>Zoom>Page**).

F8

Allows direct entry of the coordinates of points, direct entry of radii, angles and lengths (see Points).

SHIFT+F8

Alters the Zoom level in the current window so that the drawing is viewed at actual size (see **View>Zoom>Original size**).

F9

Calls the Popup Menu (see Selecting Commands).

SHIFT+F9

View and alter Line patterns (see **System>Define line patterns**).

F10

Activates Menus.

SHIFT+F10

View and alter Pen definitions (see **System>Define pens**).

F11

Changes the Reference Systems for the active window (see **System>Reference system**).

SHIFT+F11

View and edit the Coordinate System definition (see **System>Define reference system**).

F12

Choose another Layer (see **System>Layers**).

SHIFT+F12

View and edit Layer definitions (see **System>Define layers**).

Numeric keypad keys

+
: Zooms in on the current cursor position. The cursor is moved to the center of the window and the zoom factor doubles.

-
: Zooms out on the current cursor position. The cursor is moved to the center of the window and the zoom factor halves

÷
: Undo Zoom
Undoes up to the last 20 changes of zoom level

ARROW KEYS
: Moves the zoom area in the relevant direction.

All of these commands apply to the current drawing window. The Undo Zoom feature also applies only to the current drawing window - undoing a zoom operation in one window does not affect the zoom levels in the others.

If you leave a dialog box by clicking on *Cancel* or pressing ESC the changes made in the dialog will not be carried out.
Otherwise, clicking on a button will either carry out a special function within the dialog or close the dialog box carrying out the changes specified. When the dialog box is next called, it will show the currently active settings.

Clicking on a color field brings up a dialog box where you can change the color.
Colors can be chosen in three ways. The simplest way is to click on one of the sixteen standard colors (in some cases, this is the only available method).
The other methods are to specify a color as an RGB or CMY value or to specify a user-defined color. You do this by choosing an entry from a list box. The chosen color, and its RGB/CMY values will be displayed.

A color becomes active when you leave the dialog by clicking on *OK*.

Non-standard (user defined colours) can be saved from the Select color dialog. By clicking on the *Apply* button the defined color will be incorporated in the list.

This section describes how you can configure the program to your own wishes and requirements.

If you have not already done so, please read the Quick Start section.

The keyboard and Popup Menu configurations are stored in a configuration file with the ending T2B. This makes it possible to save custom Popup Menus and/or keyboard layouts. See **Edit>Save assignments as...**.

Note: When TommySoftware® CAD/DRAW Professional is started, it uses the configuration from the file TSCADPRO.T2B; this file is stored in the same directory as the program file TSCADPRO.EXE. To make new settings the defaults, they should be saved in the file TSCADPRO.T2B.

All other settings which directly affect the program are stored in a file with the ending T2I. (see **System>Settings>Load**.

Note: When TommySoftware® CAD/DRAW Professional is started, it uses the settings from the file TSCADPRO.T2I; this file is stored in the same directory as the program file TSCADPRO.EXE. To make new settings the defaults, they should be saved in the file TSCADPRO.T2I.

The following list provides a key to the settings. Refer to it when making changes.
A * shows that a particular setting is very important for the configuration of TommySoftware® CAD/DRAW Professional and you should use this setting in your own configuration (*.T2I) file
The descriptions of the individual commands, explain the meaning of each setting in detail.
The other settings have less effect on the fundamental workings of the program but more effect on the program's interface. These settings will probably be changed every time the program is used..

The settings controlling the following commands are stored in a settings file (*.T2I).

File>Load drawing
* **File>Autosave**
File>Print drawing
* **View>Display**
View>Key functions>Arc direction
View>Key functions>Catch
(see Key Assignments and Panel)
View>Key functions>Duplicate
(see Key Assignments and Panel)
View>Catch Radius (see also Panel)
View>Catch Mode (see also Panel)
* **System>Settings>Screen**
* **System>Settings>Colors**
* **System>Settings>Memory**
* **System>Settings>Time format**
* **System>Settings>Paths**
* **System>Settings>Autoload**
Draw>Freehand drawing Parameters
Draw>Zigzag line Parameters
* **Geometry>Geometry** (see also Panel)
Geometry>Marking>Layer

Geometry>Division and **Geometry>Multiple copy**
Measure>Dimension (The settings for each sub-menu are stored separately.)
Measure>Dimension>Line properties
Measure>Dimension>Text properties
Measure>Hatching types>Define
Special>Generate surface>Properties
Special>Generate surface>Parameters
* **Special>Font table**
Special>Font table>Properties
Special>Single line text and **Special>Multiple line text** (all settings except the current text)
Library>Insert Symbol/Block
Library>Symbol/Block>Properties
Library>Read Symbol/Block
Library>Generate parts list
Selection>Selection mode (see also Panel)
User defined Colors

The settings which apply to a specific drawing are saved along with the drawing itself in the drawing's T2G file. See **File>Load drawing...** and **File>Merge drawing>Standard elements...** and **File>Merge drawing>Other elements...**.
Loading a drawing from a drawing file automatically loads the settings associated with that drawing.This is very useful because settings and definitions are very much dependent on the individual drawing and vary widely from drawing to drawing.

Note: When a new drawing is started or TommySoftware® CAD/DRAW Professional is started, the settings from the file TSCADPRO.T2G are used; this file is stored in the same directory as the program file TSCADPRO.EXE. To make new settings the defaults, they should be saved in the file TSCADPRO.T2G.

The following list of commands provides a key to their settings and can be used to alter them. The supplied file TSCADPRO.T2G already contains some standard settings, in particular Pen and Line definitions conforming to the DIN standards. You should always use this file as a sample when you wish to define your own settings. Please read the descriptions of the individual commands, which explain the meaning of each setting in detail.
The following settings are saved in the drawing file:

View>Zoom areas
System>Define Layers
System>Define reference systems
System>Page format
System>Define pens
System>Define line patterns

You should exercise extreme caution when altering TSCADPRO.* files. If you do accidentally erase a TSCADPRO.* file, you can substitute a corresponding file from the SYSTEM subdirectory of your TSCADPRO directory by copying it and renaming it. The files STANDARD.T2G, STANDARD.T2B, BLACK.T2I or WHITE.T2I contain the standard settings.

You should never amend these files directly, but always work with a copy.

A drawing can be shown on the screen at various sizes. The visible part of the drawing, the Zoom area, is shown relative to the original size (that is, the size at which the drawing will be printed. See also **View>Zoom**).
This relative size is expressed as the Zoom or Zoom Factor. If the factor is 1.0, the drawing is shown at original size, in which case the drawing's elements appear on screen at the same size as that at which they will be printed.
You can view part of a drawing at original size by choosing **View>Zoom>Original size**.
There are several similar commands in the **View menu.**

By choosing **View>Zoom>Section** you can select a rectangular section of your drawing, which then becomes the active Zoom area. See also Drawing Window.

You can save frequently used Zoom areas with the command **View>Zoom area>Define**.

The command **View>Undo zoom** allows you to undo the last 20 zoom actions in each window. See also Key Assignments and Drawing Window**.**

In addition, it is possible to influence the ways in which objects are displayed; see **View>Display**.

In order to depict an object, apart from geometric information (e.g. the coordinates of a line's end points), certain other data is required which define, for example, the color, line width and line pattern with which the object should be drawn. These pieces of information are called characteristics.

An object has the following characteristics:
- Number of the assigned layer;
- Line style, made up of line pattern, line width and line color;
- Fill style, made up of fill mode and fill color;
- Number of the assigned pen.

If a Pen other than Pen 0 is assigned to an object, usually the object will be drawn with the *Line Style* and *Fill Style* assigned to that Pen. Because of this direct relationship with Pens, it is possible, with a single alteration of the Pen definition, to alter the appearance of all objects drawn with that Pen.
If Pen 0 has been used, the characteristics remain with the object. In this case, altering the Pen definition will make no difference to the appearance of the object.

Each object is assigned, via a Layer number, to a particular Layer. These layers have an organizational purpose. They make it possible to organize individual parts of a drawing and to go quickly to any drawing element. Outlines, Symmetry and Help lines, hatching, measurements and descriptive text should each be assigned to their own Layer.
To speed up screen redraws or make the drawing clearer, the layer containing (for example) hatching can be removed.

When an object is drawn, it is normally assigned to the current Layer and Pen. It makes sense to allocate specific types of object to specific layers and possibly also to specific Pens, to achieve a consistent structure to the drawing. With the help of certain commands, distinct characteristics can be specified for these purposes which can be allocated to specific object types. These are:

Geometry>Marking>Layer...
Measure>Dimension>Line properties
Measure>Dimension>Text properties
Measure>Hatch Objects>Properties
Special>Generate Surface>Properties
Special>Font Table>Properties
Library>Insert Symbol/Block>Properties

Theoretically, problems can arise if an object has been allocated to a particular Pen, but must retain some of its properties unaltered as they are decisive for that object. In these cases, it is possible to lock some of an object's properties against alteration. For example, it can be decided that a particular object will always be drawn with 0.25 mm line width, regardless of which Pen is selected.
This locking can be carried out with the command **Shape>Edit properties>Object**. The object which is to have properties fixed must be selected, and the properties must be changed in the dialog box which appears.
The line width can be altered to the desired value. By doing this, the *Transmit* check box is automatically enabled, which fixes the line width. After closing the dialog box by clicking on OK, the line width of the object will be independent of the chosen Pen.

Carrying out a similar process with symbols is slightly more complicated. A symbol may consist of several objects. Each of these objects might have totally different properties. If a symbol is instanced (see Symbols and Blocks), i.e. if a "copy" of the symbol is on screen, each of the individual objects which

make up the symbol will be displayed with its properties.
Problem: to make a symbol, regardless of the properties of its individual elements, display completely in red. The properties of the individual objects cannot be changed, because there is only one instance of the symbol. It must be possible to make all the objects in this instance of the symbol appear in red. This is done in a similar way to the fixing of a line's width described above.
To do this, choose the command **Shape>Edit properties>Object** and identify the relevant instance of the symbol. A dialog box appears, showing the instance's current properties.
Next, the line color is changed to red. By doing this, the check box *Transmit* will be enabled, which fixes the line color to the objects in the symbol. Close the dialog box by clicking on *OK*. All objects of this instance will be shown with red lines. The objects of the original symbol will *not* be altered.
It may be that some of the objects of which the symbol consists have a fixed line color. In this case, the line color will not be displayed in red, but in the fixed color..

TommySoftware® CAD/DRAW Professional drawings can have up to 300 separate Layers. Layers are used to structure a drawing by putting elements with the same theme on the same Layer. By doing this, they can be manipulated in the same way later.

Layers should be used to structure the drawing right from the start. To do this layers can be set in the program, and certain important elements will be automatically assigned to these layers. This maintains the structure automatically.
The following drawing elements can be assigned to their own layers:

Dimensions (see **Measure>Dimension>Properties**)
Hatching (see **Measure>Hatch objects>Properties**)
Marks (see **Geometry>Marking>Layer**)
Generated Surfaces (see **Special>Generate surface>Properties**)
Text (see **Special>Font table>Properties**)
Symbols / Blocks (see **Library>Insert Symbol/Block>Properties**)

All other drawing elements are always placed into the current Layer.

By organizing your work into defined layers, you can make your work a lot easier: individual layers can be frozen or removed. It is much easier to maintain a well-ordered drawing.

It is possible to organize a drawing into layers after it has been completed: however this needs so much effort that it is not really practical.

See also **System>Layers** and **System>Define layers**.

Reference Systems play a central role in TommySoftware® CAD/DRAW Professional. Reference Systems (or "Systems") contain all the important settings for screen drawing and working within a window.

A Reference System has these parts:

View

The *Display* setting influences the display of the drawing in the window. As well as the unmodified *Cartesian* display, various modified display types can be chosen, which make the production of isometric and dimetric drawings easier.
For example, if. *Isometric, left view* is chosen, the on-screen representation of the drawing will be altered so that parts of the drawing which belong to the left view are shown unmodified.
The left face of an isometric cube is, in the drawing, a parallelogram with two sides at 90 ,and two sides at 150°. Each of the sides is the same length. However, because the top and bottom sides are tilted, the parallelogram is smaller horizontally than vertically.
The display of the drawing in the window is modified in such a way that the parallelogram is shown as a quadrilateral. The display is sheared vertically by 30°, in order to balance out the vertical angle of 150° and scaled by the factor 1/cos(30°) horizontally, in order to preserve the aspect ratio.
Because all the drawing's elements are modified equally, the other views of the cube appear to be more distorted than they actually are.
The distortion of the drawing is visible because of the position of the page frame and the page symbol. (See **System>Settings>Screen**).
See also "Isometric Drawing".

Rotation

The *Rotation* setting influences the position of the coordinate axes relative to the page. Rotation of 30° means that the coordinate axes are each rotated by 30°.
Because the coordinate axes on the screen always run horizontally and vertically, the page is rotated by 30°, the effect is the same.
The rotation of the drawing is always visible from the position of the page frame and the page symbol (see **System>Settings>Screen**).

Scale

The scale influences the input and output of coordinates. Internally, the drawing is always saved at the same size as that at which, under normal circumstances, it will be printed. This is the same as a scale of 1:1.
If the drawing is to be produced at a different size, the program has to be told, so that the coordinates and lengths can be recalculated.
A scale of 1:100 means that the internal coordinates must be multiplied by 100 before output, and divided by 100 after input.
Because the scale is only indirectly connected with the drawing, altering the scale does not alter the drawing itself.
If it is to be output at a different size, either it must be directly scaled during printing or scaled with the command **Shape>Scale objects**.

Origin

This determines the position of the coordinate origin. This is the point around which the Reference System is rotated (see *Rotation*), and distortion is carried out relative to this point (see *Display*)
The origin can be either set to a defined position on the edge of the sheet, or placed relative to the center of the sheet.

Units

These settings determine in which units the input and output of coordinates, lengths, angles and other measurements are carried out.

The *Length Unit* is used for the input and output of coordinates, lengths, radii, clearances, dimensions, and all measurements which depend on the drawing scale.

The *Line Unit* is used for the input and output of line widths, text sizes, and other measurements which do not depend on the drawing scale.

Finally, the *Angle Unit* is used for the input and output of all angle types.

Numbers and Precision

These settings determine how numbers are displayed.

It is particularly important to set the number display correctly. For drawings using feet and inches, it is usually necessary to set this to *Mixed fraction*.

Display Grid and Position Grid

These grids help with the production of drawings which have to be placed on a predetermined grid.

The display grid can be set independently of the position grid.

See also **System>Reference systems**, **System>Define reference systems**.

Points can be entered in two different ways: by clicking in the Drawing Window, or by using the keyboard.

The type of point to be entered next is shown in the Status Window's *Point* field.

When using the mouse, the actual position of the cursor in the Drawing Window determines where the point will be placed. The positioning of the cursor can be influenced if a position Grid is active; if this is the case the cursor can only be moved to certain positions.

Point entry with the keyboard is context sensitive. A dialog box appears after pressing F8 (see Key Assignments) which allows you to enter absolute, relative (to the previous point) or polar co-ordinates depending on the active command (line, circle etc.) and the point which is to be entered. According to the type of command, it is possible to choose a point on the sheet or on the edge of the currently selected object.

You can remove the last-entered point by pressing ESC.

All entries use the current measurement units, as displayed in the Status Window.

An object, e.g. a line or circle (see the **Draw** menu) consists of a certain number of points from which it is calculated. These points can be displayed, appearing as small right angles, by choosing **View>Display**.

Selection of an individual object: see Identification.

By Selection, we mean choosing one or more objects. There are various methods of selecting objects; see also the **Selection** menu. Selected objects can be manipulated in various ways, e.g. rotated or scaled.

The selected area is indicated by a frame around the selected objects or by indicating the object's edge with a dashed line. You can alter this setting with **View>Display**, and change the color of the frame/line by using **System>Settings>Color**.

In many cases, objects with the same properties (e.g. line width, object type or Layer allocation) have to be selected. This is achieved with the command **Selection>Conditional selection** which brings up a dialog box where you can specify the characteristics of the objects to be selected.

There are two different selection modes: primary and secondary selection. These are displayed in different colors. The active selection mode is shown in the panel. Normally, primary selection is active; secondary selection is mainly used for hatching.

Objects remain selected until explicitly deselected, using Deselection. Deselection has the same options as selection. Selection is not preserved when a drawing is saved, i.e. when a drawing is loaded, nothing is selected.

For many commands, objects must be identified. Objects are identified by clicking on their outlines.

The type of object to be identified next is shown in the Status Window's *Point* field.

The point which is clicked on during identification is referred to as the ID-Point. The current catch radius (see Panel) determines how far the ID-Point can be from the object's outline and still be valid. If two objects overlap, or are very close together, it is difficult to clearly identify one of the objects. The Catching Radius Tolerance shows how accurate one must be to make a valid identification. If the tolerance is, for example, 3 pixels, an identification is only regarded as clear if there is no other object within a radius of 3 pixels of the chosen object. If this is not the case, you can choose from all objects which fall within the area.(See Variants).

There are commands where the position of the ID-point relative to the object is very important. One example of this is the command **Geometry>Trim line>Intersection with perpendicular**. When using this command, the line which is to be trimmed must be identified, and then the two perpendiculars to which the line is to be trimmed must be identified. The position of the ID-Point determines the part of the line which is to be trimmed the portion of line nearest to the ID-point will be removed.

In general (as far as the ID-Point is concerned), wherever you click, something changes.

Variants are used, for example, during Identification (see Identification). If an identification is not clear, a dialog box appears at the bottom of the screen to choose which object should be identified. If you reposition this dialog box, the next time it is used it will appear in the new position.

Using the dialog box, you can choose one of the possibles to be identified. To do this, one of the objects is shown with a frame and a dashed line round its outline. You can change the color of the frame or the outline with **Systems>Settings>Colors**.

Clicking on *OK* identifies the currently framed object and closes the dialog. The *Forwards* and *Backwards* buttons let you cycle forwards or backwards through the possibles. Clicking on *Cancel* closes the dialog without identifying an object.

During the construction of objects, variants are also sometimes required. In this case, the procedure is similar to that described here, except that the choice is from objects which can be drawn.

The Geometry is an aid to construction. The only objects which can be included in the geometry are lines and circles. The lines are shown with infinite elasticity, in principle they are straight.

See also **Geometry>Geometry line, Geometry>Geometry parallel**, **Geometry>Geometry tangent** and **Geometry>Geometry circle**.

The Geometry can be turned on and off, see **Geometry>Geometry** and "Panel". The Geometry is saved and loaded with the drawing. Additionally, it is possible to load the Geometry from a file into the current drawing using the command **File>Merge drawing>Other Elements**. The Geometry is never printed out and cannot be copied to the Clipboard.

Objects in the Geometry cannot be selected. In order to delete objects from the Geometry, you must choose **Geometry>Erase geometry>Object** or **Geometry>Erase geometry>Complete**.

The objects in the Geometry can be identified. All intersections of objects in the geometry can be captured.

Marks are small symbols which are used to mark positions, e.g. during construction. They can be defined during construction and are shown as small squares with a cross through them. They are only visible on screen and can not be printed or copied to the clipboard.

Marks can be made in various ways. They can be placed directly on to the screen with the cursor, or they can be constructed with commands from the **Geometry>Division** sub-menu.

In addition, marks are used in conjunction with the command **Geometry>Multiple copy>Markings**. Using this command, it is possible to copy selected objects to several marked places in the drawing at the same time.

TommySoftware® CAD/DRAW Professional uses its own fonts both on screen and for printed output. Only in exceptional cases are the Windows system fonts used.

Various vector fonts are included for this purpose, amongst others the fonts DINDRAFT and DINLQ. These are vector fonts which are particularly suitable for annotating technical drawings and for depicting measurements. DINDRAFT consists only of lines and is therefore highly suitable for output on plotters. The other fonts generally consist of filled areas and are therefore less suitable for plotters.

By using the commands **Special>Single line text** and **Special>Multiple line text** it is possible to place text on the screen. In the dialog boxes which appear when you choose one of these commands you can choose which of the available fonts is to be used from a combo box. To edit text after it has been placed in a drawing, first select it and then choose **Special>Edit text.**

The typeface used to show measurements is set from a combo box in the dialog box which appears when you choose **Measure>Dimension>Parameters**.

Fonts are administered with the command **Special>Font table**.

In order to display a font, it must first be loaded by choosing the command **Special>Load font** or by setting it to load automatically on start up with the command **System>Settings>Autoload**.

Inserting symbols into TommySoftware® CAD/DRAW Professional uses a system of external references. This means that whenever a symbol is inserted into a drawing, a reference marker is inserted which tells the program to insert a certain symbol from a certain library at that point rather than a copy of that symbol. which means that the symbol remains identifiable.

Naturally, the symbols imported into a drawing can, for example, be rotated, scaled or manipulated in similar ways. A special matrix saved with the imported drawing makes this possible. The original symbol in the library is not altered.

This saves significant amounts of memory because, regardless of the actual size of the symbol, only a reference to the symbol is ever saved. It is very easy to place a symbol from an updated library into a drawing. The updated library is simply copied on top of the old one. When the drawing is redrawn or printed, the updated symbol will be used automatically, because the references will have remained the same.

Apart from this, symbols can have attributes applied to them (e.g. article number, description, price); see also **Library>Edit Attributes**. These attributes can be used to generate a parts list; see **Library>Generate parts list**.

The command **Library>Insert Symbol/Block** can be used to insert symbols from a library into a drawing. Using **Library>Read Symbol/Block** you can convert objects from a drawing to symbols and store them in a library.

Functionally, blocks and symbols are identical. The difference is that blocks are not stored in external libraries, but each block's definition is stored in the drawing. This means that after a block has been read in using the command **Library>Read Symbol/Block** the block definition will be saved with the drawing. If a block is inserted, a reference to this definition, the so-called Instance, will be placed into the drawing.

It is possible to convert blocks to symbols or vice versa using the commands **Library>Transform>Symbols to blocks** or **Library>Transform>Blocks to symbols**.

Symbol and block definitions can be nested, i.e. symbol or block definitions can contain references to other definitions. The maximum number or references which can be nested is ten.

If you want to work on part of a symbol, you can convert symbols or blocks to their constituent objects with the commands **Library>Resolve symbol/block>Instance** (single) and **Library>Resolve symbol/block>Instance** (multiple objects). If an instance refers to a definition which again contains instances, these instances remain after the first deletion. In order to remove these instances, use the command **Library>Resolve symbol/block**. This process has to be repeated until all the instances have been removed.

In order to be able to insert a symbol from a library into a drawing, the library has to be loaded. To do this, either choose **Library>Load library** or by making the library load automatically when the program starts. See **System>Settings>Autoload**.

These commands can be use to attach dimensions to drawing elements. They can be generated in many different ways, using a multitude of settings which affect the appearance and behavior of the dimension. These commands lead to a dialog box in which the appearance of the dimension figures, the dimension line and possibly the extension lines can be set. Some commands (e.g. **Measure>Dimension>Line, point**), which produce dimension lines give rise to the following dialog box:

Starting Point

These settings influence the appearance of the terminator at the starting point of the dimension line.
The radio buttons with depictions of various terminators allow you to choose the type of terminator.

Orientation

The orientation of the terminator can be chosen from this drop-down list.
If the orientation is *Rotated*, the terminator will be rotated by 180° and the dimension line will be drawn elongated., i.e. it will extend beyond the starting point.
If the orientation is *Automatic* the terminator will be rotated by 180° if the length of the dimension line is shorter than 30 × its width.

Extension line

If this check box is enabled, an extension line will be drawn from the starting point of the dimension line..

Ending Point

These settings influence the appearance of the terminator at the ending point of the dimension line.
The radio buttons with depictions of various terminators allow you to choose the type of terminator.

Orientation

The orientation of the terminator can be chosen from this drop-down list.
If the orientation is *Rotated*, the terminator will be rotated by 180° and the dimension line will be drawn elongated., i.e. it will extend beyond the starting point.
If the orientation is *Automatic* the terminator will be rotated by 180° if the length of the dimension line is shorter than 30 × its width.

Extension line

If this check box is enabled, an extension line will be drawn from the starting point of the dimension line..

Dimension

These settings influence the appearance and position of the dimension, that is the dimension itself as well as the associated text and tolerances.

Font

Choose the font in which the dimension should be shown from this list.

Dimension

Enter the height of the dimension text in this field. Values between 3.5 and 7.0 mm are recommended.

Tolerance

Enter the height of the text used for tolerances in this field. Values between 2.5 and 5.0 mm are recommended. The size should be one step (factor 1.41) smaller than the main dimension text.

Accuracy

Specifies the number of decimal places when numbers are displayed as numbers and decimals.
Specifies the maximum power to be used when numbers are displayed as exponentials.
In both cases values from 0 to 9 are allowed.

Coordinate system

Choose the coordinate system which will apply to the dimension line from this list. This system determines the scale, units, text font etc.
The accuracy will however be taken from the previously entered field.

Adapt dimension

If this check box is enabled, the dimension will be automatically updated whenever the drawing is altered.

Center dimension

If this check box is enabled, the dimension will be placed in the center of the dimension line. Additionally it can be positioned perpendicularly.

Dimension close

If this check box is enabled, the dimension will be automatically positioned close to the dimension line. It can also be positioned parallel to the dimension line.

Rotate dimension

If this check box is enabled, the dimension can be rotated by specifying a point. Otherwise the dimension is always displayed parallel to the dimension line.

Edit dimensions and tolerances

If this check box is enabled, a dialog box appears after the dimension has been entered in which the dimension text (including the dimension text itself) can be edited.
If the Adapt dimension check box is enabled, then the dimension will be overwritten when the next operation is carried out!

Dimension line

These settings influence the appearance and positioning of the dimension line and the extension lines.

Full length, perpendicular

If this check box is enabled, the dimension line will be shown at full length, i.e. with both terminators.
Extension lines will be perpendicular to the dimension line.

Full length, moved

If this check box is enabled, the dimension line will be shown at full length, i.e. with both terminators. Extension lines can run at a slope, determined by entering a point at the end of the dimension line.

Partial length, perpendicular

If this check box is enabled, only part of the dimension line will be shown. Points must be entered to determine at which end of the dimension the terminator should be shown, and another point which determines the length of the dimension line.

Display dimension line

This check box must be enabled in order for dimension lines and extensions to be shown. Otherwise only the dimension line itself will be shown.

Enter dimens. direction

If this check box is enabled, a dimension direction different to the direction of the object whose dimensions are to be determined, can be entered. To do this, a point is entered which together with the starting point of the dimension, determines the dimension direction.

Distance by point

If this check box is enabled, the position of the dimension line is determined by entering a point through which it should run.

Fixed distance

If this check box is enabled, the dimension line will be positioned at the distance from the dimension which is shown in the field.
The point which has to be entered determines which side the dimension line should be shown on.

Multiple of distance

If this check box is enabled, the distance from the dimension will be determined by the value in

the field; a multiple is also allowed so that the position of the point which has to be entered determines how often the distance should be taken.

Some commands (e.g. **Measure>Dimension>Coordinates**) do not generate dimension lines: they bring up this dialog:

Font
Choose the font in which the dimension should be shown from this list.
Dimension
Enter the height of the dimension text in this field. Values between 3.5 and 7.0 mm are recommended.
Tolerance
Enter the height of the text used for tolerances in this field. Value between 2.5 and 5.0 mm are recommended. The size should be one step (factor 1.41) smaller than the main dimension text.
Accuracy
Specifies the number of decimal places when numbers are displayed as numbers and decimals.
Specifies the maximum power to be used when numbers are displayed as exponentials. In both cases values from 0 to 9 are allowed.
System
From this list, choose the Coordinate System which should apply to the measurement. This system determines the scale, unit, appearance of numbers, etc.
The tolerance will however be taken from the Tolerance field, described above.
Adapt dimension
If this check box is enabled, the dimension will be automatically updated whenever the drawing is altered.
Center dimension
If this check box is enabled, the dimension will be placed in the center of the dimension line. It can also be positioned perpendicular to the dimension line.
Dimension close
If this check box is enabled, the dimension will automatically be placed close to the dimension line. Additionally, it can be positioned parallel to the dimension line.
Rotate dimension
If this check box is enabled, the dimension can be rotated by specifying a point. Otherwise the dimension is always displayed parallel to the dimension line.
Edit dimensions and tolerances
If this check box is enabled, a dialog box appears after the dimension has been entered in which the dimension text (including the dimension text itself) can be edited.
If the Adapt dimension check box is enabled, then the dimension will be overwritten when the next operation is carried out.

The Parameters set here are saved with each measurement object and can be altered afterwards with the command **Measure>Dimension>Properties**.
Some special commands, for example **Measure>Dimension Line** and **Measure>Dimension** can be used for altering specific properties of measurement objects.
Because text size and spacing are not altered by operations like scaling or distortion, the dimensions appear with the correct text size and spacing after such operations.
If the setting *Adapt Dimension* has not been enabled, the dimension can be updated with the command **Measure>Dimension>Update**.

If *Edit Dimension and Tolerances* has been enabled, the dialog box described under **Measure>Edit dimension text** appears automatically, where the dimension text (preceding text, measurements, following text, tolerances) can be edited.

These, and other, settings can be saved in settings files. See **System>Settings>Save as**.

After closing this dialog by clicking on *OK*, the relevant points for the current measurement object must be entered. The settings chosen in this dialog remain active until another (Measure) command is chosen. If you want to change the settings for the current measurement command, you must choose the command again, to set new parameters. The easiest way of doing this is by pressing SHIFT+ESC (see Key Assignments).

The following two examples illustrate procedure when measuring an object.

Measure>Dimension>Length, points

Using this command it is possible to measure a length whose endpoints are entered directly. This makes it possible to measure random lengths which do not lie within one or more object.

1. Enter the starting point
2. Enter the ending point

After entering the length, several points determining the position of the dimension and the dimension line must also be entered. The number of points which must be entered is dependent on the chosen measurement parameters.

If *Display Dimension* and *Enter dimens. direction* are enabled:

3. Enter the first dimension line definition point. This determines, relative to the starting point, the dimension direction.

If *Display Dimension* is enabled:

4. Enter the second dimension line definition point. This determines the separation of the dimension line from the dimension. If *Full Length, moved* is enabled, this point also specifies the ending point of the dimension line.

If *Display Dimension* and *Partial Length, perpendicular* are enabled:

5. Enter the third dimension line definition point. This specifies the ending point of the dimension line.

If *Center dimension* or *Dimension Close* are not set:

6. Enter the first dimension definition point. This specifies the position of the dimension.

If *Rotate dimension* is enabled:

7. Enter the second dimension definition point. This specifies, relative to the dimension position, the direction in which the dimension should be rotated.

Measure>Dimension>Length, object

This command can be used to measure a length specified by identifying an object edge. This makes it easy to measure lengths within existing objects without having to enter the points individually.

The **Special>Generate surface** command allows you to generate a surface area from several overlapping or adjacent objects. Only geometric objects, e.g. lines, circles, ellipses or curves are used. Other objects such as text or measurement lines are ignored.

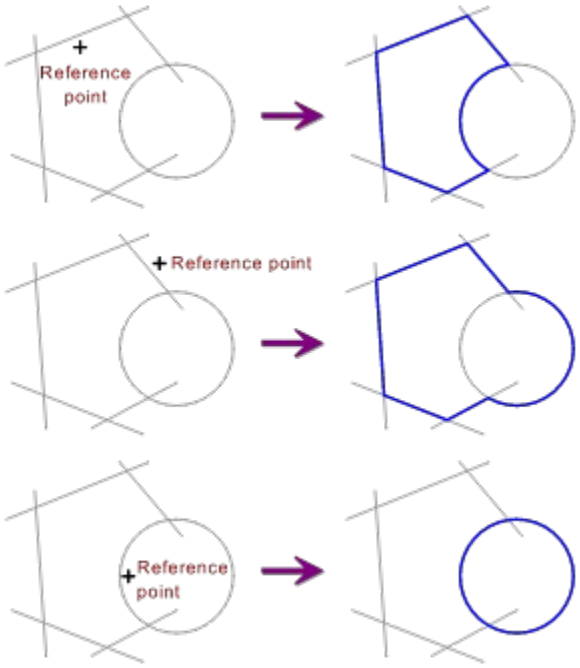
Because the outline of an object can only consist of lines and arcs, all objects are converted during processing. Objects which contain other elements e.g. ellipses, are converted into line sections. The precision of the conversion is set with the command **System>Settings>Screen.**

Generation is virtually automatic, but all objects which are relevant to the area being generated must be selected. The easiest way to do this is using the command **Selection>Select section>Overlapping**. You should select only the minimum number of objects necessary because the number of objects influences the time needed for calculation. After this a reference point, which defines the surface to be generated is entered

The position of the reference point determines which surface is generated. The generation routine's strategy can be described as follows:

Find the element from the selected group of objects which is nearest to the reference point;
Starting with this element, find a closed outline which encloses the reference point and is as short as possible;
If there is no such outline, find the largest possible closed outline around all the selected objects;
If this is not possible either, display an information message.

The position of the reference point has the following effects: If the reference point is within one or more closed surfaces, the smallest of them will be generated. If it is outside, the largest possible surface surrounding all the objects will be generated. The graphic below illustrates this.



In the case of a clearly differentiated surface (which is usually the case) you can simply click in the

middle of the surface.
You should never click on or near a line or arc whose end is not connected to another element, because in this case the generation process will not be able to find a closed outline connected with that object.

The tolerance set with the command **Special>Generate area Parameters** specifies how close two objects have to be to each other in order to affect each other. If the tolerance is set to a very low value, some tangents of lines and circles will no longer be recognised properly by the Generate Surface function.

After entering the reference point, the surface generation begins. If a surface is found, it is inserted and selected; the previously selected objects are deselected. This means that the newly generated surface can be worked with straight away; for example it can be hatched (see Hatching).

The surface will be generated using the properties set with the command **Special>Generate surface>Properties**.

Note: System memory is used during the surface generation process. With complex surfaces there may not be enough system memory. If the memory requirement cannot be reduced by deselecting unnecessary objects, available system memory must be increased using the command **System>Settings>Memory**.

These commands are used to shade objects. It is possible to define two groups of objects which can be joined in three ways.

The first group of objects consists of all objects selected in main selection mode. The second group of objects consists of all objects selected in secondary selection mode. The current selection mode can be changed from the Selection menu or by clicking on a button in the Panel.
Objects which do not enclose an area, for example lines, are ignored during hatching. If an area whose outline is made up from individual objects is to be hatched, then this area must first be transformed into a surface using the command **Special>Generate surface** (Generate Surface).
Surfaces with overlapping outlines are always treated correctly.

The hatching type which has been set using the command **Measure>Hatching type** will be used. In addition the properties set with **Measure>Hatching properties>Objects** will be respected.

During hatching, one or more hatching objects will be generated. These can be broken into individual lines with the command **Transform>Resolve objects to lines**.

Union

This command hatches the union of both groups of objects, i.e. all areas covered by at least one object are hatched. For this kind of hatching it is sufficient simply to select those objects (using the Main selection mode)which should be hatched, as no distinction has to be made between different objects.

Intersection

This command hatches the intersection of object groups, i.e. all areas covered by at least one object from the first group and also covered by at least one object from the second group will be hatched.
This command is used for example when the overlapping area of two circles is to be hatched. In this case each circle must belong to a different group.

Difference

This command hatches the difference between two object groups, i.e. all areas which are covered by only one of the object groups are hatched.
This command is mainly used when hatching an area which surrounds a smaller area which is not to be hatched. The smaller area must belong to the second group.