FelixCAD™ 3.1

User's Guide

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FELIX Computer Aided Technologies GmbH
Nestorstraße 36a - 10709 Berlin - Germany
TEL (030) 89 69 03 – 0
FAX (030) 89 69 03 – 12
E-mail 100567.652@compuserve.com

Visit our World Wide Web site at WWW.FCAD.COM

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FELIX Computer Aided Technologies GmbH
Nestorstraße 36a
10709 Berlin, Germany

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Introduction

FelixCAD™, and the *FCAD* ***Graphic Developer's Engine™*** (its companion version for the professional application or corporate developer), are advanced CAD systems designed specifically for 32-bit Microsoft Windows 95™ and NT™ desktops. Installation of FelixCAD on the Windows platforms is discussed near the end of this chapter.

Designed for the fast and effective solution of drawing jobs, *FelixCAD* offers all the power, precision, and ease-of-use that the modern CAD user or application developer require.

Range of capabilities

FelixCAD includes all of the functions necessary for constructing and detailing drawings, with many unique features aimed at saving you time and effort in your drawing tasks.

FelixCAD's **drawing commands and functions** make short work of drawing lines and polylines, circles and arcs, rectangles, regular polygons, solid entities, 3D surface entities, and other elements. *FelixCAD* allows you to identify and define **parts,** provides associative dimensioning, and helps you build libraries for use in repetitive drawing tasks.

Drawing entities may be altered as desired by using the **modify commands**. Size, position, shape, and other graphic qualities may be changed. Advanced editing functions such as **trimming, filleting, and chamfering are available to build sophisticated results.**

Attributes of any part may be defined or edited, insuring all pieces that make up a drawing retain intelligence for later recall or use. Once completed, parts may be stored as a complex object increasing their usefulness in repetitive situations.

For additional detailing, you may call on **dimensioning, cross-hatching,** and the insertion of **text** elements.

FelixCAD makes all of these functions available either by keyboard command entry or easy-to-use palettes (floating tool boxes) for true point-and-click drawing!

Layers help you organize your work. This technique also improves drawing management, and makes it easy to create effective, clear, and well-detailed drawings.

Instead of fumbling with a complex system of dividing the drawing into different modes in order to see the results in various views, angles and perspectives, *FelixCAD* allows you to open up to four drawings, with up to four viewports per drawing. All views can are displayed simultaneously on the screen for your benefit while you continue to work on the “active” view. You may have as many as sixteen viewports open on the screen during the drawing process.

At every step, you are supported by on-line **Help** as well as a very useful and detailed on-line **Tutor** that graphically illustrates the geometry behind the commands. This tutorial allows you to call up the illustrated description of a command, see how it works, and work an exercise if you wish. The tutorial includes a discussion of the user coordinate system, the layer concept, and the object snap and object selection functions.

FelixCAD has been designed as a truly open architecture system. This facilitates **industry compatibility,** and allows you to freely exchange information with other CAD applications, evaluation, and analysis programs.

FelixCAD allows the **import** and **export** of drawings in industry standard **DXF** format or **DWG** format, transferring both graphic and non-graphic information without destruction or loss of data. This ensures the file exchange with a variety of CAD programs and applications.

Flexible and Adaptable

The configuration and customization commands allow many other possibilities for the adaptation of the program. You may manage such things as the use of colors and scripts, input and output media, and “hot key” assignment of the function keys, just to name a few examples.

FelixCAD is very flexible with regard to configuration of the desktop. You can choose between a standard pulldown menu, or one of your own definition.

Menu, Palette and Dialog Box Customization

FelixCAD contains an integrated **Dialog and Menu Editor**. This editor may be used to adapt menus to specific tasks, or to create brand new menu structures. Most importantly for you, such changes do not require any programming knowledge.

Palettes are toolboxes which allow for direct point-and-click access to commands and functions. There are a number of included FelixCAD palettes designed to make drawing tasks easy. Should you require additional functionality, the Dialog and Menu Editor lets you create as many new palettes as you wish. These new palettes may be topic-related commands and functions, or application-related.

Palettes may be called upon through point-and-click techniques or by hot-keys. To insure optimum productivity, they can be freely placed anywhere on the program's desktop.

If you prefer to modify the standard FelixCAD desktop, you may simply choose to edit the included palettes.

Lisp Programming

The integrated *FelixCAD* Lisp Interpreter (FLISP) facilitates not only the adaptation and extension of FelixCAD, but opens wide the possibility of working with other programs, utilities, routines, and libraries. The function library of the Lisp interpreter includes functions to display dialog boxes and evaluate the user input. This includes those that are delivered with the program as standard, and those that you may choose to create with the Dialog and Menu Editor.

Developer's Toolkit Manual

The FelixCAD Developers Toolkit (FDT) Application Programming Interface (API) manual is included on-line for the convenience of advanced developers wishing to extend the program's capabilities using C or C++ programming languages.

FelixCAD includes an API which allows professional developers or advanced users to access nearly all FelixCAD functions through the use of Dynamic Link Libraries (DLL).

The API manual can be opened and/or printed using the included Adobe Acrobat reader. Search the MANUALS folder for any .PDF extension files starting with FDT.

Documentation

The *FelixCAD* package includes a printed User's Manual. In addition, on-line documentation in Adobe Acrobat format is included for all FelixCAD instructions for customization and programming. Of note are the on-line manuals detailing Customization and Lisp Programming and the FelixCAD Developer's Toolkit (FDT), an advanced C/C++ language programming interface available with FelixCAD.

Now let us present a short summary of the content of each of these manuals.

User's Guide

The User Manual contains a description of the general commands and functions which represent the basis for drawing. is included both in printed form and on-line form for your convenience.

To view the on-line user guide, use the included Adobe® Acrobat™ reader to open the corresponding file (userguid.pdf) contained in the MANUALS sub-directory. Instruction on Acrobat installation is contained near the end of this chapter.

The first chapter covers basics on working with the program and introduces the elements which are available.

The second chapter offers a detailed description of drawing aids. These are functions for precision drawing, the object snap functions, and other precision drawing aids. The use of different coordinate systems is explained in this chapter as well.

The third chapter explains the techniques used to view the drawings, with detailed explanations of the use of the viewports, how to achieve good results from using FelixCAD's capability to see many views at the same time, and the use of the multiple-drawing viewing capability. The World and User Coordinate Systems are also discussed.

The remaining chapters explain working with layers, the functions involving drawing, working on, and detailing (dimensioning, crosshatching) objects, the definition and insertion of parts and attributes, working with text objects, and all other topics that pertain to helping you do a fast and easy job of drawing and designing with *FelixCAD*.

Finally, importing and exporting functions, and outputting the drawings on a plotter or a printer are explained.

Programmer's Guide

The *Programmer's Guide* will appeal to users interested in further adjusting, individualizing, customizing and programming *FelixCAD* and its functions to specific application needs.

To view the on-line programmer's guide, use the included Adobe Acrobat reader to open the file PGUIDE.PDF, contained in the MANUALS sub-directory.

Chapters 1 through 3 contain a number of suggestions for possible arrangement of the user interface, working with the alias command, macros, and function key assignments, as well as a guide to the directory structure of the program.

In Chapter 4 and 5 there is information describing the meaning and utilization of template drawings and - understood in combination with this - the definition of linetypes and hatch patterns.

Chapters 6 through 9 describe in detail working with the Dialog and Menu Editor (DME), and describe the techniques used in setting up new (or editing the standard) menus, palettes and dialog boxes.

In Chapters 10 and 11, detailed information is provided about the integrated LISP interpreter allowing you to extend the program or comply it with special working requirements.

In Chapter 12 there is an overview of the global and local variables used in the program.

We look forward to your successes with *FelixCAD*. Please don't hesitate to call on us for assistance. Tell us your thoughts on the product and your experiences with its use, whether you are a user or a developer. Your input will be welcome, and we look forward to talking personally with you!

Program Installation and Setup

FelixCAD includes an automated installation routine, which is started by using the appropriate Windows “run” function to execute the program SETUP.EXE contained in the root directory on the included CD.

FelixCAD offers 3 types of installation:

a) Installation in “demo” mode, whereby all program functions and operations may be evaluated by the user. When running in demo mode, the only exclusions from a full running system are the ability to save a file. In addition, any printed or plotted output will contain hard coded text entries indicating creation from a FelixCAD demo system.

b) Full running installation, whereby you must have an authorization diskette available when prompted.

c) Update from demo mode to full running mode. As above, an authorization disk is required to convert FelixCAD from demo mode to full running mode.

Demonstration CD’s and authorization diskettes can be obtained from your Authorized FelixCAD product dealer. Contact FCAD for the dealer location nearest you.

Installation using the Windows 95 / NT 4.0 Desktop

1) Insert the FelixCAD CD into the CD drive of your computer.

2) Under the Start function, browse the CD to select the SETUP program contained in the root directory. The disk is an “autorun” CD, so it may start up automatically upon insertion into your CDROM drive.

3) Run the SETUP program, following the on-screen prompts to complete the installation.

Installation using the Windows NT 3.51 Desktop

1) Insert the FelixCAD CD into the CD drive of your computer.

2) In the Program Manager, select RUN under the File menu. Browse the CD to select the SETUP program contained in the root directory.

3) Run the SETUP program, following the on-screen prompts to complete the installation.

Installation using the Windows 3.11 Desktop

Prior to using FelixCAD under the Windows 3.11 or Windows for Workgroups desktop, you must have the Windows 32-bit sub-system installed. For your convenience, the FelixCAD CD includes this sub-system.

To install WIN32S from the FelixCAD CD:

1) In the Program Manager, select RUN under the File menu. Browse the CD to select the WIN32S SETUP.EXE program located in the \WIN32S\DISK1 directory on the CD.
2) Run the SETUP program, following the on-screen prompts to complete the installation.

Once WIN32S has been successfully installed, follow steps 1-3 as described under the Windows NT installation section.

Un-install

FelixCAD can be un-installed from your computer if desired. If using the Windows 95 desktop, the program can be removed using the “Add/Remove” function under the Control Panel.   To remove FelixCAD using the Windows 3.11/NT desktop, run the Uninstall program. The Uninstall program will remove the program groups and .ini files from the Windows directory.

Make sure not to delete any drawing (.FLX, .DWG, .DXF) files you may have saved in these directories after installation.

Adobe Acrobat Installation

The FelixCAD CD includes Adobe’s on-line document reader for Windows. To install Acrobat from the FelixCAD CD, run the ACROINST.EXE program situated in the \ACROBAT directory of the CD. Follow the on-screen prompts to compete the installation.

Technical Support

Please contact your local dealer or visit our homepage **http://www.fcad.com** to see who your local distributor is in your country or choose the Support page for hints and tips and bonus files. You can also use the FelixCAD homepage for technical requests or reports.

Chapter 1

Getting Started: The Basics of Drawing

This chapter imparts a basic understanding of the program and furnishes information regarding some of the most frequently used commands and tools.

It begins with a description of how to start the program. The chapter then deals with the ability to create new drawings and open existing drawing files. It also explains how to save work results, and how to terminate the work correctly.

You may choose to use the mouse or pointing device in picking from the menu bar, tool areas or palettes.

In addition, the program can also be operated by typing commands using the keyboard, or by using a digitizer with a tablet menu. For defining points, distances, and angles, keyboard entry of the corresponding values gives the advantage of greater precision. Ideally, a combination of mouse and keyboard command access optimizes your productivity.

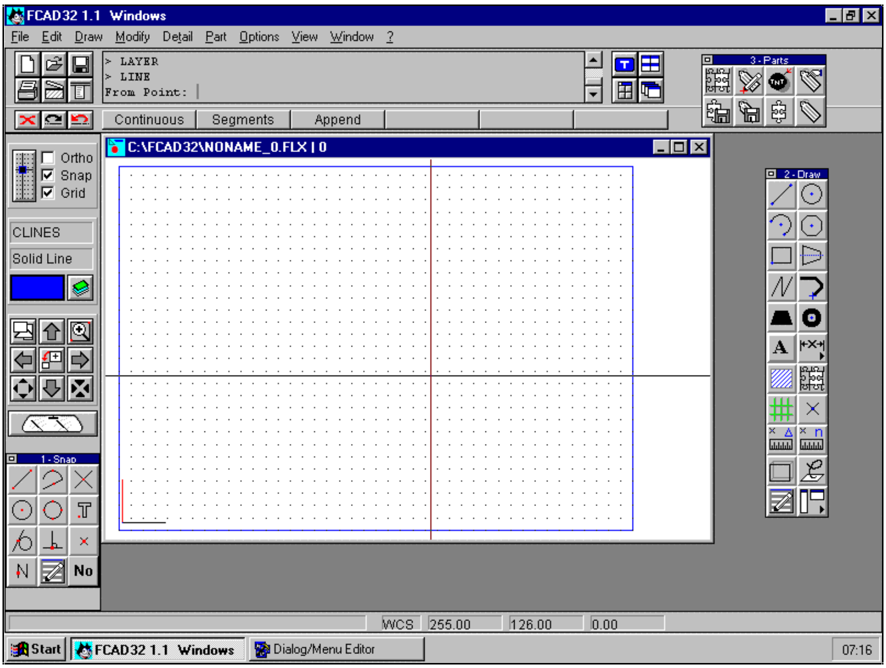
The chapter contains explanations of command selection, coordinate input and other statements, whether by keyboard, mouse or digitizer. We advise the novice user to read this part carefully and to practice the described procedures. The program provides alternatives for many inputs and selections during drawing construction. The variant that you choose will depend on your own personal style.

In this chapter there is a comprehensive description of the user interface and its elements and components. There are instructions to control the graphical interface and the individual arrangement of its elements.

General Remarks

FelixCAD runs on a Microsoft Windows desktop. All essential program steps function, therefore, according to Windows protocols. This is true for pull-down menus, tool boxes, dialog boxes, edit boxes, scroll bars, etc. The desktop provides a series of standard elements whose function and mode match the corresponding elements of the Windows desktop. For example, there are buttons for opening new or existing drawing files, to save or print drawings, as well as for changing window arrangements.

Working with the program requires at least a minimum of Microsoft Windows knowledge. The use of standard control elements will not be explained in detail in this manual. In case of doubt, please consult your Windows manual, Windows Help, or other literature on the subject of Windows.



Starting the Program

A detailed description of the installation is contained in the preceding chapter of this manual.

During the installation of the software the Windows Program Manager will add a special FelixCAD program group, and will place it in a special window. This window will contain a program symbol for the application, a program symbol for the dialog and menu editor and possibly, depending on the your application, other symbols pertaining to that application.

The program should be started by double-clicking the application symbol or by selecting the symbol and pressing ENTER.

Opening and Saving Drawings

In this chapter, the commands for opening a drawing, for saving the work results, and for finishing a work session will be described.

The following commands should be understood:

New...
Open...
Close
Save
Save As...
Save All
Delete Files...
Exit

These commands are located in the pull-down menu *File***.** There are also symbols in the function bar for the most used commands of this menu.

The commands for printing or plotting, which are also found in the *File* menu, will be explained in another chapter.

By default, the program saves drawings as files with the file extension **.flx**.

Each file contains a drawing and, if necessary, up to four views of that drawing. Each drawing as well as each view of a drawing will be displayed in a separate window. All view windows of a drawing carry in the title bar the same filename. They differ from each other in the continuous numbering (0 - 3), which is separated from the filename by a vertical line. This is discussed later in this chapter.

Further in this manual, the term *drawing* may be used as a synonym with the term *file*. When the manual mentions “opening or closing a drawing”, it refers to opening or closing of the corresponding file.

Also, the following paragraphs of this chapter explain the procedures used to save the work. Moreover, the differences between the commands SAVE, SAVEAS and SAVEALL are mentioned and explained.

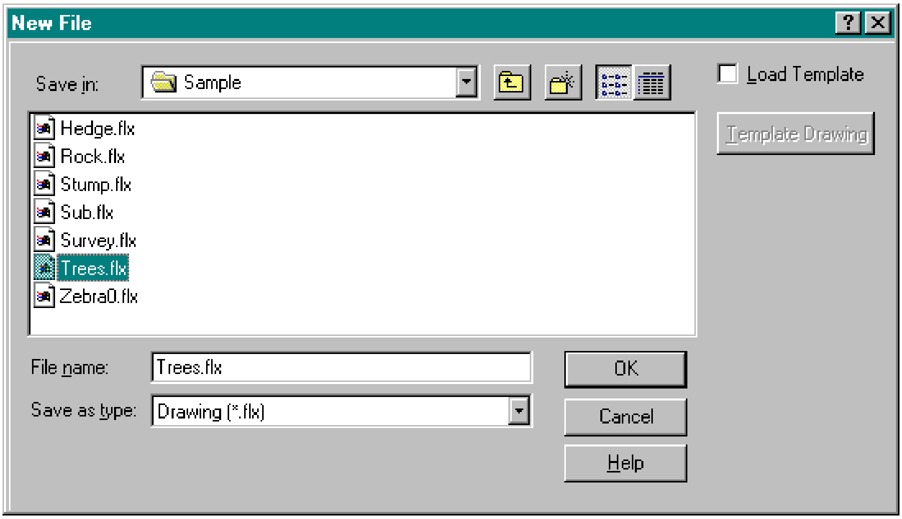
To avoid unforeseen loss of data we strongly recommend periodically saving drawings while working on them. For this purpose use the function **Auto-Save** which can be activated in the configuration dialog box (see *Programmer’s Guide*, Chapter 1).

Local variables will be saved along with the drawing and the enclosed drawing entities current at the time the automatic save occurs. Such settings as the current coordinate system, the dimensioning mode and other adjustments will remain at your disposal.

New Drawing

A new file has to be opened in order to create a new drawing. The command **New...** in the *File* menu is used for this purpose. Alternatively, the command to create a new file can called by clicking the *New* button on the function bar or by typing NEW at the command line.

The following dialog box will be displayed:



The program proposes to name the new drawing **noname_*n*.flx.** You may now specify a file name your choice. To determine the certain storage location of the file select a drive and directory in the dialog box.

If the file name and the storage location are correct, confirm the settings. An new drawing window to create a new drawing file will then be opened.

If a drawing file already exists with the same name, the program will warn you and require a confirmation to replace the existing file.

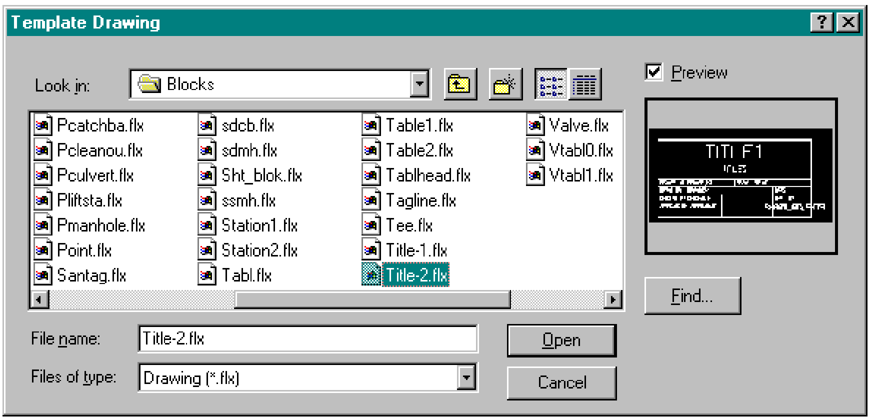
Using Template Drawings

A *template drawing* may be determined when starting a new drawing. A template drawing is a model which can be used as basis for a final drawing or for the establishment of a drawing environment.

A template drawing offers the advantage of applying settings of an already existing drawing to a series of new drawings. The setting of such parameters as the drawing boundaries, snap raster, dimensioning settings, pre-defined layers, views, etc. do not have to be repeated. Automatically, the parameter settings of so-called *local variables* are loaded and set with the template drawing. Variables that are loaded with a template drawing and variables that will have to be specified with each drawing are described in the *Programmer's Guide*.

To start a new drawing based on a specific template drawing, activate the check box **Load Template** in the dialog box *New File* and then double-click on the button **Template Drawing**.

The dialog box *Template Drawing* will be displayed. Select the drawing to be to used as a template drawing and confirm the choice.

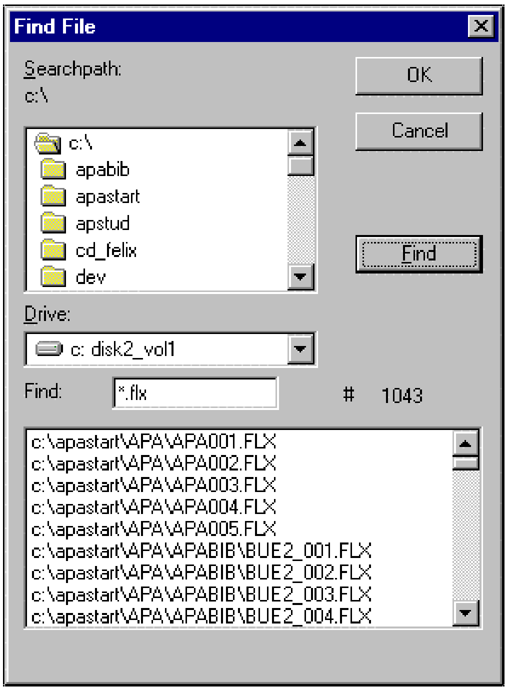


Find... (Find a File)

If you do not know the exact name or the storing location of a drawing or file, you can look for it by using the **Find** option. The dialog box *Find File* will be opened. Select the drives to be checked. Enter the file name of the drawing to be found. You may use the known wild cards * (for any group of characters) and *?* (for any single character).

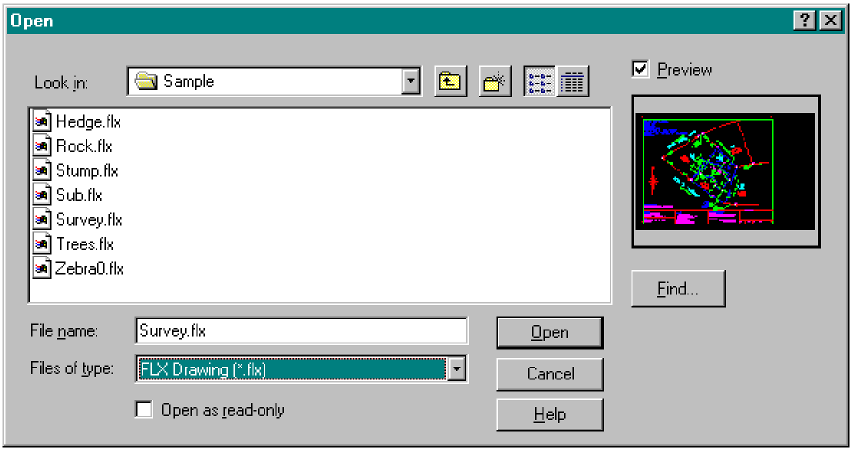
After the entries are confirmed the program will search for corresponding file names in the indicated drives.

All files matching the search criteria will be listed in a result list. The size, construction date and time as well as the location will also be indicated. You can now select the desired file from this list and confirm your choice.



Open an Existing Drawing

The command **Open...** in the menu *File*, is used to open an existing drawing file. This command can also be activated by typing the command OPEN or by clicking the standard symbol *Open* in the function bar. After the entering the command, you will see the *Open File* dialog, and you may type in or select a drive, directory, file name, and file type.



FelixCAD can directly open .FLX files, .DWG files saved in R2.5-R14 format, or .DXF files.

Read-Only Mode

You may activate the *Read-Only* option in the *Open File* dialog box. The Read-Only mode allows the file to be opened in order to be read, but it may not be modified under its original name. This is a protective mechanism against inadvertently modifying or changing the file. After opening in Read-Only mode, the file can be modified in any way, but the modifications will not be saved. If you should try to save a Read-Only file, the program will send an error message and ask, at the same time, if the file should be saved under another name. If the changes are to be saved, then you should answer *Yes* and type another file name in the *File Open* dialog box.

When opening a drawing, the above will appear in the bottom left corner of the program window to show the progress of the load. The blue bar moves from left to right as the drawing loads and disappears when completed.

Closing a Drawing

Closing a drawing indicates that the work on this drawing is finished. The program remains open.

To terminate working on the current drawing, select the command **Close** in the *File* menu or press the keys CTRL+F4. This command can also be keyboarded by typing CLOSE.

It is also possible to close the drawing window, as in all Windows programs, by double-clicking on the system menu area (left top corner for Win NT/3.11, right top for Win 95 / NT 4.0). This activates the command WCLOSE and the drawing window(s) will be closed.

If a drawing has been changed before closing there will be a prompt asking whether the file should be saved. This is to prevent unintended loss of work.

If you decide to save the drawing, the program will automatically switch to the standard routine to save the drawing under a new name. Drawings may be directly saved to .FLX format, DWG format (R12, R13 or R14), or DXF format.

Save the Current Drawing

The command **SAVE** causes the drawings in the active window to be saved.

At this time the file will be saved under its present name. The file is saved in the FLX drawing format and to the same storage drive and directory as was specified at the time of opening.

Automatically, a copy with the extension **.bak** is created to save the work done.

The command to save can also be invoked from the menu *File* or by clicking on the *Save* button in the toolbar.

Save All Current Drawings

The command SAVEALL from the menu *File* command will not only the save the drawing in the current document window but also all other open drawings.

Save As...

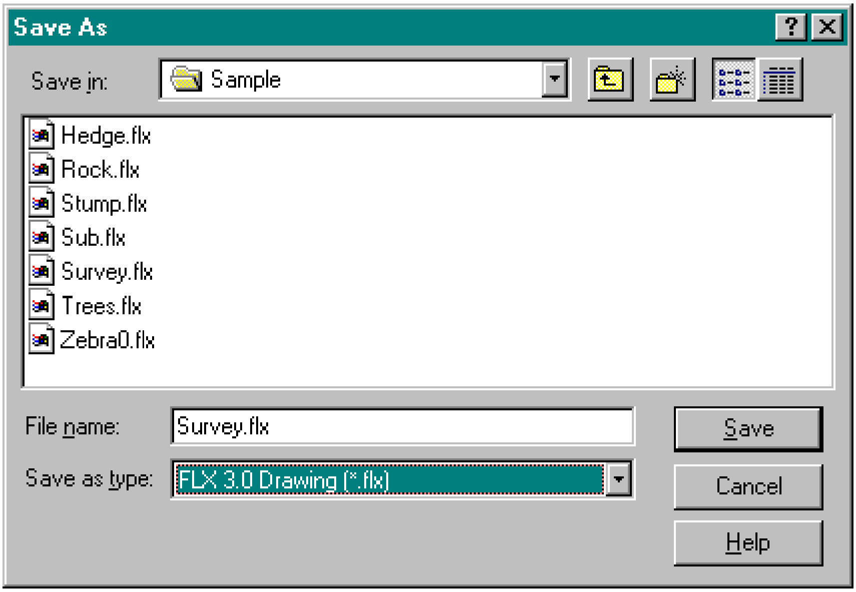
The command SAVEAS from the menu *File* also saves the current drawing, like the above mentioned command SAVE.

However, it allows you to save the drawing under a different name, or in a different area.

The command SAVEAS will require that you answer these questions, all contained in the dialog box:

- File name
- File format
- Drive
- Directory/Sub-directories

This command will then save the work done under the new filename, or in the new storage area. Drawings may be directly saved to FLX format, DWG format (R12, R13 or R14), or DXF format.



Deleting Files

The program contains a function for deleting files. It allows you to remove the so-called lock-files. The Delete function is called from the menu *File***,** or entered by typing the command DELFLK.

Lock-Files are security files which are created by the program when a drawing file is opened. They carry the same name as the original file and can be identified by the extension **.flk**. Lock-Files will be deleted automatically, if the drawing files are closed in the normal manner.

This procedure avoids two or more users working on one drawing at the same time when in a networking environment**.**

If the program is unexpectedly interrupted and the files have not been closed before the interruption, the same situation will occur. If you try to re-open a file after an interruption you will get the message *File locked*.

In order to re-gain access to the drawing, the corresponding Lock-Files must first be deleted.

In order to delete these files, first enter the command. Next, you must perform the standard steps for the selection of files (similar to the steps in opening files). The program will offer the extension **.flk** as the desired file format.

Delete the offending lock-file, and the drawing file can again be opened.

Exit: Terminating the Program

It is very important to end every working session correctly by using the EXIT command in any of these fashions:

- Select **Exit** in the pull-down menu *File* and confirm.
- Press the ALT+F4 keys.
- Double-clicking on the system menu area (see Windows manual)

If there are open drawings which have not been saved, the program will send a safety question. If you confirm, the dialog *Save File As* will be called.

After saving the drawings the working session with the program will be finished.

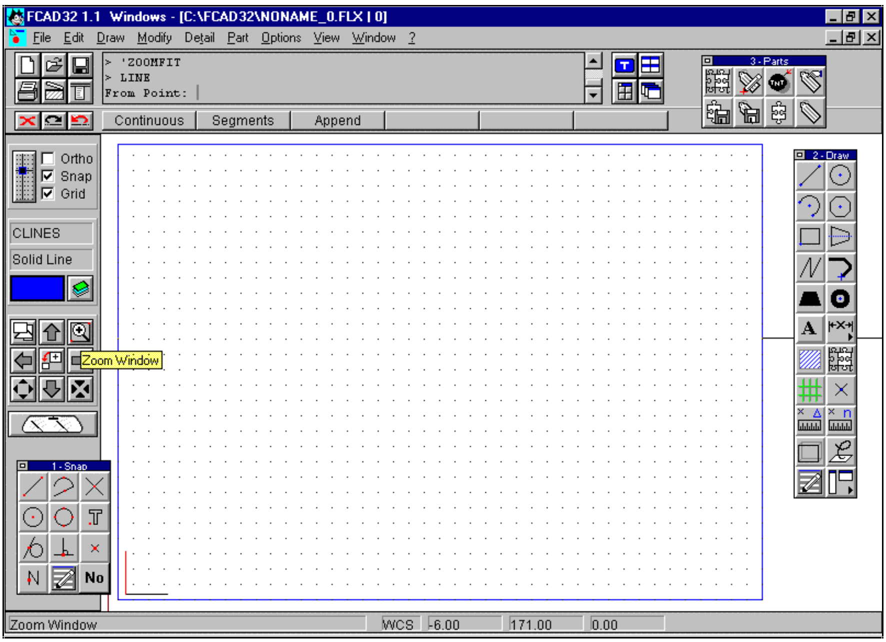
The Program’s User Interface

After starting the program you will see the *FelixCAD* desktop. It contains a series of standard elements. These are the menu bar, the function bar, the vertical symbol bar, and the status bar.

In addition, the desktop may also contain several palettes and a cursor menu when applicable.

You can find comprehensive references to the configuration and the individual customization of the desktop as well as the method of saving the settings in Chapter 2 of the ***Programmer’s Guide***.

The following picture shows the standard variant of the desktop.



Menu Bar and Pull-Down Menus

The separate pull-down menus are arranged in the menu bar. Each pull-down menu contains groups of thematically-related commands. The content of the Menu Bar varies depending on the loaded menu file. Different menu files may be loaded, adapting the menu bar to the specific necessities of the user.

Menu files can be identified by the extension **.mnu**. The program contains a standard menu file located in the sub-directory APPLIC.

The program offers you the opportunity to load different menu files. These files can be individualized menu files, which have been created and adapted by help of the Dialog and Menu Editor (explanation in the *Programmer's Guide),* or any of a number of standard menu files supplied with the program.

The menu file can be changed by using the **Desktop/Tool Manager** from the *File* menu or by typing the command MENU into the command line or the text window. For further information see the paragraph **Desktop and Tool Manager** in Chapter 11 of this manual.

To open the pull-down menus you should click on the desired menu name or press the ALT-key and the hotkey of the name. The menu will open and you may select a command which is available.

- Dimmed, or grayed-out commands are not available in the particular working situation and can therefore not be selected. The command Save, for example, can not be executed if no drawings have been created or opened.
- Three points (...) behind the command - for instance, Open... - show that no further requests will be displayed at the command line and all command parameters are set in a dialog box.
- An arrow (triangle) behind a command indicates that a cascading menu will be opened and a variant of this command must be selected.

The Toolbar

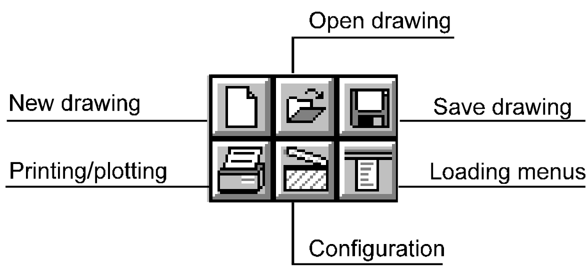
The toolbar has its position directly underneath the menu bar. It is subdivided in several areas and contains two groups of symbols, the command line area and the option bar.

The symbols represent often-used commands. Their arrangement in the Function Bar permits these commands to be executed without opening a menu. Simply clicking on the symbol with the mouse will execute the function.

In the single symbol groups the corresponding commands are arranged by subject. This explanation treats only the meaning of the individual symbols. You will find detailed explanations of the commands and their use in the following chapters of the manual.

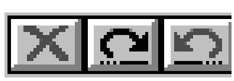
System Buttons

The first group of six buttons are located on the left side of the function bar. These are commands from the menu *File* which refer mainly to the opening, saving and output of drawings and to the configuration of the desktop.



The functions *New drawing*, *Open drawing* and *Save drawing* have already been explained at the previous paragraphs. *Printing/plotting* drawings refers to the output, and will be explained in later in this manual. Explanations of the *Configuration* of the desktop, as well as menu customization details are found in the *Programmer's Manual*.

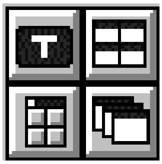
Beneath this first symbol group are three additional symbols, which pertain to execution of commands.



These are (from the left):

- Cancel the execution of the command (ESC);
- Withdraw the last work step (Undo);
- Restore or nullify the withdrawing of the last work step (Redo).

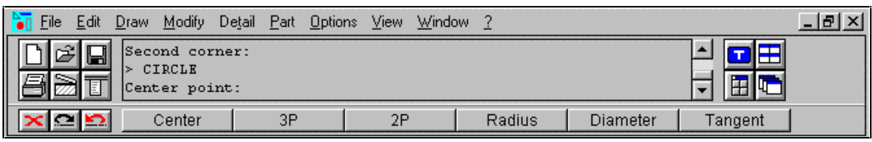
Another group of four symbols is arranged next to the command line. These symbols are for the display of the desktop and the arrangement of the drawings, and are standard Windows functions.



- Switch between Command Line and Text Window;
- Call palettes;
- Arrange drawing windows in tile format;
- Arrange drawing windows in cascade format.

Command Line and Text Window

The command line and the text window, serve as communication between the program and the user. In its default condition, the program uses a command line situated directly underneath the menu bar that looks like this:



The button **T** on the right of the Text Window will allow you to switch between this mode of display and a bigger text window, placed freely on the Windows desktop.

The default command line is fixed in its position on the desktop. It is possible, depending on the chosen script type, to change this. With the help of the scroll bar on the right side of the Text Window, or with the keys *PageUp* or *PageDown* the registrations can be "rolled" up and down.

The text window can be arranged independently on the video screen, and the size of the screen can be adapted to specific demands. The procedure for this is described in the Windows manual. There are no differences between these two alternative forms of input field.

This field allows you to:

- Specify points and values, select objects, and confirm inputs;
- Display the selection or input steps by the user (for example, *4 selected*);
- Receive messages, indications or warnings from preceding steps user (for example, *Command not found).*

All inputs made by you (commands, functions, options, selection sets, coordinate inputs etc.) will show in the text window, so that they may be verified before they are confirmed.

At the same time the command line or the text window is fulfilling a recording function, you can read or reconstruct program steps of the working session including inputs made.

Again, with the help of the scroll bars on the right side of the text window or with the keys *PageUp* or *PageDown* the inputs can be scrolled. When performing this function, the text window has the advantage of greater clarity.

The command line is not to be used for large output lists, as presented especially by the Info commands. If such functions are implemented, dialog boxes are used.

For example, dialog boxes are used when:

- displaying or setting current adjustment variables;
- listing the geometrical information of selected entities;
- displaying status; and
- for information about the table of the drawing data base (defined layer, loaded line types or text styles, defined views and user coordinate systems).

Reference:
In the following chapters of this documentation the terms command line or command line area and the term text window will be used as a synonym. If there are any differences in the function of these terms the correct term will be used, and the difference spelled out.

The Context Bar (Option Bar)

The context bar is a special area of the desktop located underneath the command line. It contains context-sensitive option buttons. Using those buttons enables you to call options when a command is performed.

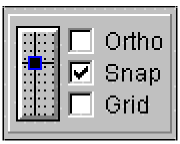
After a command has been invoked, the options currently available are displayed in the option bar. Clicking on the desired option will select that option.

Hotkeys may also be used, if they are assigned to the options.

The option fields will be changed to correspond with the current status of the drawing. Doing this keeps the number of options shown manageable. If there are no options to the command, these option fields are empty. The option field may change as often as after each step in the drawing.

The Control Panel (Side Bar)

A control panel is situated on the left side of the screen. This side bar includes a permanent series of command or function buttons. The symbols arranged there can be subdivided into three groups:

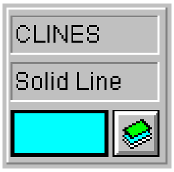


Drawing tool symbols

Clicking the button at the left side opens the dialog box for setting or adjusting the precision aid functions (snap, grid, orthogonal mode and running object snap functions).

Also, in this area you can directly control the Snap and the Grid status as well as use of the Orthogonal mode. These functions are activated by clicking on the appropriate check box. A marker in the check box indicates an active condition.

Explanations for the adjustment of these functions and the their use in drawing may be found in the following chapter 2, *Drawing with Precision*.



Property tool symbols

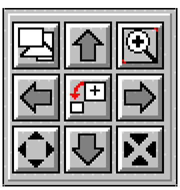
The top button of this symbol group allows to set the **current layer**. After selecting this function, the selection box with the layers currently existing in this drawing will be displayed. You may choose the layer in which you wish to work.

The second button allows to set the current **linetype**. The normal situation it will be the linetype defined by the current layer. After choosing this button, you can select explicitly a running linetype in the dialog box displayed.

Beneath this button (on the left) is the symbol for setting the line **color** applied when creating new entities. With this button you can define a line color that differs from the color defined for the current layer.

On the right is the button for layer control. Clicking this symbol will open the **Layer Manager** dialog box. In this dialog box you can create, change, and rename layers as well as define their status, the layer color and the type of line.

You will find detailed references to the layer concepts, and the linetype and color properties in the chapter 4, *Layer and Object Properties*, of this manual.



Symbols to control the drawing views

The third group of symbols serves exclusively to control the drawing views. The individual symbols correspond to commands of the menu *View*.

The four arrows represent the commands for horizontal or vertical **Pan**. Clicking one of the arrow symbols will shift the drawing in that direction.

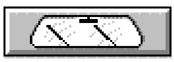
The symbol at the left means **Zoom Fit**. Clicking this symbol will cause the drawing to be displayed so that all of the included entities are shown as large as possible on the drawing surface.

The symbol at the right means **Zoom Window**. After selecting this mode, you must define the area to be “zoomed” by selecting two points representing opposite corners of the enclosing box.

The symbol in the middle of the second row represents the command **Zoom Back**. This command restores the previous view.

The two symbols in the third row, left and right, contain the function **Zoom Out** and **Zoom In**. These commands perform step-by-step increasing or reducing of the screen display by one factor. You can define that factor in the function display modes.

The explanation of these commands are given in chapter 3, *Viewing Drawings*.



Redraw

This button which resembles a windshield represents the command **Redraw** from the menu *Window*. This command will cause the screen display to be reconstructed by those pixels currently in intermediate storage. Pixels which remain on the screen from deleted or revised entities will be deleted by this command. This is faster than a REGEN which will update the current viewport from the drawing database, or a REGENALL which updates all viewports. A REGEN occurs automatically when needed, such as when you zoom or pan the view.

The Status Line

The status line is situated at the lower side of the program's desktop. It contains a series of information fields. Brief help text will appear in the first field. This line also contains the coordinates of the current cursor position.

The first area from the left serves two functions:

- When the cursor is moved over the symbols of the symbol bars or palettes, a brief description of the commands and functions which will be executed by the chosen symbol is indicated. Information on the function of a command selected from any of the pull-down menu items are also displayed in this field.
- This information field will, in addition, display the current distance and angle between the starting point drawing function and the current cursor position.

This is helpful when you are using the cursor to input the coordinates.

The next information field shows the currently used coordinate system.

- WCS = World coordinate system
- UCS = User coordinate system

The remaining three information fields specify the X-, Y- and Z-coordinates of the current cursor position, in that order.

The Title Bar

Every drawing window has at the top a Title Bar. The name of the file will be indicated in this Title Bar. In front of the file name are the complete specifications of the path, with information about the drive, the directory, and any sub-directories in which the file was saved.

If there are several views of one drawing, the separate viewport windows are marked by continuous numbering. This label will appear after the drawing name and will be separated from the name by a vertical line.

When a drawing window is maximized the described specifications will be found in the Title Bar of the desktop. If you have opened several files simultaneously, the specification of the file name will facilitate having an overview of the drawings that are working at the time.

The Drawing Area

The Drawing Area is the part of the desktop in which drawing entities are created and modified.

Up to four drawings with four viewports each can be opened simultaneously in the Drawing Area. Each drawing and each view will be represented in its own window. The individual windows can be tiled, cascaded, or enlarged to fill the Drawing Area

If more than one window is opened, only one will be the **active** window. The active window may be recognized by the presence of shading in its Title Bar. Only drawings in the active window will be working. A window is selected as the **active** window by clicking with the mouse or the digitizer on any part of the window or (if there is full screen view) via the menu *Window*.

Outside of the drawing areas the cursor assumes the typical Windows pointing shape. Within the drawing area the cursor appears as cross-hairs.

Coordinate Symbol

Normally, *FelixCAD* puts a coordinate symbol in the lower left corner of a drawing. This symbol indicates the position of the axes of the coordinate system and provides orientation of the alignment of the current coordinate system.

You can classify the individual axes of the coordinate symbol by different colors, which increases the clarity of this symbol. The coordinate symbol can be faded in and out by the transparent command TICON (Toggle Icon).

Calling Commands and Controlling the Sequence of Operations

The following paragraph explains the most important steps for the control of the sequence of operations. The sequence of operations refers to:

- Calling commands and options;
- Defining points, distances, angles etc.;
- Selecting objects and entities.

The program offers several options to do these tasks. These options differ in regard to the input medium or the use of different elements of the desktop, not in regard to the achieved results.

All of the inputs that control the program can be performed by either keyboard or some means of cursor control. Or, these possibilities can be combined. For instance, when drawing a line you can define the first point by typing in the coordinates of the start point on the keyboard and by "pointing" the second point with the cursor control. The selection of input device is your own decision.

The following explanations will deal with the differences of input by keyboard, or by cursor control device. Input by these devices will be treated as one, and be called "pointing device".

In the remainder of this section “keyboard input” will refer only to the input via keyboard in the Command Line area. The control of the typical Windows elements of the desktop will be done through the use of a pointing device. For instance, opening the menu *File* by the key sequence ALT+F will be assumed the same as clicking on the menu field *File* by pointing device and will be referred to as “**Selecting the Menu”.**

Functions of the Mouse Buttons

Left mouse button

The left mouse button serves to indicate points, select objects, and call for functions and commands via pull-down menus, palettes and other symbols.

Right mouse button

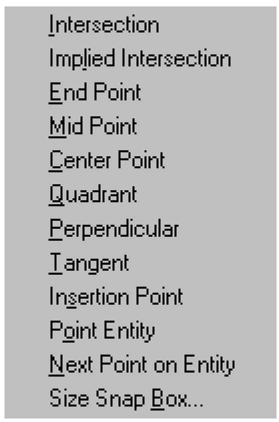
The right mouse button corresponds to the ENTER key. It is used to:

- Confirm input and output operations;
- Take over preset values;
- Repeat command.

Middle mouse button (or CTRL + Left mouse button)

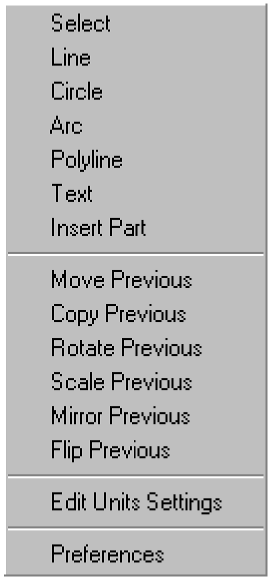
During the execution of a command points frequently are retrieved. Pressing the middle mouse button during retrieval will call a selecting window with object snap functions. This allows easy use of the object snap functions during the work session.

If the system only has a two-button mouse, the selecting window with the object snap functions can be called by simultaneously pressing the CTRL-key and the left mouse button.



When no command is running and no prompts are available, the middle button on the mouse or the CTRL-key and left button on the mouse, will bring up an additional Cursor menu. This makes it easy to grab some of the more popular commands without having to move your cursor.

The commands will be explained later in this manual, with more detail, in the appropriate sections.



Options in Calling Commands

Keyboard

Unlike many Windows applications, *FelixCAD* allows you to call commands via keyboard input. In addition to improving the working speed and simplifying the macros this technique also allows better precision.

Most of the commands, those for the creating, modifying and detailing of drawings, as well as the LISP expressions, can be called via keyboard input. Some specific Window operations functions can be called via symbol buttons (in the title bar and the function bar) or via accelerator keys (window arrangements, help, clipboard functions). This method is familiar to most Windows users.

To call commands via keyboard enter the command name behind the command prompt in the Command Line area. It looks like this: **>**

You will then be asked to enter further inputs, which would normally be input with the keyboard or pointing instrument.

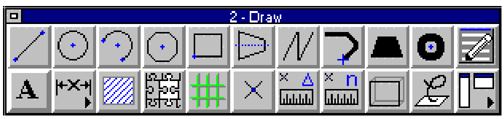
Pull-Down Menus

Commands (and their options) can be selected using pull-down menus. The menus and commands may be requested by key sequence (for example: ALT+F, N for new file) or by clicking the proper option field within its menu. For some commands, cascading menus are available. These sub-menus offer further commands or variations of the command.

Symbol Buttons

A series of frequently used commands can be called by clicking symbol buttons. There are two kinds of symbol buttons on the desktop:

- Permanently displayed symbols in the toolbar or the control panel (side bar). These buttons represent mainly commands from the pull-down menus *File* and *View*.
- Symbols in palettes, arranged in any way, which can contain any command for which the palette is designed.



Commands and functions are called from a palette by clicking on the symbol.

Transparent Commands

A series of commands and functions are at your disposal even when another command has been called and is being executed. These commands are called transparent commands.

They include commands and functions for:

- changing the view onto a drawing (for example, ZOOM, PAN);
- functions to arrange the drawing windows (WCASCADE, WTILE);
- controlling precision (Grid, Snap, Ortho, Object snap functions);
- switching between Command Line area and Text Window;
- loading or closing palettes;
- calling the Info functions; and
- calling the Help function.

After the execution of the transparent command the program continues to execute the original command. If transparent commands are called using the command line, an apostrophe must be placed before the command. For instance:

```
'ZOOMOUT
```

Otherwise transparent commands can be called as usual by clicking the proper option in the pull-down menus or by choosing the proper symbol.

Transparent commands can **not** be **executed** if the program is requesting input of text strings or if a command's dialog box is active to request command parameters.

Repeating Commands

During practical experience with the program you will often need to repeat a command. To repeat the same command either by pull-down menu or keyboard entry would be annoying and inefficient.

You can repeat the last executed command by pressing the ENTER key or the right mouse button.

For example:

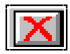
```
> LINE
From point: P1
To point: P2
To point: P3
To point: <hit Enter key>
> <hit Enter key>
LINE
From point: P4
To point: P5
To point: P6
To point: Close
```

Command repetition can be performed as often as is required. In several cases only the standard variant of the corresponding command will be repeated. Other options that were selected will not be used automatically. However, these options are at your disposal, and can be selected again.

Canceling Command Execution

It is possible to interrupt the performance of a command before the command has been completed. This can be useful if a command was activated by accident.

To cancel the execution a running command

- click the symbol  or
- press **ESC**

This will interrupt the execution of the current command, and at the command line you will see the message *** **Cancel** ***.

Withdrawing a Command (UNDO)

If you recognize that you have input an erroneous command, it is possible to withdraw the command.

The program archives all steps made during the task in a list called the Undo List. That list contains a registry of all of the working steps. The steps registered in this list can be withdrawn by using the UNDO function sequentially, each UNDO removing the next sequential step in the task.

Certain commands are not registered in the program's Undo List. These are commands and functions that control the viewport (Zoom, Pan, Window), set the parameters, and other transparent functions. These steps can **not** be withdrawn by using UNDO.

The Undo List is kept only with the current session of the program. It will not be saved with the drawing.

Calling the Command

- Click the Undo symbol 
- Select **Undo** from the *Edit* menu;
- Enter **U** by keyboard and confirm this input by ENTER.

Invoking this command will cause the last command made to be withdrawn and all changes in the drawing caused by this command to be canceled.

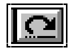
The command can be repeated until all working steps stored in the Undo List have been canceled. The program will then send out the message *Undo list is empty*.

Restoring (REDO)

This command offers the ability to REDO an UNDO. It cancels the last relevant UNDO operation.

The drawing state is restored to the last state prior to issuing an UNDO or DELETE command.

This is only possible if there has been no other command executed between the commands UNDO and REDO. REDO is used in this fashion:.

- Call the REDO command by clicking the symbol 
- Select **Redo** from the *Edit* menu;
- Enter the command name REDO at the command line.

Only the last relevant working step will be restored.

This is also possible if there have been other drawing or working operations executed between the UNDO and the UNDELETE.

Delete

During the creation of drawings, or, later, the editing, changing or detailing of drawings it may be necessary to delete certain parts of the drawing. The program offers the command DELETE to accomplish this operation.

Call the command

- by clicking the symbol ;
- by selecting the option **Delete** in the menu *Edit*; or
- by keyboard input DELETE in the command line area.

The command DELETE may be used for these drawing entities:

- Individual drawing entities, like lines, circles, rectangles, or for groups of these entities;
- Parts and attributes;
- Text objects; or
- Dimensioning and crosshatching.

To delete groups of entities, the respective group has to be converted to single entities again by the command EXPLODE (Explode Complex Objects...)

Object Selecting and Deleting

After calling the command the request to select the objects to be deleted will appear. The program offers (in the option bar) a series of methods for selecting objects.

For a deleting operation as many objects can be selected as you desire. When the object selection is complete the selected entities will be deleted without another confirmation, by simply pressing ENTER. You may wish to interrupt the delete operation by pressing ESC.

Undelete: Withdraw Deletions

Entities which have been deleted by mistake can be restored by the command UNDELETE.

Call the command

- by clicking the symbol  ;
- by selecting the option **UNDELETE** in the menu *Edit**;*** or
- by entering UNDELETE at the command line.

This will cause all entities previously deleted to be are restored.

The command DELETE can be canceled by the function **Undo** (command U). However, only those operations can be canceled which are stated in the UNDO list (see Chapter 1, ***Withdrawing a Command***).

Chapter 2

Drawing with Precision

During the drawing process it is often necessary to specify or modify the value of data. This occurs when the program needs input values for solutions. In those cases a prompt will appear in the text window asking for the entry of a value. Examples of these required values are:

- Coordinates for defining points of the drawing (start and end points, center points, insertion points, base points and target points)
- Distances
- Angles
- Width values (for example, line width)
- Numbers

In this chapter data input is discussed in detail.

Also, this chapter describes a series of commands and functions which serve as drawing aids. The commands and functions that serve as drawing aids are:

- precision aids (grid, snap grid, orthogonal mode and object snap functions);
- object snap function; and
- deleting and undeleting of drawing objects.

Most of the commands and functions described in this chapter do not create or modify drawing entities. Exemptions are the delete/ undelete commands, which act directly upon the drawing objects.

The settings (grid, snap) and functions for selecting and snapping objects do influence the process of drawing and editing functions and can make the drawing procedure much more precise.

Data Input

Coordinate Input

The different coordinate types can always be entered as absolute or as relative coordinate values.

Absolute Coordinates use as their reference (starting) point the origin of the current coordinate system, or, in the case of angle values, the angle to the XY-plane of the coordinate system.

Relative Coordinates use as their starting point the last defined point. Relative coordinates are selected by adding a sign **@** before the value.

The input @ 10, 10 defines a point which has a **distance** of 10 drawing units in the positive X- and Y-axes from the last entered (or "trapped") point.

Coordinates are entered in the following formats:

- **Cartesian Coordinates**
  Cartesian Coordinates define a point by its X-, Y-, and Z-values. These values are entered separated by a comma. A point is used as a decimal sign.
  If no Z-value is entered it will be interpreted as Z=0.

- **Polar Coordinates**
  Polar Coordinates define a point by its distance from the coordinate origin and the angle in the XY-plane (from the X-axis). The two values are separated by the sign < (smaller than).

  For example, the value input ***10<60*** defines a point with the distance of 10 drawing units from the origin of the coordinate system and an angle of 60 degree from the X-axis.

- **Spherical Coordinates**
  Spherical Coordinates define a point by its distance from the coordinate origin, the angle in the XY-plane, and the angle to the XY-plane.
  For instance, the value input ***10<60<45*** describes a point with the distance of 10 drawing units from the origin of the coordinate system, an angle of 60 degree from the X-axis in the XY-plane, and an angle of 45 degrees from the XY-plane in direction to the Z-axis.

- **Cylindrical Coordinates**
  Cylindrical Coordinates define a point by its distance from the coordinate origin, the angle in the XY-plane and its Z-value. Distance and angle are separated by < , angle and Z-value by comma.

  For example, the input ***10<60,5*** defines a point with the distance of 10 drawing units from the origin of the coordinate system, an angle of 60 degree from the X-axis in the XY-plane, and a Z-value of 5 drawing units.

The following table gives an overview of the input formats of the different coordinate types, both for absolute as for relative coordinates.

Type of coordinate	Input	Example
Absolute Coordinates		
Cartesian	x,y,z	3,8,6
Polar	distance<angle	7.5<45
Spherical (3D)	distance<angle,Z-coordinate	7.5<45,12.2
Cylindrical (3D)	distance<angle1<angle2	7.5<45<33
Relative Coordinates		
Cartesian	@deltaX,deltaY,deltaZ	@3,8,6
Polar	relative distance<angle	@7.5<45
Cylindrical (3D)	rel. distance<angle1,Z-corrd.	@7.5<45,12.2
Spherical (3D)	rel. distance<angle1<angle2	@7.5<45<33

Distances

Several commands require that distances be input. This can be done by entering numeric values or by pointing the distance with the mouse (or the digitizer's pointing device) from a base point or between two points. Entering a numeric value for a distance allows both whole numbers (integers) and real numbers to be used. The input can be made by decimal or scientific notation, or as a fraction.

Width Values

Definition of width values by numeric input or pointing is done the same fashion as defining distances.

Angles

Angles can also be defined by the input of numeric values or by pointing the two points.

Entering numeric values requires the use of the sign < before the value, for instance, 10<45.

Angles are normally specified in degrees, counter-clockwise from the starting point. The angle 0 is therefore located at the right (in direction of the positive X-axis) of the starting point.

The relative value of the starting and ending points of an angle will also go in a counter-clockwise direction.

Numbers

If a command requires the definition of a discreet number, for instance, the number of entities for the placement of sides of a polygon, then the value required must be a whole number value.

There are two different methods of inputting these values:

Input by keyboard

Input by keyboard has the advantage of precision. Using this method, you can construct drawing entities with exact dimensions or with exact placement in the coordinate system.

Values for distances, straight lines and other linear quantities are specified in drawing units, angle values in degrees.

Coordinates are given, in sequence, as the values for the X-, Y- and Z-axes with a comma between.

Please note that a point is used as a decimal sign. A comma serves as dash between the values for the single axes.

When specifying an input value for an angle the sign < (smaller than) comes before the value.

An input sequence on the keyboard might look like this:

```
> LINE
From point: 2.5,0
To point: 5.75,0
To point: 0,3.25
<hit Enter key>
```

Moving an object might would look like this:

```
Select objects: Selection
Basis point: 5,2
Target point: @3,0
```

Pointing or Identifying

Pointing, or identifying, is taken to mean the input of points, straight lines, angles and other quantities by mouse or digitizer or other cursor control device.

This form of value input is recommended when you must relate existing drawing entities. For instance, the center of a circle might be defined by the intersection of two lines. In this case it is easier to indicate a point resulting from the construction in order to define its coordinates.

The coordinates of the current cursor position will interpreted directly as a pair of coordinates or, in some cases, as differential to the coordinates of a starting point. The clarity of this kind of value input is guaranteed by the **dynamic preview**. In doing this you will get a kind of **preview image** of the result of your action.

Using the object snap functions assures that the accuracy of the coordinate input by pointing will be improved. The construction points like the starting or ending points, the intersection points, the center points, the perpendicular point etc. can be “pointed” with acceptable precision.

A summary of the available object snap functions and their use appears in Chapter 2, *Drawing With Precision*.

Coordinate Filters

The use of coordinate filters are offered when point specifications are requested. Coordinate filters allow you to relate the point value inputs to the X-, Y and Z-coordinates of existing entities.

Some possible inputs that might be requested for point values are:

x, value	x, y
value, y	x, y, z
value, value, z	value, y, z
x, value, z	x, y, value

If you answer the question with the input:

x,10

you will be requested to enter the x-value separately. It is now possible to catch a second object or the geometrically significant point of an object by using an object snap function. The X-value of this point will be interpreted as input value for the original point definition (x,10).

```
> LINE
From point: P1
To point: x,10
X-coordinate: end
of: P2
To point: 15,y
Y-coordinate: cen
of: P3
```

Relative coordinates to a given base point

When you are prompted to enter a point, the option *from* allows you to identify a relative point from a given base point. For example, in a running line command you see these prompts and responses:

```
> LINE
From point: from
Base point: Identify, usually with in object snap function
Relative coordinates: @3,3
To point:
```

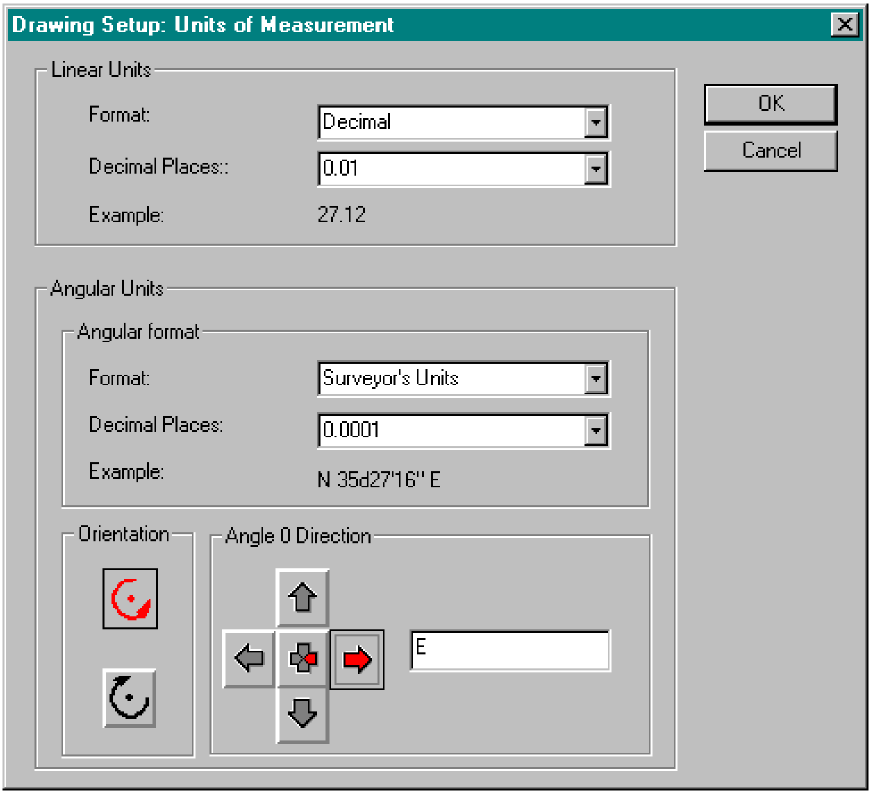
The option *from* differs from object snap functions in so far as it does not expect identifying of objects (with the snap box). Rather than this the from option serves to retrieve a relative point from a given temporary base point which then is returned to the running command.

When specifying the *base point* the absolute and relative coordinate input as well as the object snap functions can be applied as usual.

But, when you are prompted to specify the *relative coordinates* you should pay attention to specify **only relative coordinates** at this point. In practice, this means to make sure to use the @-prefix when entering the coordinate (the different formats of relative coordinate input are summarized in a table above in this chapter). You can specify an absolute coordinate at the *Relative coordinate* prompt. However, this will return the specified, but not a relative coordinate. Also, because in fact the input of an absolute coordinates cancels the specification of an relative point, you should avoid to pick a point in the drawing or to invoke an object snap option at this prompt.

Precision and Units Settings

FelixCAD allows you to set the number of decimal places of accuracy used in dimensioning and the type of angular units used by running the SETUP command. These settings are saved with each drawing, so can be set differently in each drawing if desired. Here is the setup dialog box:



The Linear Units can be set from zero to eight decimal places using:

- Decimal Units
- Fractional Units
- Architectural Units (Feet and Inches)
- Engineering Units (Feet and Inches)
- Scientific Units (Scientific exponential notation)

The Angular Units can be set from zero to eight decimal places using:

- Decimal Degrees
- Degrees / Minutes / Seconds
- Grads
- Radians
- Surveyors Quadrant Angles (N,S,E,W with deflection from North and South)

Orientation

In addition, the system allows you to modify which direction causes the angles to increase. The default setting is for Counter Clockwise increasing angles. You can set it to increase in a clockwise direction by selecting the Compass Angles (CW) radio button.

Angle Zero Direction

The system allows you to specify a different direction for Zero degrees. This is of interest to different special disiplines. For example Surveyors normally set zero degrees as “up” on the pa

ge. This allows for the correct display of angle and distance when drawing lines on the screen. The display on the status bar is updated depending upon the settings of the angle zero direction and the orientation.

Surveyor Units Entry

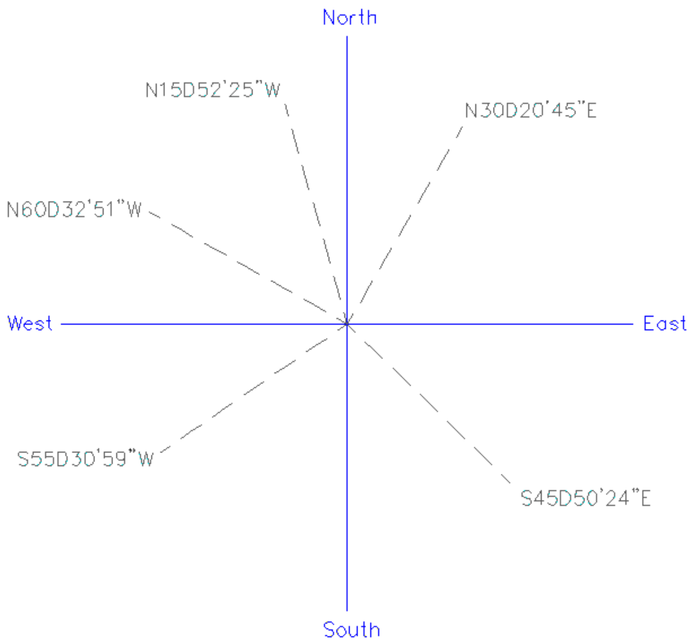
Because the Surveyor Units are so different form the other methods of entry we have included a few extra examples to help show the entry method.

Type of coordinate	Input	Example (Surveyor Units)
Absolute Coordinates		
Polar	distance<angle	7.5< N34D30’45”W
Relative Coordinates		
Polar	relative distance<angle	@7.5< N34D30’45”W

The entry of the Angle is done with survey notation where the N stands for North, the 34D is 34 Degrees from North, the 30‘ is an additional 30 Minutes from North, the 45“ is an additional 45 Seconds from North, and the W indicates that the angle is towards the West.

When entering the N or S, the D and the E or W, be sure to enter them as all capital letters. Small letters will not work.

There are 60 seconds in a minute, 60 minutes in a degree and 360 degrees in a circle. Surveyors divide this into 4 equal parts each with 90 degrees measured from the North or South poles.

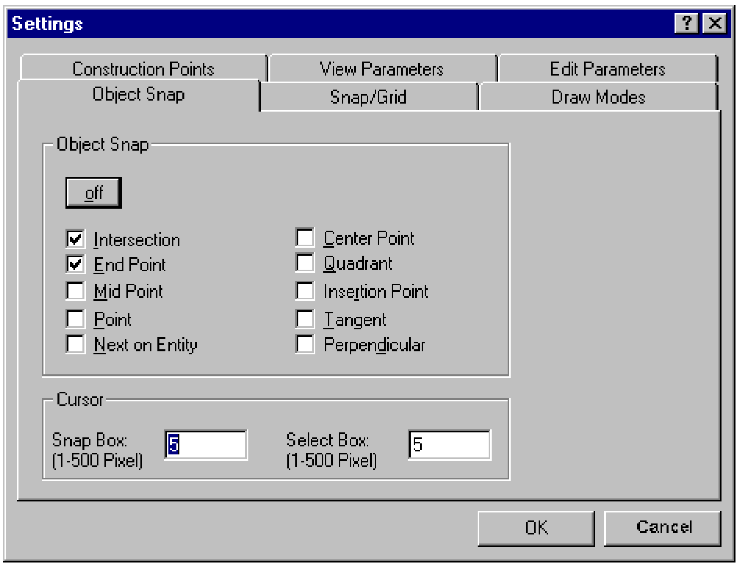


Above we see the four points of a compass, North, East, South and West. To further show how the Surveyor Units work, several lines have been drawn from the middle of the cross out in various directions. We have labeled these lines with the correct Bearings. Note that all the letters used in the bearings are in Capitals as required by the program for input.

Settings

The command SETTINGS, which can be selected from the *Options* menu or by entering SETTINGS, is the main command to set the parameters and modes of the drawing environment. There are tabs in the dialog giving access to the following commands described at other places of the manual:

Tab	Command	Described ...
Object Snap	PRECPAR	... in Chapter 2: Drawing with Precision
Snap / Grid	PRECPAR	... in Chapter 2: Drawing with Precision
Draw Modes	DRAWMODE	... in Chapter 5: Basic Drawing Elements
Construction Points	DRAWMODE	... in Chapter 5: Basic Drawing Elements
Edit Parameters	EDITPAR	... in Chapter 6: Modifying Drawing Objects
View Parameters	VIEWPAR	... in Chapter 3: Viewing the Drawings



Precision Parameters

The term *precision aids* covers functions which support you when drawing new entities, modifying already existing objects, and doing detailing operations (dimensioning, for instance). They help you precisely define specific points of a drawing.

The precision aids can be turned on or off and can used with fixed parameters.

The precision aids include GRID SNAP, GRID DISPLAY, SNAP MODE, ORTHOGONAL MODE, LIMMIN, LIMMAX and OBJECT SNAP functions.

Grid Snap covers the drawing area with a raster. With snap activated, a point definition made by picking with the cursor will select (“snap” to) the nearest grid point.

The distances between grid points and the zero point can be chosen.

Grid Display covers the drawing area with a visible latticed point grid and offers a visual impression of the distances, angles and other relations of the objects with themselves.

This is only a display function to help you get oriented. The distances between the points of the grid display can be set.

Limmin sets the lower left corner (limits) for the Grid Display area.

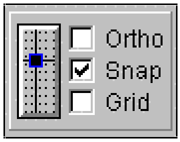
Limmax sets the upper right corner (limits) for the Grid Display area.

Orthogonal Mode limits the movement of the cursor so that it will only move parallel to the axes of the current coordinate system. This makes it much easier to design and place parallel, or “running”, lines or objects.

Object Snap functions facilitate finding and "catching" geometrically significant points of already existing drawing objects. These can be end points, intersections, center points, and so forth. The individual modes of object snap functions can be called for any single selection operation from the proper palette or via the cursor menu (middle mouse button) or be activated (permanently) using the dialog box *Precision Aids.*

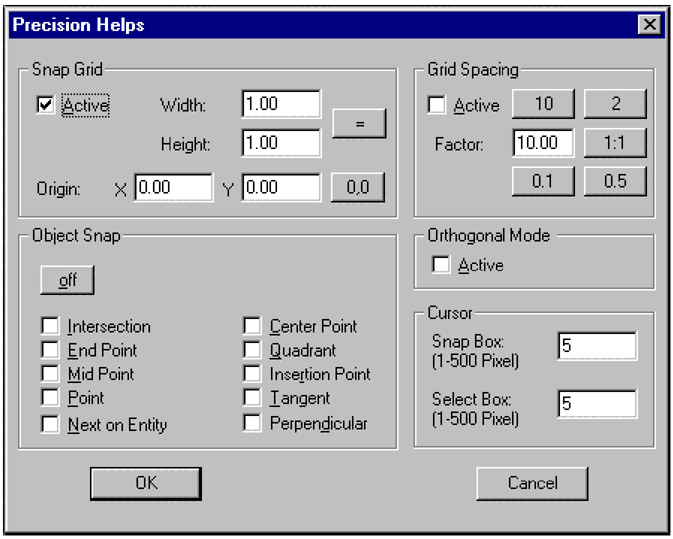
Control of Precision Parameters

The precision aids can be controlled by the common dialog box, which is activated by clicking the left button of the following shown symbol in the vertical control panel, or by the command PRECPAR.



With the buttons in the control panel on the left side of the screen the functions ORTHOGONAL MODE, SNAP, and GRID DISPLAY can be turned on and off. Selection of any of the aids with the cursor will open the dialog box “Precision Aids”.

Individual Precision Aids can be switched on and off and their parameters can be defined within this box.



Within this box, there are input fields and control elements for the individual settings. You may enter numeric values using these fields.

The control window allows you to switch on and off the individual functions. You must click on proper window to accomplish this. A check mark in the box means that the function is active.

Snap Grid

Snap Grid covers the drawing area with an invisible grid. With snap grid activated, the cursor can only select points which are positioned directly on the grid. This means that starting and ending point, center, and any other specific point which can be defined will always be exactly on a grid point.

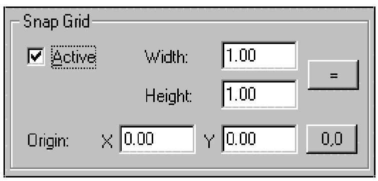
The distance between the point of the grid as well as the position of its zero point can be selected. The grid will always follow the axes of the current coordinate system.

The grid will not influence points defined by coordinate input. Even with snap grid activated, you can define points not on the grid by keyboard entry.

Activate or de-activate snap grid by clicking the check box on the upper left side in the area **Snap Grid.** A cross in the check box signals that the snap grid is switched on.

From the dialog box, select the input fields **Width** and **Height**, and enter values for the horizontal and vertical distances between the points of the snap grid. Units of measurement for these values are, again, drawing units. The value 1.0000 represents the distance of one drawing unit.

Using the palette button (=) for the value **Width** enables you to choose a vertical value equal to the horizontal value.



The input fields marked *Origin* in this box enable you to define the origin of the snap grid on the X and the Y-axis. The desired coordinates for the origin should always be entered.

Example: Entering the origin 1.0 (X) and 1.25 (Y), assuming the horizontal and the vertical distance between the points of the snapping grid is one drawing unit will enable you to pick points:

- whose X-coordinates have the value 1 or a multiple of it; and

- whose Y-coordinates exceed the whole-numbered coordinate values by 0.25 drawing units.

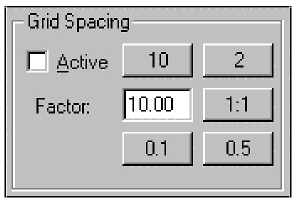
Note: Changing settings for the grid will only be effective if the dialog box is closed by pressing ENTER or by clicking the OK button.

Note: A grid rotated around a certain angle can be created with the option Z-Direction when setting a user coordinate system if you move to the top view.

Grid Display

Selection of **Grid** will furnish a visible Cartesian Coordinate grid. This grid offers a visual impression of the distances, angles, proportions and other relations of the objects to themselves. This facilitates the subdivision of the drawing area as well as aiding in creating and working with drawing objects that have free measurements.

You may select the distances between the points of the grid. The grid will follow the axes of the coordinate system.



Activate or de-activate the grid by clicking the check window on the upper left in the box **Grid Spacing**. A cross in this check box indicates the grid display is active.

In the input area **Factor** enter the value that defines the relation of the grid points to the snap grid. Alternatively, you can specify a pre-defined value by clicking available buttons.

Note:
Changing settings for the snap grid will only be effective if the dialog box is closed by pressing ENTER or by clicking the OK button.

The Grid will only appear in the area specified by the commands LIMMIN and LIMMAX. By defining the limits with these two variables you control the amount of the drawing that will be covered by the grid, when it is turned on.

Orthogonal Mode

Orthogonal Mode is a special tool to be used while drawing or modifying entities. It creates a defined relation between a starting and an ending point.

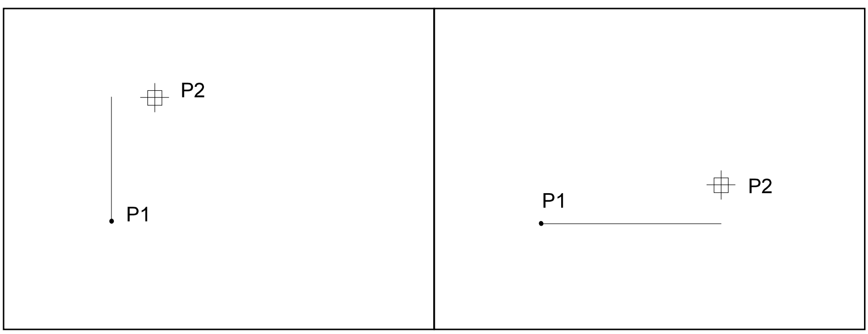
If orthogonal mode is activated, it is only possible to indicate points which are located on a line running parallel to one of the axes of the coordinate system. Such a line is also called an orthogonal line.

In the standard World Coordinate System (WCS) the orthogonal lines will run horizontally or vertically from the base point.

Note:
When using other coordinate systems the orthogonal lines may not run horizontally or vertically, but will always run parallel to the axes of the coordinate system in use at the time.

The rubberband line of the dynamic preview will clearly indicate which points can be chosen. Moving the cursor to a position off of the orthogonal will result in the next nearest point which lies on the orthogonal to be "snapped".

Note:
Orthogonal mode is effective only when picking points using the pointing device. Input of non-orthogonal points is possible by keyboard input while the orthogonal mode is activated.



Orthogonal mode aids in designing by allowing:

- Lines, chains and polylines to be designed along lines parallel to the axes of the coordinate system;
- Changes of direction when drawing lines, chains or polylines to be perpendicular to the current axes;
- Construction points for geometrical objects (such as radius, diameter, arcs, ellipses, ending points for polylines, chains and so forth) to always be parallel to the axes of the coordinate system.

When you are modifying objects, orthogonal mode means:

- Points for scrolling, scaling, rotating and so forth will be exactly parallel to or perpendicular to the axes of the defined base point. Objects or copies will be scrolled, stretched and so forth either parallel or perpendicular to the starting point.

The orthogonal mode is effective to use in drawing and modifying rectangular aligned geometry.

Object Snap Functions

Many functions and commands require the definition of an exact position in the drawing.

The determination of exact points can be done either by coordinate input or by indicating points. The input of coordinates has the advantage of precise placement, whereas indicating points often has an advantage in speed and ease of handling.

A very efficient method to take advantage of both techniques is to indicate points with precise references to a given geometry with the help of the object snap functions.

Object snap allows geometrically significant points of existing drawing entities to be located. These points are then used as inputs for a new drawing step.

Object snap functions can be activated permanently or be called for a single operation. These techniques are described in the section titled *Control of the object snap function* later in this chapter.

Mode of Operation

The object snap functions can not be called separately, but may be used only with a command which requests the input of a point.

The operating principal when using the object snap function is always the same:

- Call a command or a function which requires a point;
- When the point is requested, answer by calling the object snap function or a special mode of the object snap function. The cursor then takes the form of a selecting window. The command line area displays a message which requests the point;
- Select the object with the cursor.

Which geometrically significant point is "picked" during the activity depends on:

- the type of the drawing entity;
- the respective object snap mode; and
- the specific point of the object which is chosen.

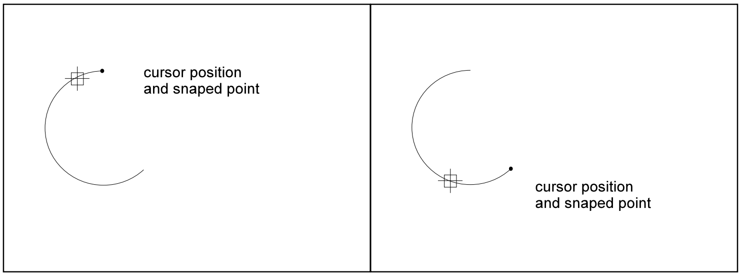
Object snap modes occasionally offer no practical use. For instance, you can not define the ending point of a circle, while a line segment has no central point.

On the other hand, it is possible that a drawing entity contains several points for which the criteria of the object snap function correspond. A line entity or an arc, for instance, always has two ending points.

(The object snap function does not differentiate in regard to the starting or ending point of an object like the drawing functions do.)

The object snap function selects, in those a cases, the nearest of the two or more possible points. It determines, when selecting an object by positioning the cursor, which of the possible points will be snapped. You should always place the cursor as near the desired point on the object as possible.

The illustration below shows the connection between the point selected and the point snapped:

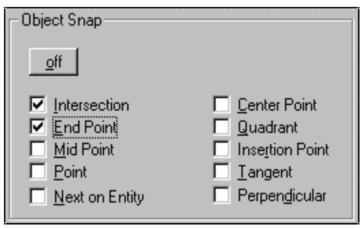


Controlling the Object Snap Functions

To locate a geometrically significant point on an entity, you can activate the modes of the object snap function in several ways. How the mode is activated will distinguish between the permanent activation and calling for a single operation.

Explicit Object Snap: Permanent Activation

Using the dialog box *Precision Aids***,** individual modes of the object snap function can be permanently activated. This is also called running mode.



Select the check box for the desired mode(s). A cross in the check box indicates that the mode is active. Clicking the button **Off** will de-activate all of the running object snap modes.

If several object snap modes are to be permanently activated, you can differentiate between the modes by the selected point of a object. For example, if the modes End Point and Intersection are activated, then if the point selected on the entity is nearest an end point, that will be selected. If it the selected point is nearest the intersection of entities, then the bisecting point will be selected.

Implicit Object Snap: Activating from Case to Case

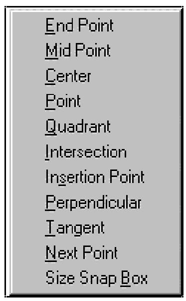
For one-time activation of a mode of the object snap function, you may choose between several options.

Snap Palette

You may use the Snap Palette to call individual modes of the object snap function for singular use in combination with a designing or editing command. This standard palette can be opened when needed and be placed anywhere on the drawing area. You may now select modes using the cursor. The desired object snap modes can also be included in user-defined palettes with the help of the Menu and Dialog Editor.

Cursor Menu

Another very effective method of calling individual object snap modes is by activating the cursor menu using the third mouse button (the middle button of the mouse respectively holding the CTRL key and clicking the left mouse button). The following menu is opened. From this list you can select an appropriate object snap function.



Keyboard Input

Using the keyboard to input a command abbreviation is another method of activating the object snap function.

The object snap modes and their abbreviations are shown in this table:

Object Snap Mode	Input
End point	end
Mid point	mid
Center point	cen
Quadrant	qua
Intersection	int
Implied Intersection	imp
Perpendicular to	per
Tangent to	tan
Next point on entity	nxt
Insertion point	ins
Point entity	pnt

The Object Snap Modes in Detail

When you are working with drawing entities there are differences between the object snap functions and the modifying commands or the object selection function.

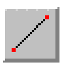
Object selection functions treats a group of entities (for example, a rectangle, a chain or a polyline), as **one** object, presupposing that the drawing entity was created by the corresponding command and not by individual lines. Therefore, defined objects will always be selected and handled "as a whole".

With regard to the object snap function, the mention of “entities” always refers to separate lines. This is also true for the polygons and the 2D and 3D faces. The edges are interpreted as “lines” and can be selected as relating entities. In this way the object snap functions can be used for the edges of planes as well as for segments of chains and polylines.

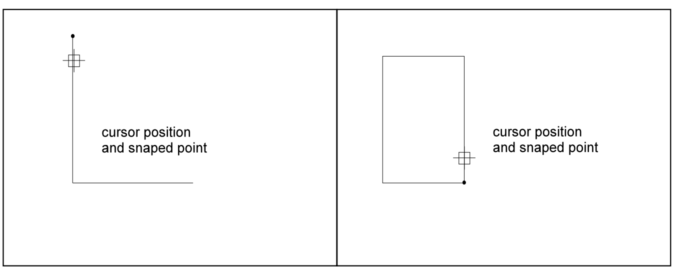
In the following explanations of the use of individual object snap modes this difference will be assumed.

For simplicity and clarity, this text will refer, for instance, to the *End point of a line* and not repeatedly explain that this term may also refer to the segment of a chain or polyline or the edge of a plane. This will be true for center points and intersecting points of lines, as well.

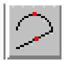
End Point (end)



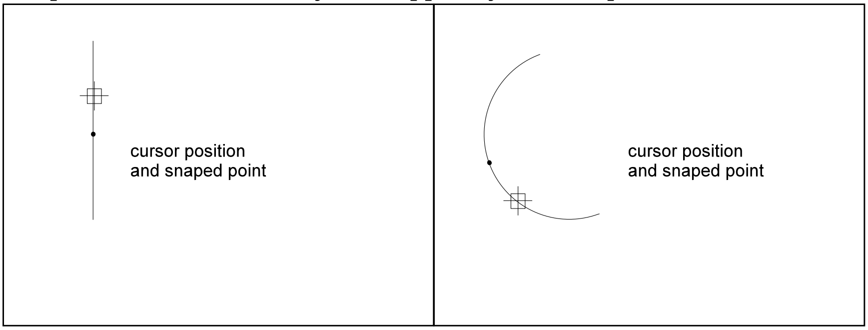
The object snap function **End** snaps to the end point of a line or of an arc. Planes or the edges of faces are also interpreted as lines, and end points of planes and faces can be trapped by the *End point* mode.



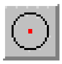
Mid point (mid)



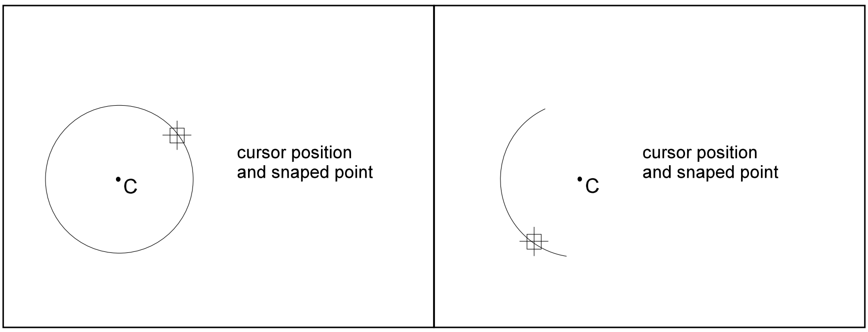
Snaps to the center of a line or an arc. Planes, or the edges of faces, are also interpreted as lines, and may be snapped by the *Mid-point* mode.



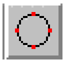
Center Point (cen)



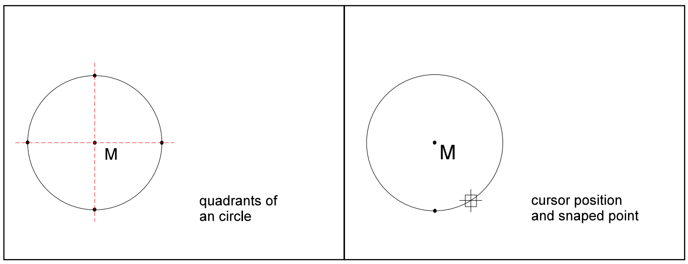
Snaps to the *Center* point of a **circle** or an **arc**.



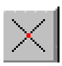
Quadrant (qua)



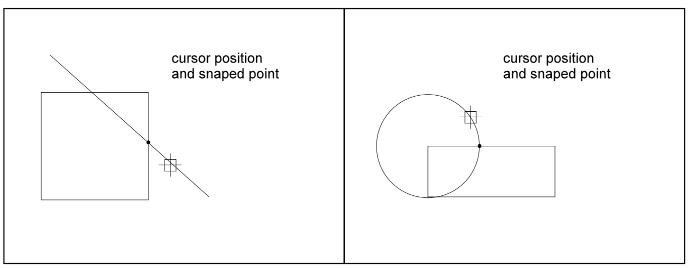
Snaps to the nearest *Quadrant* of a **circle** or an **arc**. Quadrants refer to the points of a circle or an arc, situated on the circumference at 0, 90, 180 and 270 degrees. These are the intersecting points of the axes of a coordinate system upon whose origin lies the center of the circle or the arc with the circumference of this circle or arc. Only the next visible quadrant of an arc can be snapped.



Intersection point (ins)



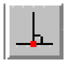
Snaps to the *Intersection* points of **lines**, **arcs** and **circles** or any other combination of them.



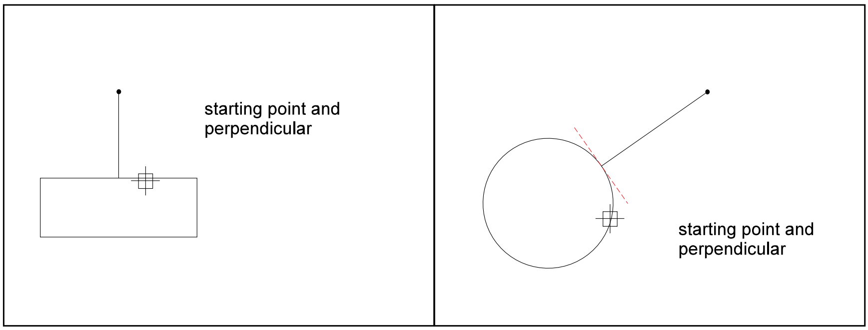
Implied Intersection (imp)

Whereas the object snap option *int* only returns an intersection point of objects that have a real intersection, the object snap option *imp* snaps - in addition -also to an implied (or implicit) intersection point. The *imp* function returns the point where the extension of two entities would intersect.

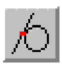
Perpendicular to ... (per)



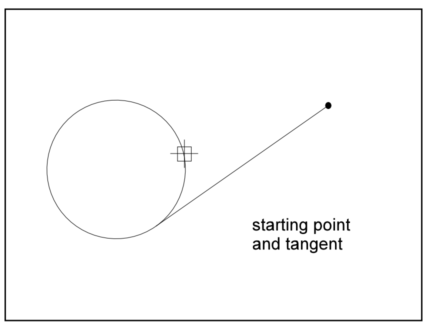
This mode requires that a starting point for a function has already been selected. Activating the object snap mode will relate this point to the snapped point. For instance, when a perpendicular is desired, the ending point for the perpendicular is picked, and the object snap mode *Perpendicular* is chosen. The result will be a perpendicular line from the ending point to the chosen entity.



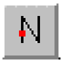
Tangent to... (tan)



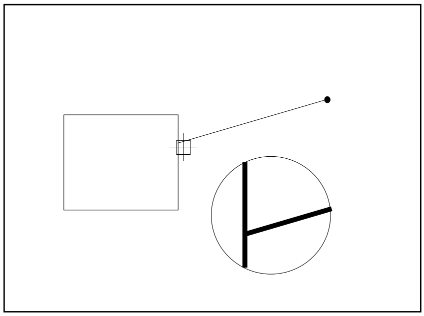
Similarly, this mode requires that a starting point for a function has already been selected. Activating the *Tangent* object snap mode will relate this point to the snapped point. For instance, when a tangent to a circle is desired, the ending point for the tangent line is picked, and the object snap mode *Tangent* is chosen. The result will be a tangent line from the ending point to the chosen entity.



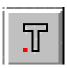
Next Point on Entity (nxt)



This mode snaps to the next situated point of an object when at least one point of this object lies within the borders of the selecting cursor box. This ensures that entities which are designed or modified by use of the object snap function share actually a point with the selected object. If you select a point on the space without using this object snap function it is possible that the selected point is not on the desired object. The two objects would not have a common point, or a “closure”.

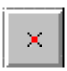


Insertion Point of a Text or a Complex Object (ins)



Snaps to the *Insertion* point of a part insertion, an attribute or a text object.

Point (pnt)



Snaps to a *Point* which was created by the command POINT.

Object Selection

Many commands, especially the commands for modifying and detailing of objects, presuppose that the objects to be worked on are **selected** or **marked**. This can be done by the function object selection.

Object selection will always be activated by the program when the current operation requires selection of entities. The request for selection occurs at the command line with this prompt:

```
Select objects:
```

Simultaneously, the program displays functions which may be useful in the current operation in the context bar (option bar).

You will recognize the continuous display of the request for selecting objects when marking entities.

All objects selected in subsequent operations will be added to the selecting set. The selecting set includes the sum of all entities selected for an operation. Using the Add/Remove status, it is possible to change the selecting mode between **Adding Objects** and **Removing Objects** from the current selection.

Selected entities will be shown in color. After marking all desired entities you may finish the selection by pressing ENTER: The program will continue with the execution of the command.

Object Selection Modes

In order to select objects you can employ several modes of OBJECT SELECT. Two standard modes are at your disposal immediately. Other modes are available in the option bar. The modes can be called by:

- Selecting the proper switch;
- Inputting of the name of the object-selecting mode;
- Inputting the abbreviation for the name of the object-selecting mode.

The options as well as the command abbreviations are:

All	a
WPolygon	wp
CPolygon	cp
Fence	f
Last	l
Previous	p
Add	ad
Remove	re
Window	w
Crossing	c
Single	si

During the selection of objects by these modes two proceedings can be employed.

Window / Crossing

- This mode, based on the **Window** principle, will mark all entities which lie entirely **inside** of the window. Entities which are touched by the selecting frame or are situated partly outside will not be selected.
- The selecting mode based on the principle of **Crossing** will mark all entities which **cross,** or are intersected, or lie totally **inside** of the window.

After calling the object selection function using the proper command, the option bar will offer these selecting modes:

WPolygon	CPolygon	Fence	Last	Previous	Remove

Simultaneously, the cursor takes the form of a little box. This indicates that you are in object selection mode.

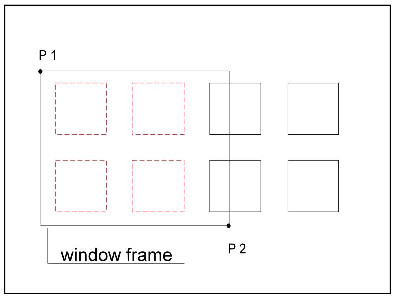
The standard modes **Window** and **Crossing** are available during the whole selecting procedure and can be used in any sequence. All other modes are active only for the selecting step. Later on the two standard options will again be at your disposal.

Window (W)

A (rectangular) **Window** is defined by two diagonally-opposite corner points. Position the cursor on the first corner of the **Window** and select that point.

You must take care that no object is inside the cursor window (box) during the selection of the first corner point. Otherwise, the option **Crossing** will be active instead of the option **Window**, and the object which is situated in the cursor box will be marked.

Determine the second corner point of the **Window**. This will select all of the entities which are entirely within the **Window** described.



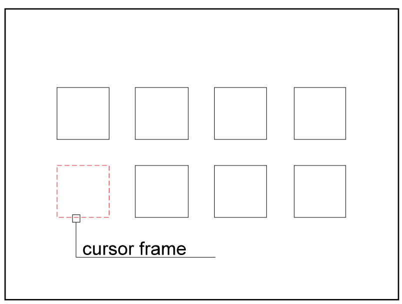
The command line will again display the prompt *Select Objects:*

Continue the object selection operation with the same or another selection mode, or terminate it by pressing ENTER.

Crossing (C)

Crossing will select only entities whose contours touch, or “cross”, the window frame will be chosen.

Position the cursor so that the window crosses the contour of the desired drawing entity. Select to mark the object.

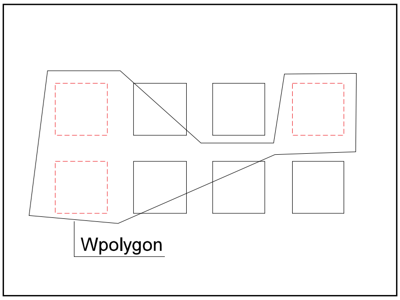


An object will not be marked if the cursor is just situated on the area which is **included** by the contours of the entity, without touching the contour itself. In this case the program will activate the selection mode Window and interpret the input as the first corner point of the window.

WPolygon (WP)

The mode **Window Polygon** functions in a manner similar to the option “Window”. Again all drawing entities will be marked whose contours are **entirely inside** of the selection window.

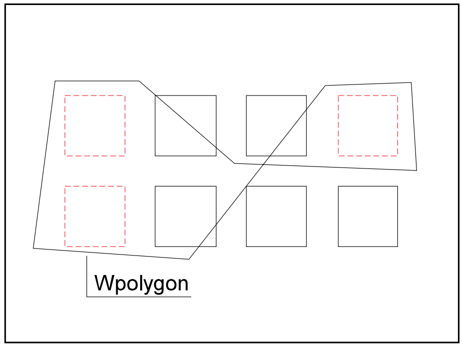
The difference between the two modes is in the form of the selecting window. The mode **Window Polygon** allows you to design an irregular window with as many corner points as are desired. This allows a more flexible selection than does **Window**.



To use **WPolygon**, define, in succession, the corner points for the selecting window. The **Preview** mode (from the option bar) will offer a preview of the emerging polygon.

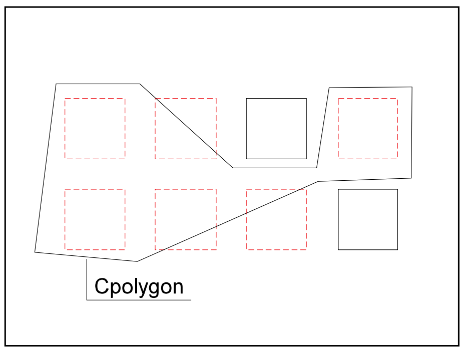
The corner points for the selecting window can be defined in a clockwise or counter-clockwise series. The program allows the polygon lines to cross or intersect themselves and to shape "loops" (see the following picture).

ENTER will conclude the definition of the selecting window. All objects that lie totally inside of the window will be marked.



Cpolygon (CP)

Crossing Polygon will create an irregularly formed window with as many corner points as you like. All objects that lie **inside** of the selecting window or are **crossed** by the window frame will be marked.



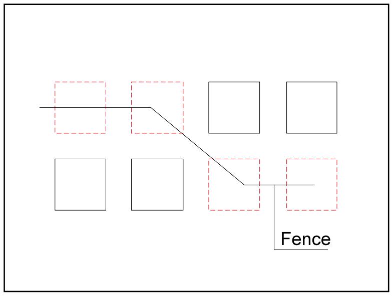
Define in succession (clockwise or counterclockwise) the corner points for the window. The preview mode will offer a preview of the emerging polygon.

The program allows the polygon lines to cross or intersect, and to shape "loops" (see the following picture).

ENTER will conclude the definition of the selecting window. All objects that lie totally inside of the window or are crossed by it will be marked.

Fence (F)

Fence represents an effective variant of a crossing selection mode. The object to be selected will be not be closed or crossed by a window, but only "cut through" by a line, or “fence”.



The line of the fence can be defined by as many points as you desire, and the fence may intersect or cross itself.

ENTER will conclude the definition of the fence line. All objects which are touched or crossed by the line will be marked and added to the selecting set.

Last (L)

The object selection mode **Last** marks the last drawn entity and adds it to the selecting set.

It does not have a memory function; that is, it is not possible to add to the selecting set the entity created during subsequent “last” operations by repeated calling of the option.

Note:
If the last drawn entity lies on a “frozen” layer you will receive the message in the command line area that no entity was selected.

Previous (P)

The option **Previous** (P) restores the previous selecting set. All entities are marked (and highlighted), which have been selected by the previous selecting operation.

Note:
This mode has the same restrictions as the modes Last and All, in that entities which are located on a frozen layer will not be marked, entities on a locked layer will be marked but cannot be handled.

All (A)

All, called by the input **all** or the abbreviation **a,** marks all entities included in the current drawing with the exception of those located on frozen layers and adds them to the selecting set.

Remove Mode (R) / Add Mode (A)

The last field of the option bar contains a toggle button to switch between **adding objects** to the current selecting set and **removing objects** from the selecting set.

The mode ***Add Objects*** is the default active mode, so you can select further objects and add them to the selecting set at any time.

Select the button **Remove** in the option bar to change the selection mode to ***Remove Objects***. Objects already selected and marked will be removed from the selection set by this change.

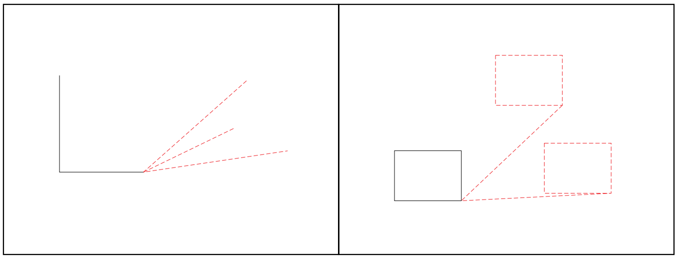
Single Selection Mode (SI)

The option **Single,** which can also be called by the abbreviation **SI,** de-activates all object selection modes for the next and subsequent operations, with the exception of the standard modes Window and Crossing.

Note:
This option is not offered in the option bar, but may be selected by keyboard input.

Dynamic Preview Modes

Dynamic Preview mode is a visual construction aid which allows you to watch the result of the pending operation before it is executed, and, if required, to make corrections. Dynamic Preview also aids in user understanding of commands and options, and helps the user learn the program.



Rubberband: A Marking Line

In any command that requires selection of two or more points, a marking line (rubberband line) follows the cursor movement. This line connects the last selected point with the current cursor position.

This gives you a visual impression of the operation and allows a better estimate of distances and angles between the last fixed point and the following point.

The rubberband line is useful both in drawing and in modifying objects.

Used during the drawing process it illustrates:

- the ending point of the next segment of a line, chain or polyline;
- the radius or diameter of a circle;
- the distance between starting- and ending-point of an arc; or
- the next corner point of a 2D or 3D plane.

When used during modification of objects it helps show the distance and angle from one base point to a second point, as well as the target point for moving, copying, scaling or other modifying steps.

Object Preview Picture

The Object Preview Picture shows in advance the contours of an object to be designed or modified. It is possible to recognize the consequences of changing of parameters like the radius of a circle or like removing or changing the angle of a copy of the original object. This option allows you to interactively influence these changes.

Chapter 3

Viewing Drawings

These are the procedures described in this chapter:

- Working with drawing windows, creating new view windows or switching view windows
- Viewing the drawings from different directions
- Enlarging and reducing (Zoom) as well as moving (Pan) the visible drawing portion
- Creating a new drawing
- Use and creation of a coordinate system.

There are two options for viewing drawings. These options may be combined for any one drawing:

- Changing the viewing angle while keeping the drawing portion or the copy/scale. There are standard viewing angles. For example, the view from the front, from the top, the view from the left side, and so forth. It is also possible to define a 3D viewing point from which to look at the drawing.
- Changing the drawing or the drawing portion while keeping the view angle.

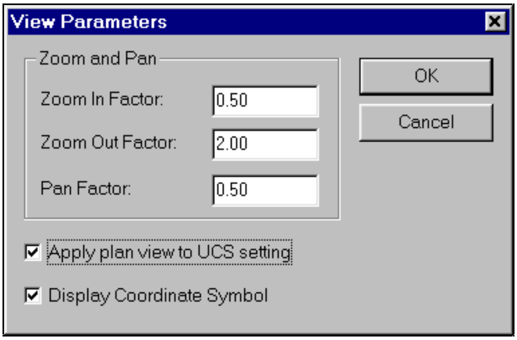
There are different ways to accomplish this, by using either:

- Commands which change the scale of the drawing and thus also enlarge or reduce the visible drawing portion, such as the **zoom** command.
- Commands which keep the scale of the drawing constant but move the drawing portion, such as the **pan** command.

View Parameters

Various parameters may be selected to define the direction and general size and shape of a given view. There are values for ZOOM and PAN of the view on the screen, and selections to be made to determine visibility of the grid and the coordinate system.

To activate the dialog box **View Parameters...** choose the item of that name from the *View* menu or type the command **VIEWPAR**.



In the edit boxes **Zoom In Factor** and **Zoom Out Factor** of the dialog box, you may choose the factor by which the drawing on the screen is to be enlarged or reduced when using the ZOOMIN or ZOOMOUT command.

In the edit box **Pan Factor** you can specify the factor by which the visible portion of the drawing is moved one of the panning commands PANUP, PANDOWN, PANLEFT, or PANRIGHT. A pan factor of 0.5, for example, moves the drawing portion to about half of the visible drawing area in a certain direction.

3D View and Drawing Viewports

The program is capable of opening as many as four drawings at the same time. Each of these drawings can be viewed in its own window. Further, it is possible to open as many as four viewing windows (viewports) for each drawing so that the drawing may be seen from different angles or so that portions of the drawing can be examined.

The different viewports of one drawing are kept in order by a running count in the title of the window. This number appears right after the name of the drawing and is separated from the name by a vertical line. These different windows can be arranged in Tiles, Cascaded, or enlarged to fill the entire drawing area. If more than one window exists, then only one window can be the active window. The active window is identified by the color of the title panel. You may pick a window to be the active window by clicking on any part of the window with the cursor.

Opening Viewing Windows

In order to open a viewing window a drawing must have been created, or an existing drawing must be open. If more than one drawing has been opened, you should remember that all commands are executed in the active drawing. Choose **Open New Viewport** on the *Window Menu* or enter **WOPEN**. Before the program opens another window you will be asked to determine the type of view in this new window. Choose a standard view from the menu, and click on OK.

The chosen angle of view can be changed at any time. The new window will be created in standard size and arranged in the manner proscribed with the existing windows. (Tiled, cascaded, etc.)

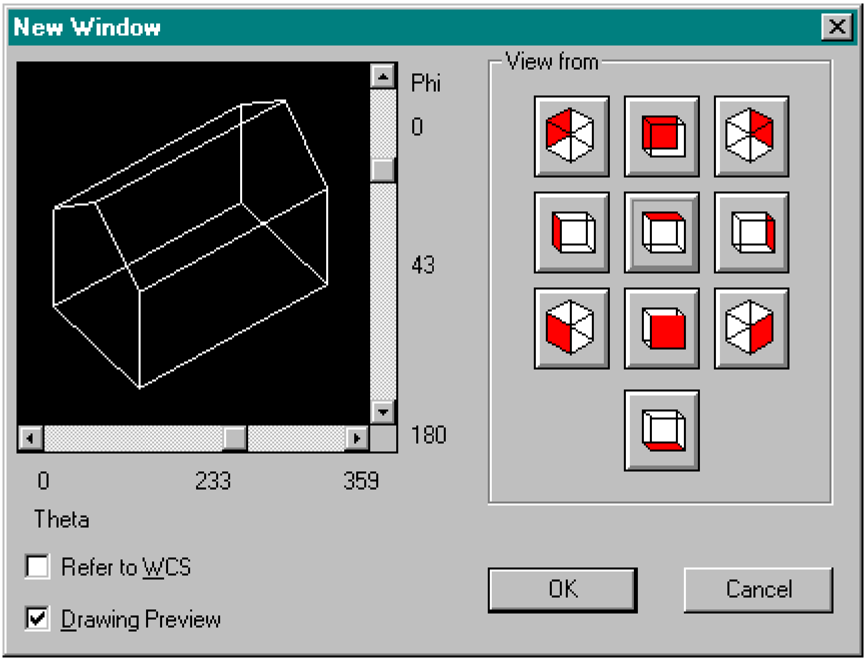
Angle of View / Point of View

The direction from which an object is viewed influences the view of the object, and may be changed in many ways inside one drawing window. Each additional view will be portrayed in another viewing window.

Standard Views

The program uses as its standard viewing angle the top view (also called Plan View). At this angle the xy-plane is viewed from the direction of the positive z-axis, and the drawing or construction plane is parallel to the screen.

Additionally, FRONT, BACK, LEFT, RIGHT, and BOTTOM are possible standard views and may be activated through the dialog box resulting from the selection of **Open New Viewport...** in the *Window* menu. Choose an option and confirm through OK.



Plan view

To quickly switch to a plan view, select **Plan View to Construction Plane** on the *View* Menu, or enter **PLANVIEW**.

Changing the current view direction

The command **3D View…** from the *View* Menu (Command **3DVIEW**) allows you to look at a drawing from any point in space that you wish to define. You must define the view direction as described above.

SETVIEWDIR

This command can be used in place of the above graphical method, by typing;

```
> SETVIEWDIR
Viewpoint (X,Y,Z): Rotation
```

The option *Rotate* allows to specify a new view direction to the drawing based on two angles entered as real numbers:

```
Angle theta (rotation angle in x-y-plane) <0.0>:
Angle phi (rotation angle to x-y-plane) <0.0>:
```

The Angle Theta is specified with respect to the X axis, in the XY-plane

The Angle Phi is specified up or down from the XY plane

The command SETVIEWDIR cannot be used in paper space.

Changing Viewing Windows

Once you have several views open, there are two ways to change the "active" window:

1. Clicking on any part of the chosen window with the cursor will activate it.
2. Open the pulldown menu ***Window***, and find where all open drawing windows are listed. The active window will be marked by a check in front of its name. When a new window is opened, it is marked by a number in front of the specific entry.

You may choose one of the listed windows as the active window by clicking on that view, or by entering the number assigned the view.

Copying a Viewport

The command **Copy Viewport** on the *Window* Menu (Command **QWOPEN**) allows you to make a quick copy of the current viewport or a plan view by choosing the desired option button.

Closing a Viewport

The command **Close Current Viewport** on the *Window* Menu (Command **WCLOSE**) allows you to close the current viewport. Or you may use the standard Windows methods of closing a window by pressing CTRL and F4 or click on the X in the upper corner of the desired window.

Zoom

The zoom commands change the viewing scale of the drawing, or of the portion of the drawing chosen by the user. The absolute ratios of scale of the entire drawing to the included objects chosen remains unchanged.

These variations on the zoom command are available:

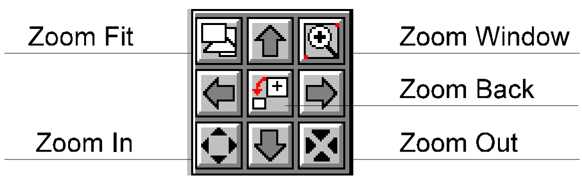
- Zoom Window
- Zoom Back
- Zoom Fit
- Zoom In
- Zoom Out
- Zoom Scale

The zoom commands, like most other commands for view control, are **transparent** commands that can be called up and executed at the same time another command is being executed. For example, it is possible to execute the command ZOOMOUT while at the same time entering points when drawing an element. The screen plot will be enlarged by a pre-determined factor. Remember that to call up transparent command you have to put an apostrophe before the command (example, 'ZOOMWIN).

```
> LINE
From point: P1
To point: P2
To point: 'ZOOMOUT
To point: P3
To point: ...
```

To Call the Zoom Commands

The zoom commands can be activated either by selection with the cursor or by entering commands by hand. The symbols used to activate the zoom variations are shown below.



To activate a zoom command, click on its symbol. The manual commands for the activation of the different zoom variations can be found in the following table.

Zoom-Variation	Command
Zoom Window	ZOOMWIN
Zoom Back	ZOOMBACK
Zoom to Fit	ZOOMFIT
Zoom In	ZOOMIN
Zoom Out	ZOOMOUT
Zoom Factor	ZOOMFAC

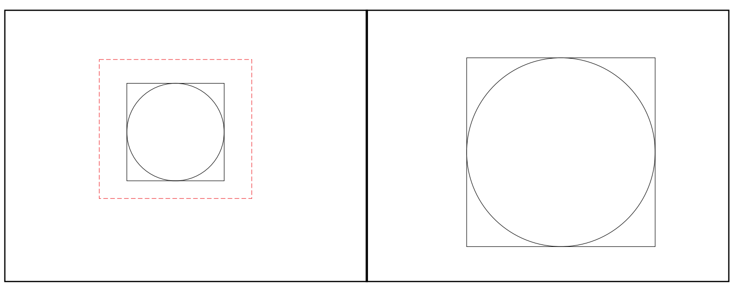
Zoom Window

The ZOOMWIN command allows a chosen portion of a drawing to be shown in the largest possible scale. The drawing portion is specified by a box. Identify opposite corner points with the cursor or enter the coordinates manually.

```
> ZOOMWIN
First corner: P1
Second corner: P2
```

Differences between the chosen window and the following depiction on the screen can be due to the different ratios of height and width of your chosen box and the screen.

The following picture shows the usage and result of the zoom command window.

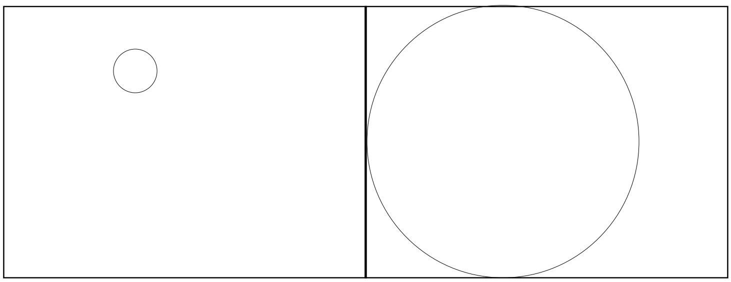


Zoom Back

The command ZOOMBACK deletes the last ZOOM command and recreates the prior drawing portion. The deleted scale is saved as the last used scale and may now be called again. Using this command allows you to switch quickly between two definite portions in order to view an object as a whole, or to see its details more clearly. It is still possible to recreate the last scale if another operation has been performed.

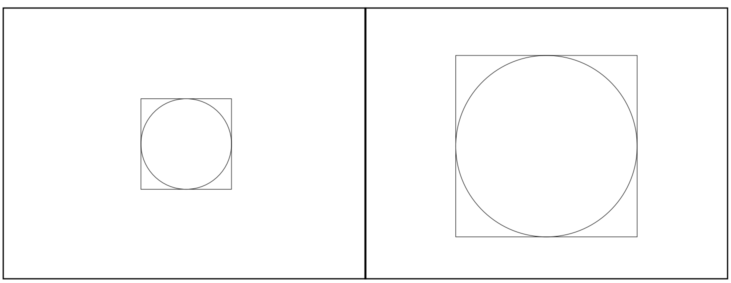
Zoom to Fit

The command ZOOMFIT allows you to show the drawing with all its elements as large as possible on the viewing area.



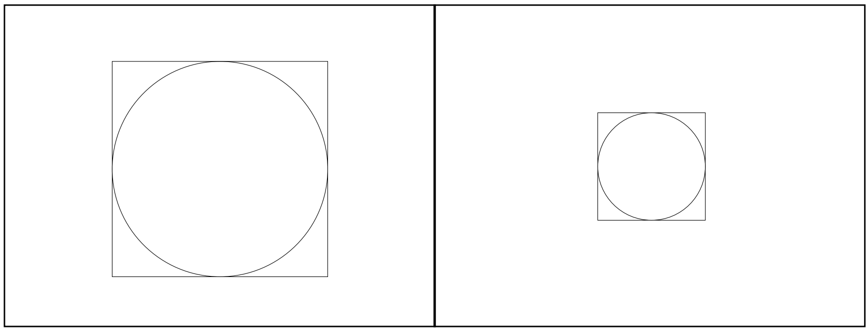
Zoom In

The ZOOMIN command enlarges the view with each successive use by a factor defined by the **View Parameters…** option on the *View* menu.



Zoom Out

The function ZOOMOUT is the opposite of the zoom-in function. It reduces the view with each successive use by a factor defined by the **View Parameters…** option on the *View* menu.



Zoom Factor

The function ZOOMFAC prompts you to enter the Zoom Factor. Numbers less than 1 reduce the size (Zoom Out) and greater than 1 enlarge the size (Zoom In)

Zoom Page

The function ZOOMPAGE zooms out to the page size defined for your printer (or plotter)

Panning

The command PAN moves the visible portion of the drawing to the right, to the left, upward or downwards. It is now easy to look at objects that were previously not visible because they were off the viewing surface. In contrast to the command ZOOM, the drawing scale does not change.

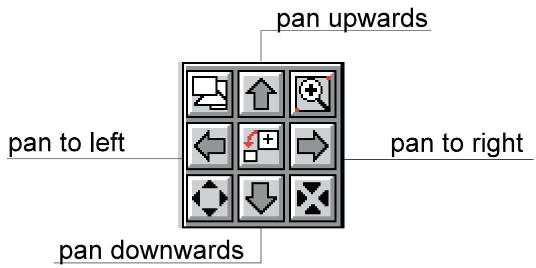
The panning commands are transparent ones which may be executed while other commands are being executed. It is possible, for instance, to execute the command PANDOWN (move downward) while entering points for drawing an element.

```
> LINE
From point: P1
To point: P2
To point: 'PANDOWN
To point: P3
To point: ...
```

You should remember that, when moving the drawing, it appears that the visible objects are moving in the direction of the command. The reason for this is that the panning commands act, as does a photographer, to move the camera, but not using the zoom lens.

How to call a Pan Command

The panning functions can be chosen through picking a symbol or by entering the short commands manually. The symbols for the different PAN versions are shown in the following picture.



To activate a panning command, simply click on its symbol.

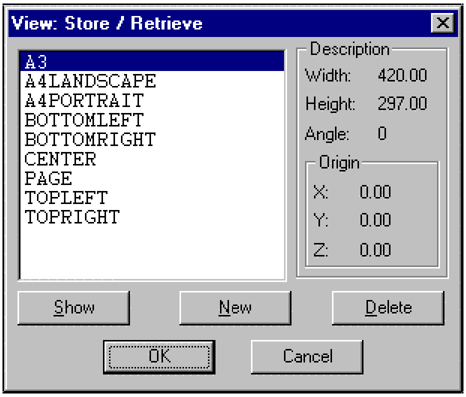
The manual commands for the activation of the different panning variations can be found in the following table. Please remember that to call up transparent commands you have to put an apostrophe before the command (for example: 'PANLEFT).

PAN Variation	Command
Portion to the left	PANLEFT
Portion to the right	PANRIGHT
Portion down	PANDOWN
Portion up	PANUP

After selecting one of these commands, the visible drawing portion will immediately be moved in that direction. The amount of movement is determined by a factor defined by the **View Parameters…** option on the *View* menu.

Saving / Retrieving Views (View Manager)

It is convenient when working with different drawing portions and view directions to save and later to retrieve a previously used view or drawing portion. Saving and retrieving user-defined views may be accomplished by choosing the option **View Manager...** from the *View* menu or by entering the command VIEW.



Saving the Current View

First, use the zoom and pan functions and/or the commands described previously to set up a desired view direction, in order to save the wanted view or portion of the drawing. Next:

- In the dialog box *Save/Retrieve View,* choose the option **New**.
- It is possible to choose between the options *View* and *Window*. When selecting the option *View*, the present portion will be saved as a user-defined view. The option *Window* allows the selection of a part of the present portion and will save this part only as a user-defined view.
- Enter the name for the view to be saved in the edit box.

During the drafting and construction process, named views can be applied especially in two other situations:

- Printing: Initially the PRINT command requests to specify an area of the drawing to be printed. The VIEW command allows you to predefine a region of the drawing which then can be referred to in the PRINT command.

- PaperSpace: When calling the command VIEW in layout mode outside the command VIEWPORT, you can create named views within paper space (for example to zoom fast to portions of the drawing sheet).

Retrieve a Named View

A previously saved view can be retrieved through the same function, and can then become the present view.

Follow these steps:

- In the dialog box *Save/Retrieve View* choose one of the saved views from the list box.
- Click onto the button **Show**. The saved view will then become the present view.

Delete a Defined View

Views that are no longer needed can be deleted as follows:

- Select one of the names of defined views in the list box of the *Save/Retrieve View* dialog box.
- Click onto the **Delete** button. The named view will be removed from the list immediately.

User Coordinate Systems

The user coordinate system can be freely defined by the user, having the world coordinate system as a basis. The following changes can be made in a user coordinate system:

The origin can be moved to any point in the world coordinate system.

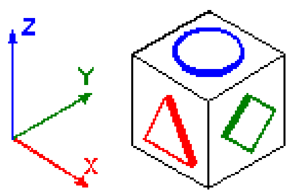
The user coordinate system can be angled or turned around one or more axis in relation to the world coordinate system.

Using the user coordinate system, for example, it is possible to change the location of the drawing plane in such a way that it always lies parallel to the objects that are to be drawn, making the construction of 3-dimensional objects much easier.

The coordinate system of this program allows you to determine every single point of a drawing surface or a drawing object unambiguously through coordinates. These coordinates originate in the coordinate system. The program uses a Cartesian coordinate system consisting of three coordinate axis, of which every pair is arranged orthogonally and which cross at the origin. All axis have the same measurements.

To determine the coordinates of a point, the distances of this point to the x- and y-axis, in a 3-D system also from the z-axis, are measured.

The program differentiates the coordinate system between a world coordinate system (WCS) and the user coordinate system (UCS).



World Coordinate System

The world coordinate system is a permanently installed Cartesian coordinate system that is used as a standard for all drawings. The x-axis measures the horizontal distance, the y-axis the vertical distance from the origin. The z-axis is orthogonal to the x- and y-plane and measures the distance of a point to this plane. The world coordinate system cannot be changed by the user.

UCS Command

It is possible to create as many user coordinate systems for one drawing as wanted. The user coordinate system will be saved along with the drawing and can be used again when opening the drawing later on.

Active coordinate system

From all the user coordinate systems in one drawing only one or the world coordinate system can be the active coordinate system, that means with its parameters active.

Choosing a coordinate system as active is done in a dialog window. Using the same dialog window it is possible to rename, create or delete already existing user coordinate systems.

Create new user coordinate system

User coordinate systems are always created in two steps. First the parameters of the user coordinate system (place of the origin and direction of the coordinate axis) are determined, second follows the saving under a specific name. Thus it is possible to use the user coordinate system as an active coordinate system again, without having to determine the parameters again.

To create a user coordinate system enter the command UCS. The prompt UCS origin <0,0,0> appears. Enter the coordinates of the origin of the user coordinate system, or press Enter to use 0,0,0.

After calling upon the command UCS, the following options for interactively determining the place of the origin of the user coordinate system and the direction of its axis are presented.

Control	World	Z-direction	3point	View	Rotation

The option control calls up a dialog window, that can be used to create or choose a new active coordinate system, or to rename or delete already existing user coordinate systems. How to use this dialog window will be shown later on in connection to the saving of the parameters of a user coordinate system.

The option world chooses the world coordinate system to be the active coordinate system. This allows to go back to the world coordinate system quickly without having to open the dialog window of the command control.

Z-Direction

The option z-direction changes the parameters of the coordinate system in the following two steps:

1. The origin is determined through entering the new coordinates in the format of x,y,z or through choosing a point .
2. The direction of the axis of the coordinate system can be determined through fixing the z-axis..

To do that, choose a point through which the positive z-axis, starting from the (perhaps newly defined) origin, should go. The x- and y-axis will not be turned and remain the same. The xy-plane will now be lifted to correspond orthogonally to the z-axis.

```
UCS origin <0,0,0>:Z-axis
UCS origin <0,0,0>:Enter
Point on the positive z-axis:P1
```

3Point

The option 3point makes it easy to adjust a coordinate system to a drawn object in space. This is advantageous for constructing elements, where the drawing plane is neither parallel to the viewing plane nor to the axis of the presently active coordinate system.

It is easiest to adjust a user coordinate system with the 3point option when the points that are to be defined are selected with the cursor using the object catch-function. This way the coordinate system is easily adjusted to the specific working situation.

1. Choose the origin first through entering the coordinates in the format x,y,z or through pointing to it.
2. After that, choose a point, through coordinate entry or through pointing to it, through which the positive x-axis should run. This way the axis is unambiguously determined.
3. The place or direction of the other axis can be determined through the fixation of a point in the positive area of the xy-plane. This point does not necessarily have to lie on the y-axis.

For practical reasons you should locate the necessary points so, that they are oriented at the plane of the drawing object at which the user coordinate system should be adjusted. For example, choose the corner points of this plane with the object catch option. The chosen plane then becomes the drawing plane. All elements drawn after that will be drawn according to this plane.

```
UCS Origin <0,0,0>: 3Point
UCS Origin <0,0,0>: P1
Point on the positive x-axis: P2
Point on the positive xy-plane: P3
```

View

This option adjusts the user coordinate system parallel to the active viewing plane, the screen Through this the viewing plane and the drawing plane are brought into harmony.

To call up this option, click onto its operation field. No further points will be asked before adjusting the user coordinate system. The x-axis of the coordinate system now lies parallel to the actual viewing plane horizontally, the y-axis is oriented vertically and the z-axis is orthogonal to that.

Rotation

Through the option rotate it is possible to rotate the active coordinate system around one axis. The angle of rotation and the direction of the rotation can be chosen randomly

To call up the option rotation, please click onto its field among the options.

Wit one call-up it is possible to rotate the coordinate system around one axis only. If you want to rotate the coordinate system around 2 or 3 axis, please repeat the following steps for each axis.

1. First determine the axis around which the coordinate system should be rotated. Either enter its character or click onto its option field (X,Y,Z), which will appear after the activation of the rotate option.
2. Now determine the direction and angle of the rotation. This is done by entering or pointing an angle.

The angle of the rotation can be determined either by entering it manually or by the pointing of 2 points. The orientation point for the angle of the rotation is always the positive x-axis of the active coordinate system.

Entering a positive value for the angle of rotation results in a rotation to the left, a negative value (for example -30) results in a rotation to the right.

```
USER COORDINATE SYSTEM origin <0,0,0>: Rotate
Rotate around x-, y-, or z-axis: Z
Rotation around Z-Axis <0>: 30
```

Control the user coordinate system

Of course it would not be very user friendly and not very effective for the precision of your work, if you had to make the above adjustments every time. Therefore FCAD offers a comfortable possibility to save and organize pre-defined user coordinate systems. The parameters of the active coordinate system are gathered in a user coordinate system and saved along with the drawing. When saving, the user coordinate system gets a definite name. Once a user coordinate system is saved, it can always be called up again to be the active coordinate system. They can also be renamed or deleted.

The necessary steps to gather and organize user coordinate systems are executed through a dialog window. To open this window please choose the option user coordinate system-manager, which is found when using the standard menu in the pull-down menu view

Also it is possible to open a dialog window by entering the command UCS and choosing the option Control.

Create a new user coordinate system

The creation of a user coordinate system is done in two steps

1. Arrange the coordinate system with the previously described options z-direction, 3point, view and/or rotate.
2. Open the dialog window through the option command and arrange a new user coordinate system. Now the parameters of the active coordinate system are saved.

For the following directions it is essential that the first step has already been taken, e.g. the coordinate system has been adjusted.

The dialog box has an option which is called user coordinate system name. Enter the name of the new user coordinate system in this field. The name can fill as many as 32 spaces and may contain alphabetical characters, numbers, as well as the fill-in characters - (dash) and _ (underline), but no free spaces. It is not differentiated between capital letters and small letters, all characters will be changed to capital letters.

After entering the name click onto the field new and verify your entry. The name of the new user coordinate system appears in the window below the entry- and change window. All useable coordinate systems in this drawing are listed there.

Please remember, that all changes made in the dialog window user coordinate system are registered only when you leave the dialog window through clicking OK.

Renaming a user coordinate system

It may be necessary to rename user coordinate systems in order to have a clear name or association

1. Choose the user coordinate system to be renamed out of the list in the choice window by clicking onto it with the cursor. The chosen coordinate system is accentuated through color. At the same time the name of the chosen user coordinate system appears also in the entry- and change window.
2. Click onto the window with the cursor or enter it by repeatedly pressing the tabulator key, and then change the name of the user coordinate system into its new version.
3. After that, click the field rename. This verifies the renaming and the new name of the user coordinate system appears in the choice window.

Please remember that the world coordinate system cannot be renamed.

Please remember, that all changes made in the dialog window user coordinate system are registered only when you leave the dialog window through clicking OK.

Erasing a user coordinate system

Erasing a user coordinate system is similar to renaming a user coordinate system

1. Choose the user coordinate system to be deleted in the choice window by clicking onto it with the cursor and marking it so.
2. Click onto the field delete. The user coordinate system is then deleted, its name no longer appears in the choice window.

Please remember, that the world coordinate system cannot be deleted.

Changing the active coordinate system

In every drawing it is possible to create as many user coordinate systems as you wish next to the world coordinate system, but only one user coordinate system can be the active coordinate system.

The active coordinate system is that coordinate system, whose parameters are active (e.g. place of the origin, adjustment of the axis).

1. Choose the user coordinate system to be active in the choice window by clicking onto it with the cursor and thus marking it. The name of the chosen coordinate system appears at the same time in the entry- and change window.
2. Click onto the field active. The user coordinate system thus becomes the active coordinate system.
3. The field active user coordinate system shows the name of the active coordinate system. Please check there, that the name of your chosen coordinate system appears there.

Change to the previous coordinate system

To change to a previous coordinate system there exits an option in the dialog window user coordinate system. It enables to quickly change between two coordinate systems. When clicking onto this field you will change to the previous coordinate system without another entry or verification.

Chapter 4

Layer and Object Properties

The concept of working with layers is analogous to manual drawing using layers of see-through foil. Parts or portions (which may be functional portions, such as construction lines) of one complete drawing are drawn on different “foils”, which, imposed on top of one another, form the entire drawing.

Layers are useful in separating objects, such as parts, sub-assemblies, or structural components such as pipe sizes or building materials. They also serve to separate elements such as construction lines, center lines, outlines, and dimension lines, or things that are to be placed in a group, such as foliage in an architectural drawing, or parking areas. The advantages of creating a drawing in this fashion, whether working manually or electronically, lie in a clearer construction of the drawing and better control than drawing on a single foil.

This chapter describes how to create and use layers, as well as different ways of controlling them. *Control* refers to the visibility of the layers, their colors and the types of lines used. In the second part of this chapter you will find explanations on object characteristics and the various advantages of working with them. Objects may be parts, sub-assemblies, groups of like-attribute entities (such as, again, parking spaces in an architectural drawing), or any other entity that you wish to separate from other similar entities in the drawing for particular treatment.

In the layer technique a new object will always be placed on the current layer. At the same time the standard properties linetype and color of a layer definition will be applied to it. Additional commands allow you to change the object properties later on.

Layer: Basic Information

Instead of working with foils as one might do manually, computer programs for drawing construction work with layers, which have the same characteristics as do foils . It is possible to structure a drawing through formal or content characteristics.

Layers are drawing layers that lie exactly on top of one another. All parameters of a drawing which are active for the entire drawing, like the coordinate system, the Zoom factor, the view arrangement, fonts, attributes, and so forth, are valid for all layers and are always the same.

Different layers do not necessarily follow one another in numerical order. It is not necessarily true that layer 2 lies “on top of” layer 1 or “underneath” layer 3. All layers are arranged in one single plane, which is not possible when drawing manually on different foils.

For elements that “hide” other elements on different layers, the rule is that the object last drawn lies over already existing objects and hides them. The association of one particular drawing object to one particular layer in some particular order is not important

The standard layer is always called **Layer 0**. It is generated automatically by the program when a new drawing is created. The layer color 7 (black) as well as the linetype continuous is automatically established for this layer. This layer cannot be erased or renamed. It is possible to create up to 1024 layers, a number which should be more than sufficient for any practical drawing.

Layer 0 has a special function when defining or inserting parts. You can find more on that subject in the chapter on *Parts and Attributes*).

One layer in each drawing will be defined as the **current layer**. All new drawings will automatically be placed on this layer. As long as Layer 0 is the only one existing, it will also be the current layer. If there is more than one layer in a drawing, as there should be, it is possible to switch current layers using Layer Control.

The drawing commands work on the current layer. You will always be drawing on the current layer. However, it is possible to edit any elements in the drawing, even those not in the current layer, using the Edit or Modify commands, if the layer has not been **frozen**, **turned off** or **locked**.

Each layer has a color and linetype associated with it when it is created. Anything drawn on this layer will be drawn with this linetype and color. It is also possible to associate a different color and linetype by editing the layer.

Aside from the color and the linetype, there are other attributes of the layer that allow you switch between "states". The states are **visibility** (on/off), **freeze/thaw** and **lock/unlock**.

The layers with their status and their attributes are part of the drawing. They are saved together with the drawing and can be called when re-opening the drawing.

Layer Attributes

Layer Name

To identify individual layers and differentiate between them, each must have a distinctive name. The name can consist of up to 31 characters. They can be letters, numbers, or special characters like _ (Underline) and $ (Dollar). All characters are automatically capitalized.

Color

Every layer has a color that is user-defined. If a new color is not chosen, the layer will be colored black. The standard color of the layer can be marked in the color box as BYLAYER.

It is possible to choose the standard color of a layer from among 255 color shades. Every color has its own number.

Since it is possible to draw on the current layer only, objects still to be drawn will be in the standard color of the current layer, unless a new color was chosen with the option Object Color. If the standard color of the layer is changed using the dialog box for layer control then all drawing elements of this layer that are marked with the color BYLAYER will assume the new color.

Linetype

Similarly, there is a standard linetype that is assigned to the line. If you do not choose another linetype, the linetype CONTINUOUS will automatically be assigned to it.

The chosen linetype is marked in the dialog box BYLAYER. All elements are drawn in the linetype of the layer unless another linetype is chosen for the element.

If the linetype of a layer is changed using the dialog box layer control, all of the drawing elements of that layer which are marked with BYLAYER will assume the new linetype.

The linetypes that are available are shown in a choice window. They are made available through a linetype definition file. Please remember that only those linetypes can be shown which have previously been loaded.

How to Load Linetypes

The linetypes that are used to draw objects depends largely on the industry for which the drawing is being prepared. The program, therefore, does not work with a fixed set of linetypes, but allows you to load linetype files for working in particular disciplines.

Linetype files have the extension **.lin** and can be found in the application directory.

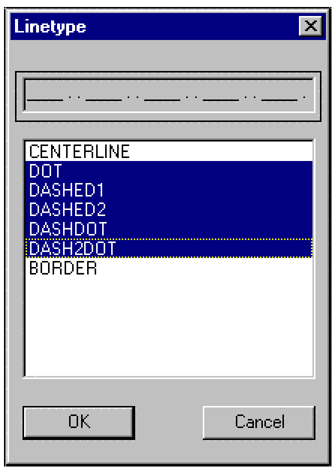
To load a linetype file, select from **Linetypes ...**, in the menu *Options*, or type the command LINETYPE.

When this command is selected, the dialog box *Open Linetype File* will be opened.

Enter the name of the linetype file in the input field *Filename* or choose a file from the file list. In the *file format* field the format **.lin** will appear.

Next, choose one or more linetypes from the list in the dialog box *Linetype*.

The preview/description area at top of the dialog box will assist you to find an appropriate linetype.



After you choose and load the linetype(s), they may be associated with a layer by using the Layer Manager, or they may be associated with already existing drawing objects by the command PROPEDIT.

Layer Visibility

One of the major advantages of working with layers is that it aids you in visualizing the part, or drawing. The visual clarity of a drawing can be improved greatly through choosing layers properly and “hiding” the drawing elements. Concentration on meaningful drawing elements makes the drawing and construction much easier.

Visibility of a layer can either be turned on or off. Every layer can be turned “on” or “off” independent of any other layer(s). Changing the visibility is done through the layer manager (command LAYER).

Locked / Unlocked Layers

It is possible to “lock” layers so that the objects or elements contained within the layer may not be changed. Normally, new layers will be created with the status **UNLOCKED**. There are no limits inside the normal program functions when working with an unlocked layer.

When a layer’s status is **LOCKED** it is not possible to change, add, or delete objects from this layer. The status must be changed to **UNLOCKED** if you desire to work with the objects of this layers. This function makes it difficult to make an unauthorized drawing change.

It is possible to draw new elements onto locked layers but it is not possible to work with them further. Other options of layer control can also be affected when working with a locked layer. For example, the standard color of the layer may be changed. A locked layer may be designated active, visible or invisible, frozen or thawed.

Thaw / Freeze Layers

The options **Thaw** and Freeze are functions which control the visibility as well as the influence of the objects within the layer .

A frozen layer is both invisible and locked and thus impossible to change. A frozen layer will not be included in drawing regeneration. When working with very complex drawings, freezing some unneeded layers can increase the speed of drawing considerably.

The option **Thaw** undoes the freezing of a layer. The layer will become visible and changes may be made once more.

Note:
The current layer of a drawing cannot be frozen.

Layer Manager

Creating new layers, choosing an current layer and controlling all previously described layer characteristics is accomplished through a dialog box. This dialog box is called **Layer Manager** .

The control panel (side bar) contains control fields for the following layer functions:

- Changing layers;
- Choosing a drawing color other than the one selected in the layer definition; and
- Choosing a linetype other than the one selected in the layer definition.

The Dialog Box used in layer control is made available by choosing the *View* menu, then clicking on **Layer Manager...**, or by entering the command LAYER.

Dialog Control Elements

The first section of the **Layer Manager** dialog box contains a field for the name of the active (current) layer and a field for the number of defined layers in this drawing.

Later fields contain combined view and entry fields for layers that still have to be worked on, and a selection box that allows you to choose between creation of new layers, renaming layers and choosing an current layer.

There are fields for the layer state and for choosing line types and layer color.

Directly beneath that, in the Dialog Box, all the existing layers of the drawing are shown. Any of these may be selected and activated with the cursor. The layer color and linetype are also shown there.

The meanings of the abbreviations used are shown here.

Abbreviation	Meaning
On	Visibility on - The layer becomes visible
Off	Visibility off - The layer becomes invisible
Unlk	Status unlocked, drawing elements can be edited
Lock	Status locked, drawing elements can not be edited
Thw	Layer thawed, visible and can be edited
Frz	Layer frozen, invisible and can not be edited, will not be regenerated

Creating a New Layer

A new layer can be created at any time. Newly created do not have to have drawing elements right away.

The layers of a drawing are not listed alphabetically or with respect to the content of the layer, but according to the time of the layer's creation. To work effectively it makes sense to plan the layers to be used in advance and create them in a systematic manner.

A new layer is created by performing two steps:

1. Enter the name for the new layer in the entry- and view field for layer adaptation (second row). Remember the limits of naming (see page 96). After that, select the button *New*. The layer will appear in the list following the layer last created.
2. Every new layer has a color 7 (black, if no other is chosen) and the linetype (CONTINUOUS, if no other is chosen) assigned to it. Other linetypes and colors may be assigned through selection.

Renaming a Layer

1. Choose the layer to be renamed from the list of layers;
2. The name appears now in the entry window. Write the new name over the old one;
3. Finish renaming by clicking on *Rename*;
4. The renamed layer will be listed under its new name in the dialog box.

Changing the Current Layer

1. Choose the layer which you want to become the current layer from the list in the dialog box. The name of the chosen layer will appear in the **Current Layer** window;
2. Verify the choice of the new current layer by clicking onto the button *Active*.

Check the name of the current layer in the dialog box. If the layer change has been performed correctly, the name of the chosen layer should appear there.

Define Layer Linetype

A newly created layer will automatically have the linetype CONTINUOUS assigned to it. A new linetype can be defined with the button *Linetype*.

1. Click on Linetype;
2. The available linetypes are shown in the window that appears. Select the linetype to be used; and
3. Verify your choice by clicking on OK.

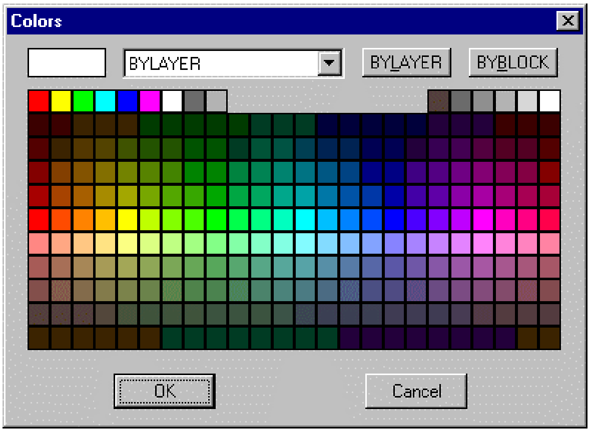
This will make the selected linetype the standard linetype of this layer. All drawing elements will be drawn with this linetype, which now becomes the linetype BYLAYER.

When the linetype is changed in this manner all elements of this layer that were drawn with the linetype that was previously standard will now be drawn with the new standard linetype.

Elements of the layer that were drawn with a linetype different than the BYLAYER linetype will **not** be affected by the change. Further instruction on drawing elements with a linetype different than the standard linetype can be found in the section *Object Properties* in this chapter.

Define Layer Color

When a new layer is created it will automatically be assigned the layer color 7 (black). A new color can be assigned to the new layer via the click-on field COLOR.



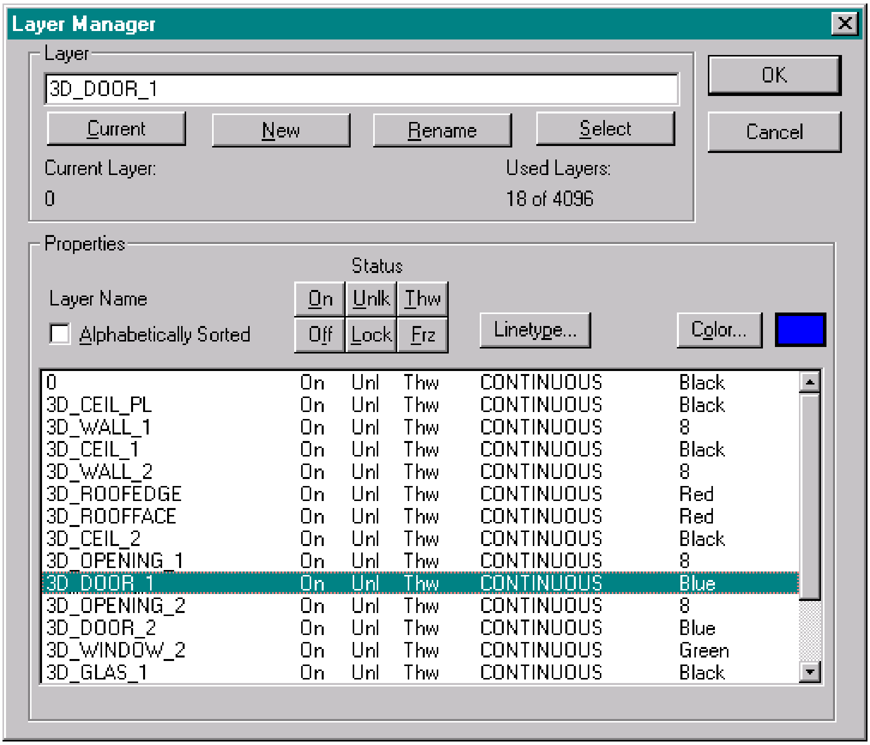
1. Click on **Color...** . The window shown above will open;
2. Choose a new standard color from among the basic colors or enter a color number (1-255) into the entry window on the right. The chosen color will appear in the window on the bottom;
3. Verify your choice by clicking OK.

This makes the chosen color the “standard” color of this layer. All elements of this layer that were drawn with the standard color will be drawn with this color, which is also called color BYLAYER.

The elements of the layer that were drawn with a color different than the standard color will not be affected by the change.

Controlling the Layer Status

The layer characteristics that comprise the *state*, or the status, of the layer have been discussed earlier in this chapter. Changing the status of a layer is accomplished by a clicking on the proper field. Characteristics of each layer are displayed in this portion of the Layer Manager dialog box.



The layer status ON-OFF (controls the display visibility of the layer), UNLK-LOCK (controls access to the layer, UNLK allowing editing, LOCK prohibiting editing), and THW-FRZ (controls ability to see and to edit or change the layer, THW unlocking and making visible, FRZ locking and making invisible) may be changed in two steps:

1. Mark the layer to be changed in the Dialog Box by selection with the cursor. The status changes will be executed for the marked layer.
2. Select the field required for the status change(s).

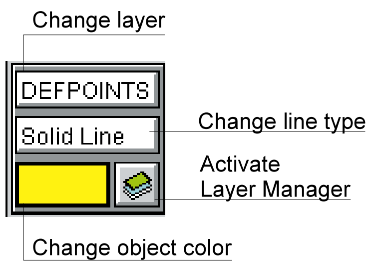
Remember that the current layer **cannot** be frozen.

Defining Layer Properties Further

FelixCAD contains three functions other than the Layer Manager that can be activated through selection. These functions are used to:

- Change the current layer;
- Draw with a different linetype than the linetype BYLAYER; and
- Draw with a different color than the color BYLAYER.

To call on these functions, select the buttons that are found on the palette on the left-hand bar of the desktop.

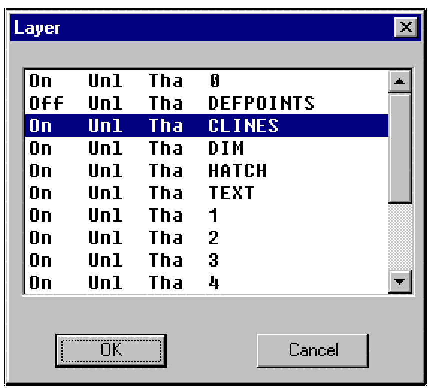


Set Current Layer

This function allows you to set a new current layer.

Note:
It is not possible to create a new layer or to change any characteristics of the existing layer with this button. These changes must be made in the fashion described above in the section *Layer Control*.

Selecting these buttons will open the appropriate dialog box, and the proper selection can be made.



The letters in front of the layer names show the status of the characteristics visibility, access, and thawing/freezing. They have the following meaning:

Abbreviation	Meaning
On	On (Visibility)
Off	OFF (Visibility)
Unl	Unlocked (Access)
Loc	Locked (Access)
Tha	Thawed (Thawing/Freezing)

A frozen layer cannot become the current layer, so that choice will not appear for the current layer.

Select a layer in the list box and verify your choice by clicking on OK.

Set Current Linetype

For reasons of clarity, you may want to draw different objects with different linetypes than the one marked BYLAYER.

Remember that a new linetype has to be chosen before drawing a new object. To change the linetype of an object afterwards use the command PROPEDIT (Property Edit) described later on in this chapter.

To change the current linetype, select the button **Linetype...** in the control panel. The dialog box that opens is the same as that known from the ***Layer Manager*** dialog box (see above).

1. Select the button *Linetype* in the control panel (side bar);
2. Choose a linetype from the list and verify the choice by clicking on OK.

Set Current Color

For reasons of clarity, you may want to draw different objects with different colors than the one marked BYLAYER.

Please remember to choose the color of the object before drawing it. To change the color after drawing the object, use the command PROPEDIT. More specific instruction will appear on that subject later on in this chapter.

To change the color, select the function **Color...**.

1. To open the *Set Color* dialog box click on the button;

2. Choose a new standard color from among the basic colors, or enter a color number (1-255) in the entry window on the right. The chosen color will appear in the view window on the bottom;
3. Verify the choice by clicking on OK.

Set Layer by Reference

The QLAYER utility provides an alternative way of layer control which is, in some situations, more appropriate and direct. You can set, turn off, freeze, lock, or unlock a layer by picking a reference entity. Select the function **Set Layer by Reference** from the menu *Options* or type in QLAYER. Choose one of the options from the options bar:

Off	Lock	Freeze	Unlock	On	Thaw

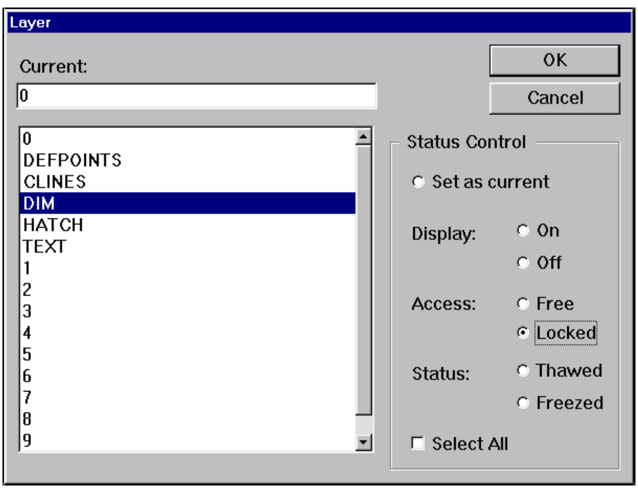
The option **Set** is the standard option for QLAYER; and is used if no other option has been selected.

For example, if you choose the option **Set**, the routine will prompt:

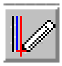
```
Select reference entity to set layer:
```

The layer on which the picked entity is located will now be the current one.

The command also provides options to turn disabled layers on or to thaw frozen layers in a dialog box. All layers with the corresponding mode are highlighted in a list box. When you choose the OK button all marked layers in the list box will be turned on or frozen.



Modify Object Properties



These characteristics are said to be “properties” of an object:

- objects may be placed on a different layer;
- objects may have a different color;
- objects may be drawn with a different linetype; and
- objects may be drawn with a thickness (extrusion).

These characteristics are associated with a drawing automatically when it is created. This association is made through the parameters called BYLAYER which are determined for the layer when it is defined. Alternatively, you may choose (in the manner just described) a special color or linetype for an object.

It may be desirable, in certain circumstances, to change these parameters once into the drawing or layer. In order to do this, the proper command is PROPEDIT.

The command PROPEDIT allows you to make these changes:

- objects may be placed on a different layer;
- objects may be given a different color;
- objects may be given a different linetype; and
- the thickness of objects may be modified.

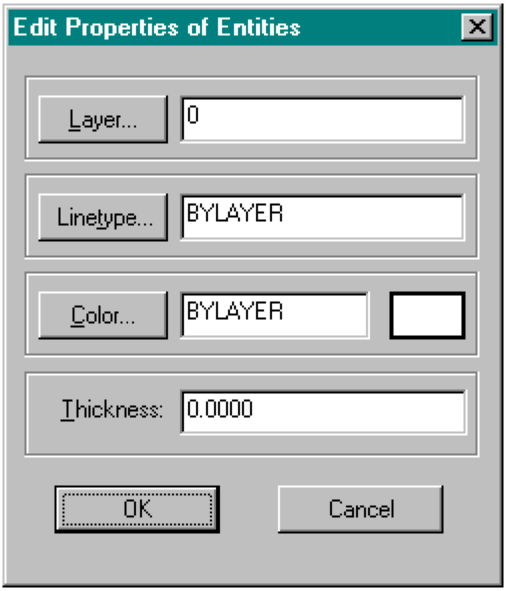
You may change any one, or all, of these properties at once.

The command to change object properties is called up by entering the command PROPEDIT or by selecting the option **Modify Object Properties...** from the *Edit* menu.

Mark the drawing elements whose properties are to be changed. Remember that all marked objects will be changed. If three objects are marked, they will all receive the same change. If different objects are to be given different changes, each must be specified, in turn, in the same manner, and the changes made to each separately.

Options

When the object selection has been completed, the following dialog box opens. Choose the object properties that are to be changed by clicking the proper button.



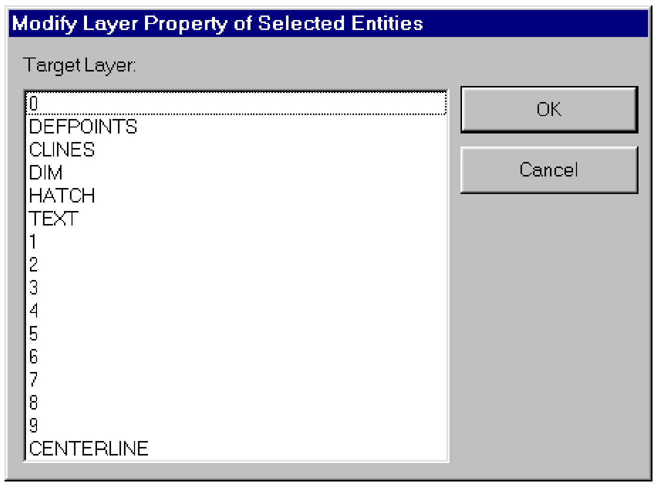
The characteristics color, layer and linetype each have a separate dialog box. These dialog boxes are identical to those opened for layer linetype and color control. Choose the desired new characteristic in each case. Thickness is applied directly and does not require a dialogue box.

Modify Object Layer

The command PROPLAYER transfers selected objects from their current layer to a chosen layer.

Call the command entering PROPLAYER or by selecting **Modify Object Layer...** from the *Edit* menu.

Select the drawing entity to be transferred. The following dialog box appears. Choose a target layer and confirm the selection by OK.



Copy Object Properties

The command COPYPROP allows you to change the properties of objects to be selected to those attached to a specified reference entity. Only properties that are common to all objects (layer, linetype, color) are transferred to the selected objects.

Use this command to make changes to the properties (the line color, layer, etc.) of selected objects. Call the command by entering COPYPROP or select **Copy Object Properties** from the *Edit* menu. To change the properties of the selected objects:

- Select a reference entity
- Select one or more objects to be changed.

Thickness

By applying the Thickness property to drawing entities, you can create a three-dimensional effect of certain entities. Such extrusion of entities creates the illusion of depth.

Thickness is displayed with the following objects:

- Line
- Arc
- Circle
- 2D-Polylines
- 2DFace

The Thickness property is also displayed, if the entities listed above are drawn with a dash-dot pattern or if these are wide polylines.

As commands like NGON, RECTANGLE, ELLIPSE, RING, or CHAIN create 2D-polylines, Thickness setting is applied them.

Thickness can also be assigned to other drawing objects (3D-polylines, 3D-faces, text, attributes, attribute-definitions, block insertions, dimensions, hatchings, ...), but the program will not display the extrusion of those objects.

To set a current preference for subsequent drawing, set the **THICKNESS** system variable. Type in the system variable name and enter a numeric value. To turn off applying Thickness for further drawing, type in THICKNESS and set the value to zero.

To alter the Thickness of drawing object, use the **PROPEDIT** command (see above in this chapter)

Applying Object Selection and Object Snap on extruded entities

When selecting entities which are displayed with extrusion thickness or when snapping to them, keep these two rules in mind:

- Object Selection: When you are requested to select objects, the program will recognize picking extrusion lines as if the base entity has been selected.
- Object Snap: When you snap to extrusion line of entities that have a thickness, the program will find the appropriate point of the base entity.

Treatment of extruded objects in Rendering and Hidden Line Removal

Objects that have a thickness are recognized in the Render and Hidden Line Removal commands (ARENDER, RENDER, HIDE, FHIDE).

Chapter 5

Draw: Drawing Basic Entities and Objects

The commands discussed in this chapter are used to draw basic geometric elements. These entities may be further defined, later, using the *Edit* and *Modify* commands. Most of the following functions are available from the menu *Draw*.

Line
Circle
Arc
Ngon
Rectangle
Trapezoid
Ellipse
Chain
Polyline
Pcontour
Filled Faces
Rings
Construction Points
3D-Faces
3D-Polyline

Options for each command will be shown in the context bar. The program will usually be performed using the first option (the option of the left-most button) as the default option.

You will be prompted for input such as the starting point of a line, length of a line, center point of a circle, points to define an arc, and so forth. The points or distances required can be entered either by pointing with the cursor, or by keyboard entry as discreet coordinates or values.

Object Snap

When defining points by pointing with the cursor, you are advised to use object snap options in order to gain the precision provided by these functions. If more information is required about object snap, you should refer to Chapter 2, *Drawing With Precision*.

Linetypes

Linetype... allows to choose a linetype definition that may be used for subsequent entity creation in the current drawing. Linetypes that were loaded within the active drawing are available in the dialog box to set linetypes. The linetype chosen with this option will **not** become the active linetype. It will only be added to the list of linetypes that are available.

To add a linetype to the linetype table of the current drawing, proceed in this manner:

- in the menu *Options* select **Line Types** (or enter command LINETYPE);
- the dialog box *Open Linetype File* will open;
- the file filter has already been set to be **.lin**. Usually linetype files are found in the application directory.
- The dialog box *Linetype* will open:
- Choose one (or more) linetypes to be loaded.
- Repeat this procedure to add other linetypes to a drawing.

Linetypes that were loaded within a drawing will be saved along with the drawing so as to be available at a later time.

Fonts...

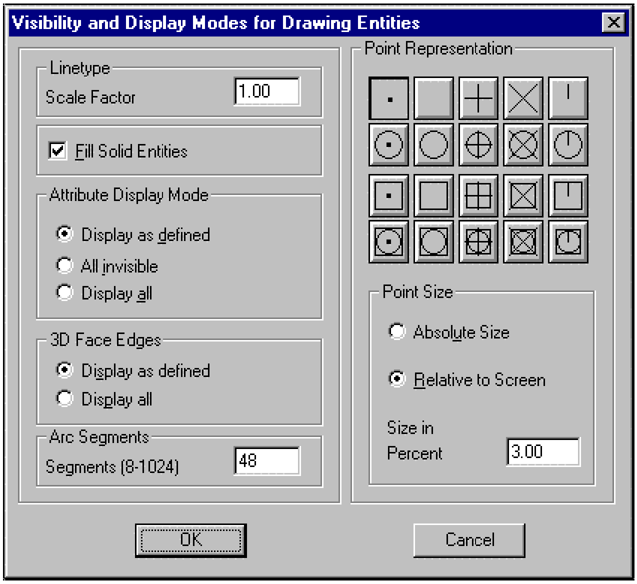
The **Fonts...** option from the *Detail* menu allows you to load text styles from font files. It possible to adapt existing fonts for special drawing needs, and use them repeatedly, with this function. There are more detained explanations of this function in the chapter *Text Objects*.

Dimension Type...

The option **Dimension Type...** allows you to choose the type of dimensioning to be used. More detailed explanations of this function are found in the chapter *Dimensioning*.

Drawing Modes...

 **Drawing Modes...** (command DRAWMODE) provides a set of parameters that are helpful in the proper visibility and display control of drawing entities. This dialog box will appear:



Parameters may be activated, inactivated, or changed.

Changes in parameters will affect only the objects drawn later. If further changes are required, they must be made again. The changes will not affect all drawn objects until the active view has been regenerated, using the **Regenerate** command on the Window menu.

Linetype Scale Factor

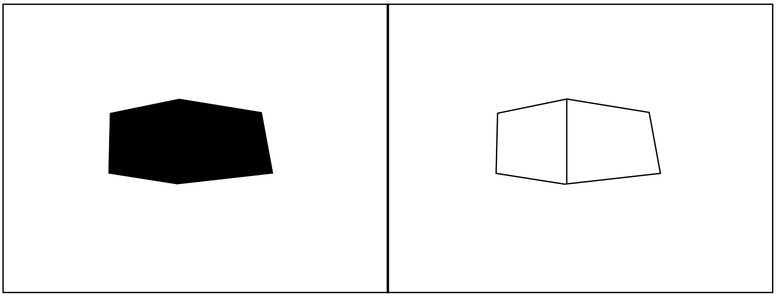
The *Linetype Scale Factor* (LTSCALE) determines a scaling factor for lines drawn with a the linetype. The line patterns are shown extended (long dash lines, for instance) or contracted (short dash lines, for instance) according to this scale. To change the scale factor, enter a new number in the edit box.

Remember that, after a regeneration of the drawing, the scales of line patterns that have been drawn already will be adjusted to the new value.

Fill Solid Entities

The *Fill Solid Entities* option specifies whether or not to fill polygons. Polygons are polylines with a width greater than zero or 2D-faces. A check mark in the box indicates whether or not the fill function is activated.

Remember that activating the function is required in order to draw filled planes. If the option *Fill Solid Entities* is deactivated, then the option *Fill Polygons* will not be available in the dialog box *Print/ Plot* (details in chapter *Print / Plot*).The examples shown below are illustrative of the affect of filling or not filling a plane or polygon.



Option **Fill Solid Entities** activated (left) and deactivated (right)

Attribute Display Modes

This option allows you to choose the desired display mode of attributes. The options are:

Display as defined	All attributes are displayed according to their definition.
All invisible	All attributes are made invisible, e.g. their status is defined as invisible.
Display all	All attributes are displayed, independently of how they were defined.

Choose a display option by clicking its field.

3D-Edge Visibility Mode

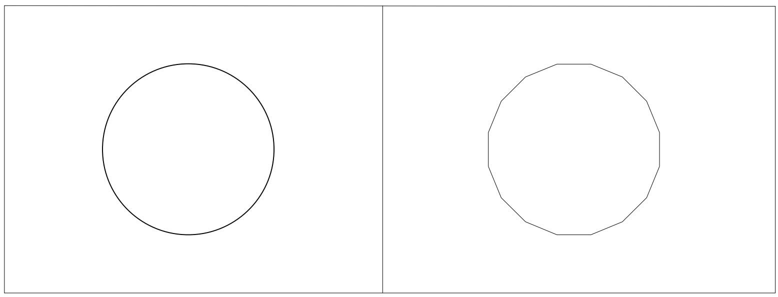
If, in the display of 3D views, intersections or lines that are “behind” other drawing segments are to be hidden, you should select the option **Displayed as defined**. If, on the other hand, all intersections are to be displayed, then the selection should be **Display all**.

Arc Segmentation

Arcs and circles are displayed on the screen as a series of short straight-line segments.

In this edit box specify the number of segments that any arc or circle that is drawn will appear. The number selected may be any whole number between 8 and 1024. A smaller number will make the arcs or circles appear “jagged”; a larger number will tend to smooth them out. The default number used is 48.

Note, that a large number of line segments in an arc or circle will tend to a slow the performance, whereas a small number of segments will increase working speed. Some Printers / Plotters are affected by this number for their output quality.



Display of a circle with 48 (left) and 18 (right) straight line segments

Point Representation

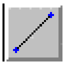
This area of the dialog box allows you to choose between 20 different display forms for construction points. The point mode will be actualized when the drawing is regenerated.

Point Size

This area of the dialog box allows you to choose the size point entities. This parameter defaults to *Absolute*, and uses 3 drawing units to display construction points.

Absolute Size	The size of points is determined in the absolute scale of drawing units.
Relative to Screen	The display of points is determined relative to the size of the visible drawing plane.

Line

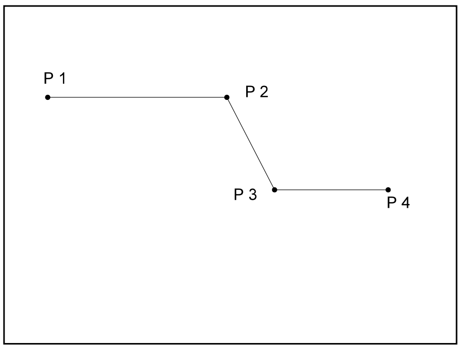
 The command LINE is used to draw lines or line segments. Inputs to the command are two points; the starting point and the ending point of the line. The command will immediately display in the option bar four choices:

Continuous	Segments	Append	TT		

Continuous

Continuous is the standard option when drawing lines, will automatically be used when no other option is chosen. It allows you to draw line segments in sequence until the command is terminated.

The command will request a start-point for the line, and will continue to request points until it is terminated. Each subsequent point will be connected to the previous point by a line.




```
From Point: P1
```
(points are defined by the coordinate system chosen, with the first coordinate the X coordinate, then the Y coordinate The coordinates are separated by a comma, thus:)
```
To Point: 2.5,2.5
```
(+ 2.5 units in the X direction, and 2.5 units in the Y direction).
```
To Point: P2
To Point: P3
To Point: P4
```

When continuous line mode is chosen, the options **Undo** and **Close** are available in the option bar.

The option *Undo* can be used repeatedly to erase the last drawn line segment.

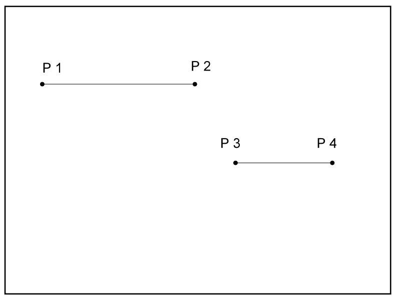
The option *Close* connects the last drawn point with the start point of in the Line Continuous command.

To terminate the command, press ENTER or ESC.

Segments

The option *Segments* allows the creation of single line segments, as opposed to continuous lines.

As with continuous lines, determine the start- and end-point of the first segment. The program will then continue to ask for start- and end-points. The difference is that the lines drawn usually have individual start points and are not appended to end point of the previous line segments. To end the drawing of line segments, press ENTER or ESC.




```
From Point: P1
To Point: P2
From Point: P3
To Point: P4
From Point: <hit Enter key>
```

Append

The option *Append* allows the connection of a line to either the start or end point of another line or arc. Choose the line or arc that is to be connected to the line still to be drawn.

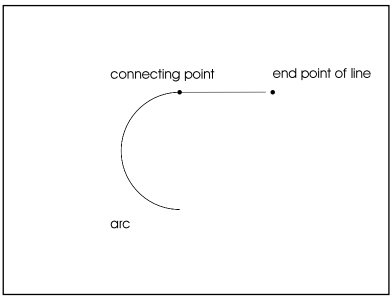
Line to Line

Select an element by choosing an already existing line or arc. Remember that the new line will be connected to that point of the line or arc that lies closest to the chosen point.

When appending a line to a line, select the line to which the appended line will connect. The point (start or end) of that line closest to the selected point on the line will be the start point for the new line.

Line to Arc

To append a line to an arc, select the arc to which the line is to be appended. The prompt will ask for the length of the line. Either keyboard the length, or pick the length with the cursor. For precision, it is better to keyboard the length. The appended line will be connected to the end point of the arc as a **tangent**.




```
Select line or arc: P1
To point: P2
To point: <hit Enter key>
```

The following options will be available in the context bar:

Undo	Close				

The options *Undo* and *Close* are available at the context bar when drawing a line, chain, 2D or 3D polyline.

Undo

The option *Undo* allows you to undo the last step taken. The end point of the last line segment may then used as a starting point of another segment if required. Undo's can be repeated as long as a segment exists. The last segment will be erased with each successive Undo.

The option *Undo* allows you to undo the last step taken. In other words, the last-drawn segment will be erased. The end point of the previous segment is then used as a starting point of the next segment. Undo's can be repeated as long as a single segment exists.

Close

This option allows you to draw a closed line contours or closed polygons. Choose the option *Close* from the context bar or enter the option manually. The two end points of the line contour will then be connected to each other.

The option *Undo* allows you to undo (erase) the last drawn segment.

The option *Close* allows you to create closed 3D polylines. The option will cause the two remaining end points to be joined together and terminates the command.

The end point of the last remaining segment will act as the starting point for the next segment. The *Undo* function can be repeated as many times as necessary until the Undo List is empty.

TT

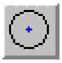
Tangent Tangent option. Allows you to draw a line tangent to two arcs or circles. Useful in creating compound or reverse curves with a line segment in between the arcs. You will be presented with 2 prompts as follows;

Tangent from:

Tangent to:

After picking the two arcs or circles the line command will continue as noted above.

Circle

 There are several ways, geometrically, to describe a circle. The program allows you to choose from among these ways, and draw the circle in the most convenient manner. A circle can be drawn by choosing of one of the following options:

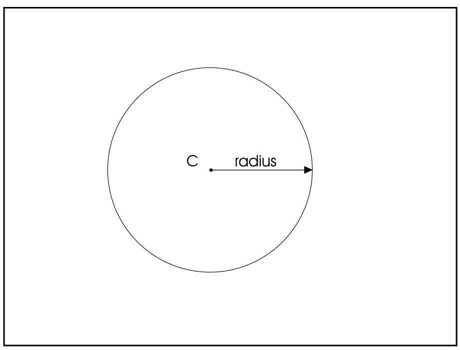
- Center of the circle and radius or diameter;
- Two Points on the circumference; or
- Three Points on the circumference.
- Two Tangents

The following options will appear on the option bar when calling the command CIRCLE:

Center	3P	2P	Radius	Diameter	Tangent

Center (Center Point)

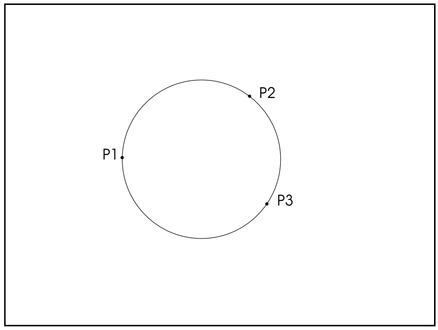
Since this option is the most often chosen, it has been selected as the "standard" option and is activated automatically if no other option is chosen. The first prompt for this option will be the location of the center point; the second will ask for the radius. If the circle being drawn is preceded by another circle, the last radius will appear at the prompt as an option. To verify the default radius value, just press ENTER; to change the option, enter or point a new radius.



If it is more convenient to enter the diameter than the radius, you may choose that option from the context bar.

3P (Three Points on the Circumference)

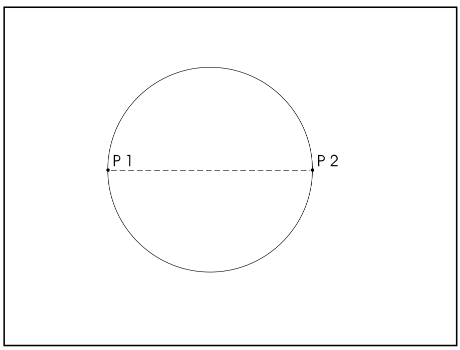
This option allows you to draw a circle by choosing three points on the circumference of the circle. It is convenient to use when the circle must intersect three tangent lines, or when drawing the circle within a polygon.



The prompt will ask for three points successively. You may point to these points on the screen, perhaps using the object snap options, or enter the coordinates through the keyboard.

2P (Two Points on the Circumference)

The prompt will ask for two points in succession. Again, the points may be picked using the object snap functions, or may be entered from the keyboard.



Radius

This geometry is particularly useful when the radius of the circle is a known, and the center point is undefined, or when the radius follows from geometry already established.

This function starts drawing the circle from the center. The circle is defined by a value for the radius and the determination of a center point.

These options are for entering the radius:

- Verify the last entered radius by ENTER;
- Enter a new value for the radius manually; or
- Pick two points for the radius in the drawing. This is especially useful when the radius follows from an existing geometry.

After entering the radius, the center point of the circle has to be identified.

Diameter

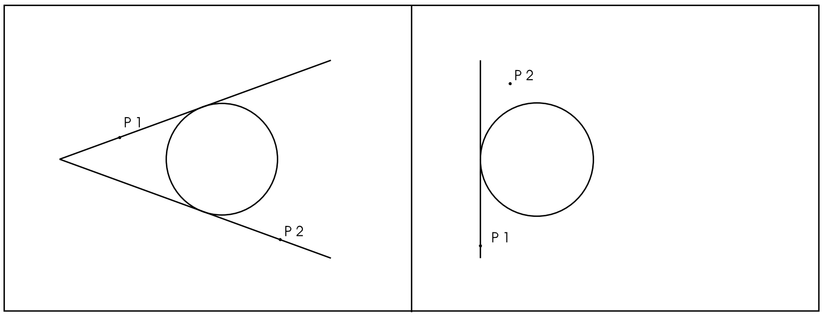
The use and the implementation of the *Diameter* option follows exactly the use and implementation of the option *Radius* except that instead of choosing the radius of the circle, you must specify the diameter.

Tangent

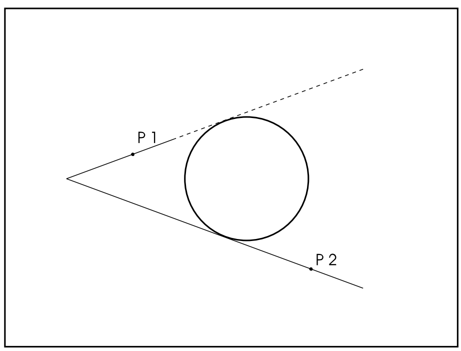
This option, frequently used, will draw a circle of given radius or diameter tangent to two lines.

The first prompt will ask for the tangent elements. First, select these entities. After choosing the second object, enter the radius of the circle to be drawn. The radius of the last circle drawn is displayed as the default value by the program. Verify this choice with ENTER or choose a new value manually or by pointing it on the screen.

The circle will be placed so that it touches, tangentially, the two chosen entities.

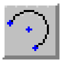


If the chosen radius is smaller than the distance of the two objects to each other, the circle will be moved in the direction of the imaginary point of intersection, so that the defined radius will touch the extension of the elements.



If the specified radius for a tangential construction is not valid, the program gives the message: *Circle does not exist!*

Arc

 There are several ways, geometrically, to describe an arc, just as with a circle. The program allows you to choose from among these methods, and draw the arc in the most convenient manner.

The following options for drawing an arc are available:

- Three Points (starting point, second point, end point);
- Two points and radius;
- Two points and angle of opening;
- Two points and center point;
- Two points and tangents through starting point;
- Center point, starting point and angle; or,
- Center point and two angles.

Variations of the options of this command may combine some of the basic geometry inherent to each method.

Note:
All arcs will be measured to the left, starting from the positive x-axis.

It is important to recognize this directional convention. To pick points for the construction of arcs, you should use the object snap options.

The options are displayed in the option bar when the command is active:

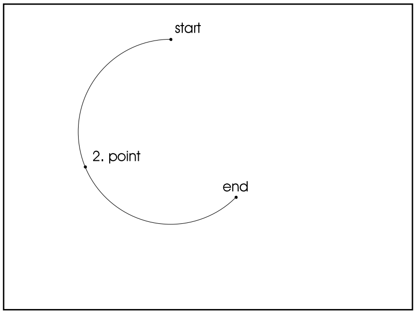
3Point	SEnd	SCenter	CStart	Cradius	Append

3Point

Since the method most used for drawing arcs is to select three points on the arc, that is the option chosen as the "standard", or default, option. It will be used if no other option is chosen.

The arc to be drawn is defined by three points. At the prompt, enter the starting point, a second point on the arc's circumference and the end point of the arc.

Remember the directional convention; the arc is described to the left starting with the positive X-axis.

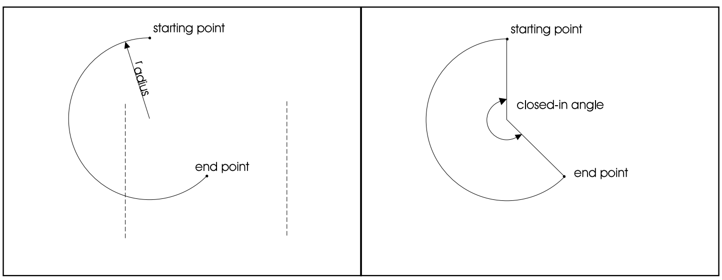



```
Start point: P1
Second point: P2
End point: P3
```

Starting Point - End Point - Radius/Angle

Using the option *SEnd***,** you may create arcs with only the starting point and end point defined. The arc is defined by specifying a radius or an angle.

The basic variation of this option simply follows the prompts, entering the starting point and end point and the radius of the arc by picking or by entering through the keyboard.




```
Starting point: P1
End point: P2
Radius: Value
```

It is also possible to enter a radius, an angle or a direction from the starting point instead of the starting point and the end point. To do that, choose the option *Angle* or *Startpoint*.

Angle

With this option the arc is defined by the included angle of the arc (angle between the starting point, the center point of the arc and its end point), the starting point, and the end point. The prompts will request a value with the words *Included angle:*

Enter that value either through the keyboard, or by picking a value. When picking a value, remember that the x-axis is the point of reference, not the last point entered.

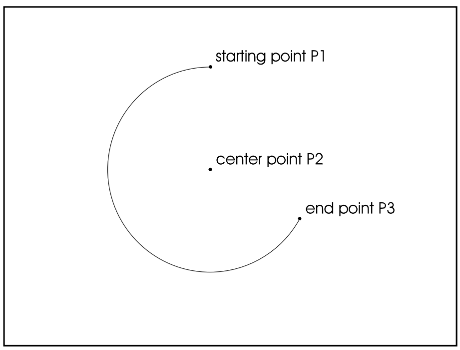
Direction

The arc is defined by a tangent which touches the arc at its starting point, next to its starting point and end point.

Since the starting point of the arc is already defined, it is only necessary to define a tangent by keyboarding or "picking" another point on the element to be tangent. The program will prompt you to specify the coordinates of a point with the word **Direction**.

Starting Point - End Point - Center Point

The Option **SCenter** is a variation of the option *SEnd***,** in which the arc is drawn by determination of the starting point, the end point and a center point .



Choose the option *Center* from the options bar after having entered the starting point. Enter the coordinates of the center point manually or pick them with the cursor. The arc is defined when the end point is entered. It is also possible to end the arc by entering an angle manually or picking it instead of the end point. The program expects a closed angle and defines it as the angle between the starting point, the center point and the end point of the arc.

CStart

The option **CStart** draws the arc starting from the center point. After choosing the option from the options bar, define the center point. Then enter the starting point and the end point of the arc (it is possible to enter a closed angle instead of the end point).

```
Center point: P1
Start point: P2
End point: Angle Specification
```

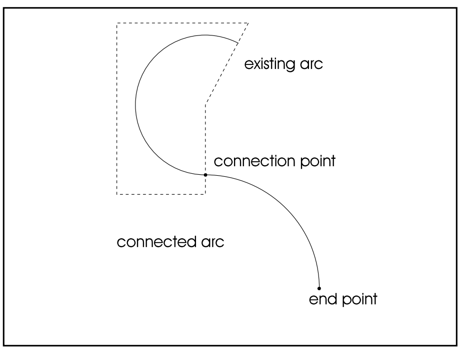
CRadius

CRadius also starts an arc by defining the center point. After that, determine the radius as well as the starting angle and the ending angle of the arc to be drawn. A rubberband line that starts in the center of the arc helps you to determine the angle by pointing.

```
Center point: P1
Radius: Radius Specification
Starting angle: P2
Ending angle: P3
```

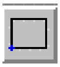
Append

It is possible to append, or connect, arcs to lines or other arcs with the option *Append*. The arc is determined by two points, the starting point being the connection point, and the end point of the new arc. First choose a line or an arc to which the new arc should be appended. Then determine the end point of the arc to be drawn.




```
Choose the line or arc: P1
End point: P2
```

Rectangle



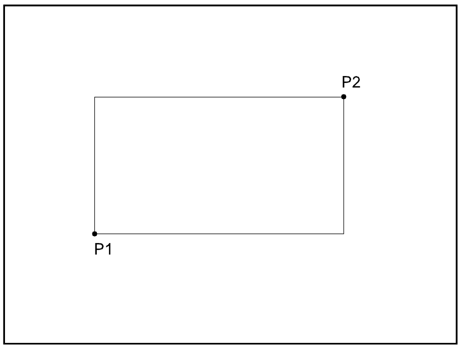
The command **RECTANGLE** allows you to construct rectangles of any size and side ratios.

When the command is active, the following option bar is available:

Width					

Rectangles are drawn by entering or picking two opposite corners.

The option *Width*, shown in the context bar, allows you to enter or pick a numeric value for the thickness of the line. To use the last entered value (which will be offered by the prompt) just press ENTER.

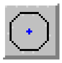



```
First corner of rectangle: P1
Opposite corner: P2
```

or

```
Width <0.00>: 0.2
First corner of rectangle: P1
Opposite corner: P2
```

Ngon

 The command **NGON** is used to draw regular polygons. It is possible to draw polygons with up to 1024 sides, a value which should lie above any practical need. When using this command, three options, based on different principles of construction, are available:

- definition of the length of sides of the polygon;
- definition of a center point and a radius of a circle outside; or
- definition of a center point and a radius of a circle inside.

Select the command for drawing a polygon and indicate how many sides it will have. You can also choose to set a width if desired.

Width					

Next, choose one of the options from the options bar.

Side	Outside	Inside			

Length of the Sides

Side is the standard option for drawing regular polygons. It will be used if no other option is chosen.

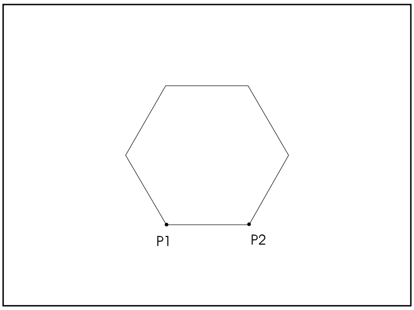
The regular polygon is defined by the length and placement of one side .

To draw a regular polygon, enter the number of its sides first. To do that, enter a value between 3 and 1024.

If there have been polygons drawn previously, then a number will appear in the prompt. That number is the number of sides of the polygon drawn last. You may confirm that number by ENTER.

After specifying the number of sides, determine the starting point and the end point of the first side either by entering the coordinates or by picking it with the cursor.

Remember that the polygon will be drawn counterclockwise, starting from the first side.

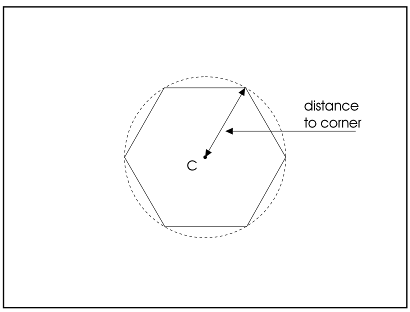



```
Number of sides <4>: 6
First corner point of the side: P1
Distance or second point of the side: P2
```

Outside: Circumscribing the Polygon

The option *Outside* defines a polygon by determining a center point and the distance to its corner points. This distance is the radius of a circle that contains the polygon, a point on whose circumference all corner points of the polygon lie. After entering the number of sides, choose the option *Outside* from the option bar.

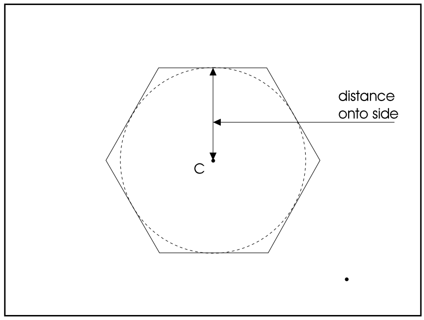
Determine the center point of the polygon by entering its coordinates or picking it. Next, determine the distance to its corner points. The polygon will be drawn counterclockwise, starting from the first corner point.




```
Number of sides <4>: 6
Center point of the polygon: P1
Distance to its corners: P2
```

Inside

Inside allows the polygon to be defined by the circle whose circumference touches each side of the polygon at its mid-point. Choose the option *Inside*, then enter the number of sides. Next determine the center point of the polygon by entering its coordinates or picking it, then determine the radius of the circle. The polygon will be drawn counterclockwise, starting from the thus defined side.




```
Number of sides <4>: 6
Center point of the polygon: P1
Distance to its corner points: P2
```

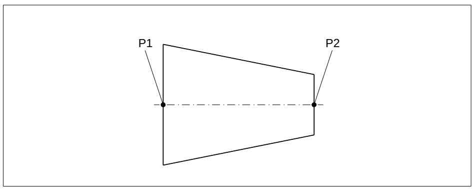
Trapezoid

The command TRAPEZOID allows you to draw a single conical object (trapezoid) on the current layer.

After entering the insertion point and angle of inclusion, the cone is defined by its length and the diameter of its top and bottom. To do so, determine the needed points successively through pointing or entering the values or coordinates manually.

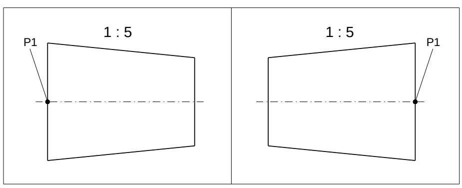
The cone can also be drawn using a ratio of **tapering**. To do so, press the ENTER key after the question of the diameter of the second side and enter a tapering ratio at the prompt.

Trapezoid via length and diameter of the top and the bottom




```
Insertion point: P1
Insertion angle <0>: <hit Enter key>
Diameter of the top: 40
Length: 50
Diameter / <hit Enter key> = Trapezoid tapering: 20
```

Trapezoid via Tapering




```
Insertion point: P1
Angle of inclusion <0>: <hit Enter key>
Diameter of the top: 40
Length of the Trapezoid: 50
Diameter of the bottom/ <Return> = Trapezoid tapering:
<Hit Enter key>
Entry of the cone tapering: 1 :     5
```

Ellipse

The command ELLIPSE allows to draw a full ellipse. Two initial options are provided to draw ellipse-shaped polygons: *Center* (default) and *Endpoint*.

Center	Endpoint				

The default variant is to identify *Center* point of the ellipse first and then specifying the primary radius (endpoint of the first axis) and secondary radius (endpoint of the second axis). Note, that the specification of the primary radius already specifies the orientation of ellipse.

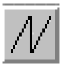
```
> ELLIPSE
Center point:
Primary radius: @3<45
Secondary radius: 2
```

An alternate way to draw an ellipse is provided by the option *Endpoints*, where you first specify the startpoint and the endpoint of the major axis and then specifying the secondary radius (by entering a numerical value or by rubberbanding and picking the radius from the midpoint of the major axis).

```
> ELLIPSE
Center point: Endpoint
First axis endpoint:  Startpoint of the major axis
Second axis endpoint: Endpoint of the major axis
Secondary radius:
```

The ellipse will be creates as closed polyline with straight line segments.

Chain

 The command CHAIN is somehow a combination of the commands Line and Polyline. The differences in drawing lines, polylines, or chains are:

- Chain allows you to draw continuous lines in a manner similar to the LINE command using the option *Continuous*. Continuous Lines draws each different segment as a separate element. That way each element can later be chosen separately using the object selection functions. The continuous line in a chain, in contrast, is treated as *one* object which can only be chosen as a whole for editing or for other purposes.
- A continuous line drawn as a chain allows you to choose the width for the entire chain object. This is similar to the POLYLINE command. The width value last entered for a polyline will be used as default for the next chain, and vice versa. A single width of the chain line must be specified.

A chain is drawn by specifying the points that will be the starting point and end point of each chain segment. The end point for each segment will serve as the starting point for the next segment. Terminate the command by pressing ENTER or ESC.

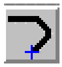
```
First point: P1
Next point: P2
Next point: P3
Next point: <hit Enter key>
```

Width	TT				

The option **Width** allows you to *define* the width of the entire chain. The width must be specified before entering the first point. Picking or entering the first point is understood as a verification of the default width value. The last entered value (also from the POLYLINE command) will be used as preference.

The option **TT** allows you to draw from a tangent of one arc or circle to a second tangent of an arc or circle.

Polyline

 Polylines are drawing elements that consist of connected lines or connected arc segments. Different elements can be of different widths. The width of a polyline segment can be defined differently for the starting and end point.

A polyline is defined as **one** object and treated as such for further editing operations, in contrast to entities drawn with the commands LINE or ARC.

Polylines will always be drawn with solid fill.

Select the command for drawing polylines and choose an option from the options bar:

Arc	Width	Halfwidth	Append	TT	

Polyline segments are always drawn by defining their starting point and end point and their width, regardless if they are arcs or lines.

The program expects that you will want to draw more than one segment when drawing a polyline. After drawing one segment you will be asked to name another end point for a new polyline segment. The starting point will be the end point of the segment last drawn.

To terminate drawing the polyline press the ENTER key or the ESC key.

```
First point: P1
Next point: P2
Next point: P3
Next point: P4
Next point: <hit Enter key>
```

Width / Halfwidth

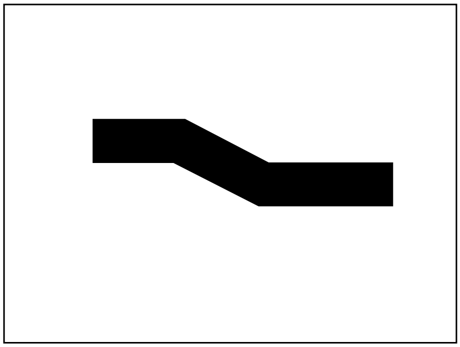
The options *Width* and *Halfwidth* are available from the options bar to assist in defining the width of a polyline.

Width defines the entire width of a polyline segment, whereas *Halfwidth* defines the width as the distance of an imaginary centerline to one side of the polyline segment.

Verify the last entered value by ENTER or enter a new width manually or by picking it.

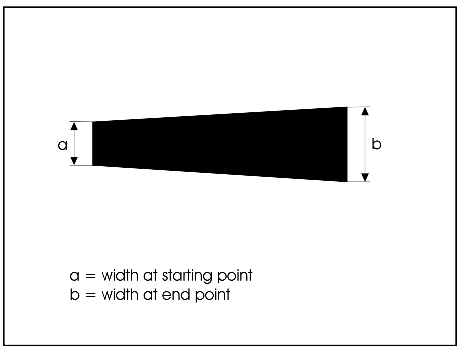
The following rules are in effect when using *both* options:

- If you choose the option *Width* or *Halfwidth* before determining the starting point for the first polyline segment, then the chosen width is valid for the entire length of the segment.



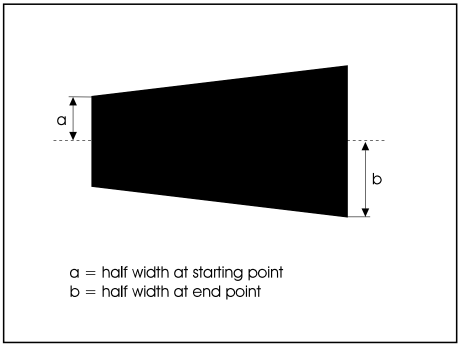
```
Width for the entire polyline <0.500>: <hit Enter key>
First point: P1
Next point: P2
Next point: P3
```

- If you choose the option *Width* or *Halfwidth* after entering the starting point for the first polyline segment, it is possible to enter different values for the width at the starting point and end point.




```
First point: P1
```

```
Next point: Width
Starting width <0.500>: <hit Enter key>
Ending width <0.250>: 0.750
Next point: P3
Next point: P4
```




```
First point: P1
Command: Halfwidth
Starting width (half) <0.50>: <hit Enter key>
Ending width (half) <0.25>: 0.75
Next point: P3
Next point: P4
```

- The last chosen width will be used for all following polyline segments on its entire length. To determine another width, choose the option *Width* or *Halfwidth* again.

Arc

Polylines has an option for drawing arcs, and a width may be correlated to the arced polyline. The method of drawing arcs is identical to drawing elements using the command ARC.

To draw polyline arcs with a special line width, follow these steps:

- Determine the first point of the arc. This can also be the end point of the previous polyline segment when drawing a polyline;
- Call up the chosen option, *Width* or *Halfwidth*, and enter the parameters for the starting width and ending width.
- Determine the other parameters for drawing arcs using the same procedure as drawing "regular" arcs.

Append

The option *Append* allows you to connect a polyline with another line or an arc.

Choose the element (line or arc) to which the polyline still to be drawn is to be appended. Call the options *Width* or *Halfwidth*, if required, and determine the parameters for the starting width and the ending width.

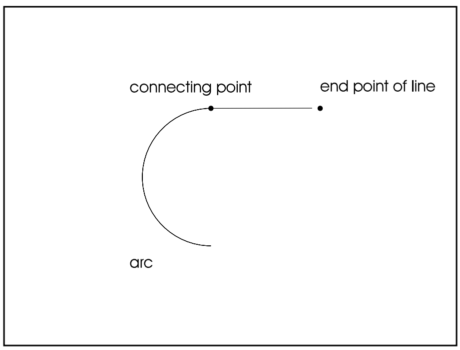
Polyline to Line

To choose an object, select an already existing line. Remember that the polyline will be connected to the end point of the line which is closest to the point selected when choosing the element.

To append a polyline, you must pick the end point of the line to be connected. This point will be appended to the end point of the chosen existing line.

Polyline to Arc

To append a polyline to an existing arc, you must enter a polyline length after choosing the existing arc. A polyline with the specified length will be connected tangentially to the end of the arc.




```
Select line or arc: P1
Length of line: P2
```

To end the polyline (appending), press ENTER.

TT

The option **TT** allows you to draw from a tangent of one arc or circle to a second tangent of an arc or circle.

Pcontour

Some will recognize this command by the terms Boundary or Bpoly. The Pcontour command creates a boundary polyline from existing objects that form an enclosed area. The original entities can be made up from a combination of Lines, Arcs, Circles, Polylines, chains and ellipses. They can be in any arrangement as long as the boundary will create an enclosed area without looping back upon itself. This command used in conjunction with the Area command can make your job a little easier.

The result is a closed polyline, created on the current layer.

When you run the command Pcontour the following prompts will be displayed;

Select boundary edges: (any valid selection method is allowed)

13 selected. (indicates the number of objects selected)

Select boundary edges: (press enter)

*** 13 selected. ***

Ray cast to find start entity <+X>: (described below)

Next	+X	-X	+Y	-Y	

Select point inside boundary contour: (pick inside the enclosed area)

If you pick a location that will not work or one of the boundary entities is not a valid entity then you will see the error;

Point located outside of a boundary contour or given geometry too complex!

Ray Cast

Usually the Next option produces a desirable result. However, it may not be satisfactory when the space between two possible boundaries is very narrow. By selecting one of the other options, you can look for the boundary in a specific direction from the point you select.

Next

Runs a line from the point you specify to the nearest object and then traces the boundary in a counter-clockwise direction.

+X

Runs a line in the positive X direction from the point you specify to the first object encountered and then traces the boundary in a counterclockwise direction.

-X

Runs a line in the negative X direction from the point you specify to the first object encountered and then traces the boundary in a counterclockwise direction.

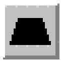
+Y

Runs a line in the positive Y direction from the point you specify to the first object encountered and then traces the boundary in a counterclockwise direction.

-Y

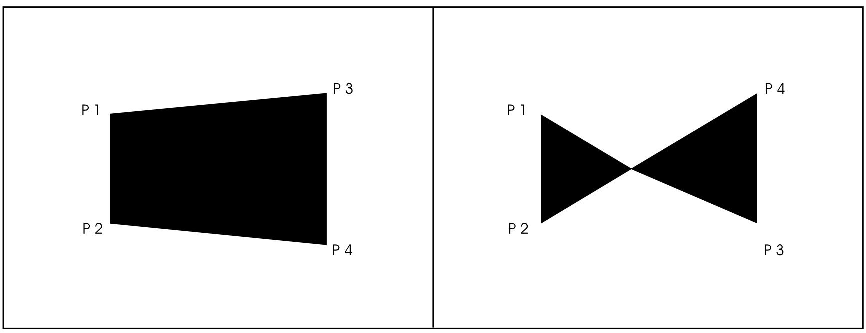
Runs a line in the negative Y direction from the point you specify to the first object encountered and then traces the boundary in a counterclockwise direction.

Filled Faces

 The command 2DFACE makes it possible to draw solid filled planes that have straight-line borders. This is done by creating and connecting any number of three- or four- cornered planes. It may be done continuously or using segments. It is also possible to connect 2D planes to already existing 2D faces.

2D Faces are always drawn by determining the corner points (three or four). The option *Close* is available to draw triangular planes so that the third entered point is connected with the starting point of the plane. Note, when defining the end points of the edges, that the succession in which these points are entered influences the resulting shape.

You should always specify the end points of the edges in the same direction. If the end points of the edges of a plane are determined clockwise or counterclockwise, a “butterfly” effect results. This drawing clarifies the consequences of the succession of specification of points.



Enter the command 2DFACE for drawing filled faces and choose an option from the options panel:

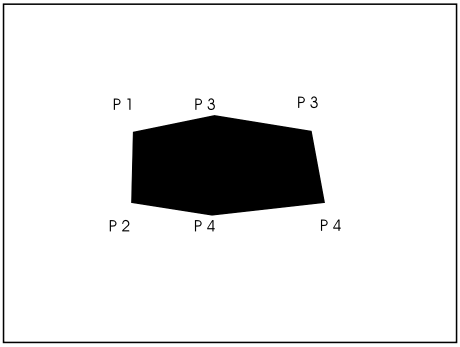
Continuous	Segments	Append			

Continuous

Continuous is the standard option for drawing filled planes; it will be used automatically if no other option is chosen.

When drawing rectangular planes, determine the end points of the edges. Remember that the first and third and the second and fourth point are connected. The edges will make up the plane.

After determining the end points of the first two edges, you may enter additional points, and they will be used as third and fourth edge (and so on) points for another plane immediately connected to the first one. To terminate the command, press ENTER.



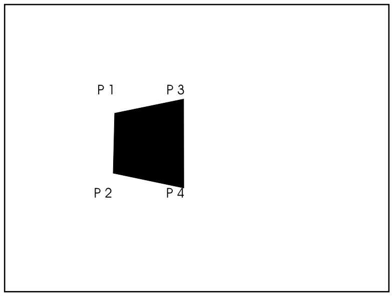

```
> 2DFACE
First point: P1
Second point: P2
Third point: P3
Fourth point: P4
Third point: P5
Fourth point: P6
...
```

To draw a triangular plane, specify three end points for two of the edges successively, and then choose the option *Close*.

```
> 2DFACE
First point: P1
Second point: P2
Third point: P3
Fourth point: Close
```

Segments

The option *Segments* is used to draw single rectangular or triangular filled planes. In contrast to the option Continuous, this option will end after the entry of the points required for the plane. For triangular planes, there is no difference from the option Continuous.

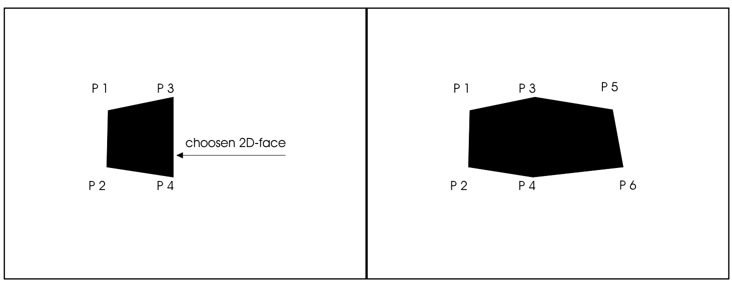



```
> 2DFACE
First point: P1
Second point: P2
Third point: P3
Fourth point: P4
```

Append

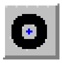
This option allows to append newly created planes to already existing ones. It is only possible to append a plane to the edge last drawn. After choosing the option *Append*, select an edge of an existing plane to which the new plane is to

be appended. Then determine the corner points of appending plane.




```
> 2DFACE
Choose a side: Identify edge
Third point: P5
Fourth point: P6
...
```

Rings

 The command RING is used to draw solid rings or circles and covers a special case of a closed polyline.

Two concentric circles are drawn, the smaller one is the inner diameter, and the larger one the outer diameter, of the ring. The space between the circles will be filled. Creating filled circles is a special case of a drawing rings. It is a ring with the interior diameter equal to zero.

Rings can be edited using any of the polylines editing commands. They will behave just like a polyline object.

The last entered diameter values will be used as an option for the next entry when drawing rings. If you want to keep using the same values, just enter a new center point. This step can be repeated as often as needed, always creating identical rings.

```
> RING
Center point: P1
Center point: P2
Center point: <hit Enter key>
```

Ring Diameters

To determine new values for the inside or outside diameter, just choose the

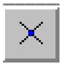
option *Diameter* from the context bar. With this option it is possible to determine the inside and outside diameter of the rings to be drawn either via keyboard or by pointing on the screen.

Diameter					

```
> RING
Center point: Diameter
Inside diameter <0.50>: 0.75
Outside diameter <1.00>: 1.25
Center point: P1
...
```

It is possible to draw as many identical rings as required by simply determining their center points.

Points

 It is possible to insert construction points into a drawing using the command POINT. The following variations are available:

- Individual insertion of points of construction;
- Insertion of points at a specified distance from each other (Measure);
- Insertion of a number of points along a line (Segments).

Point	Measure	Segments			

The form and size of the points of construction to be inserted is determined by specifying these parameters using **Drawing Parameters...** (see the beginning of this chapter).

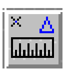
The construction points will always be inserted on the current layer.

Point (Individual Construction Points)

This option is activated automatically when calling upon this command and thus does not have to be called up explicitly.

Determine the position of the points to be inserted either via keyboard or via pointing on the screen. The command remains active after the insertion of the first point, and you can repeat this procedure as often as is required. End this function by pressing the ESC key or the ENTER key.

Measure

 The option *Measure* allows you to insert points of construction on lines, polylines, circles and arcs in a defined spacing.

That dissects the elements into sections of a certain size. The drawing object is *not* cut into different pieces but remains one complete object.

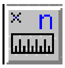
Select the option *Measure* from the context bar when the command POINT has been called.

Remember that all objects that are not an exact multiple of the length of the segment will have a remainder.

Choose a drawing element and determine the spacing of the segments (in drawing units), either by entering a numerical value or by cursor picks.

The insertion of the points into lines, polylines and arcs will be done beginning at the starting point of the segment and progressing to the end point. When inserting points of construction into a circle, the starting point lies at the intersection of the radius which has its starting point at the center point of the circle and is in the direction of the positive x-axis.

Segments

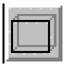

The option *Segments* allows you to separate lines, polylines, circles and arcs into segments of equal length by inserting points of construction. The size of the segment is determined by the length of the distance to be dissected and the number of construction points to be inserted.

In contrast to the option *Measure***,** there will not be a remainder when using the option *Segments*. The object being segmented is *not* cut into different pieces but remains one complete object.

Choose the option *Segments* from the options bar. Now choose an object, and determine the number of segments into which the object is to be segmented. The number of segments should be a whole integer between 2 and 4096.

The insertion of segmentation points on a line, polyline, or an arc is done starting from the beginning of the segment and progressing to its end point, without inserting a point at the beginning or at the end. When inserting points into a circle, the starting point lies at the intersection of a radius which begins at the center of the circle and continues along the positive X-axis.

3D Faces

 The 3DFACE drawing command creates three or four sided surfaces "in space", or, in other words, in an X, Y, Z coordinate system. The difference between 2D and 3D surfaces is that the 3D surfaces are not shown as solid areas, but are drawn as wire frame models.

The 3D-face definition points are fixed in either a clockwise or counter-clockwise direction so that one edge of a 3D surface is used as the base line for the next 3D Face.

You may also draw invisible edges, a function which may increase the clarity of complicated 3D structures.

These options to draw 3D faces may be selected from the options bar:

Continuous	Segments	Append	Invisible		

Continuous

The option *Continuous* is the default option when drawing 3D surfaces.

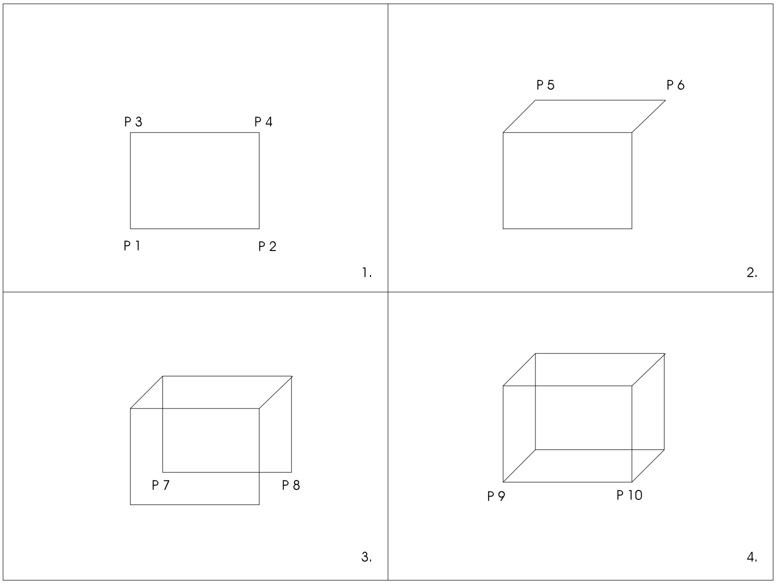
To draw a 3D-face, define the corner points in a clockwise or counter-clockwise direction.

For a triangular face, define three corners and then select the *Close* function located in the options bar. The last defined point will then be joined to the first point, forming a triangle, and the drawing will be completed.

➢ **3DFACE**
```
First point: P1
Next point: P2
Next point: P3
Next point: Close
```

For a rectangular 3D-face, define four points. After having completing the fourth entry the command creates a four sided object.

This process is then continued. You will be asked to define further points. The next 3D-face to be defined will use the last drawn edge as the base line for the next 3D-face.



End the 3D-face drawing process by pressing the ENTER key.

```
> 3DFACE
First point: P1
Second point: P2
Third point: P3
Fourth point: P4
Third point: P5
Fourth point: P6
Third point: P7
Fourth point: P8
```

and so on.

Segment

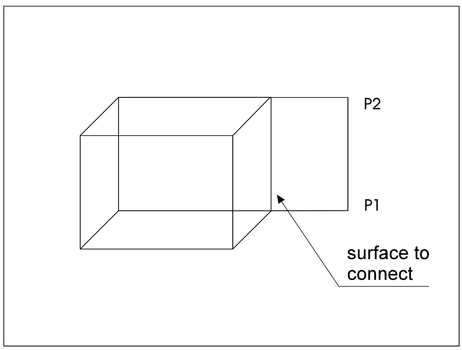
To draw a single 3D-face use the *Segment* option. The method is almost identical to that just described.

```
> 3DFACE
First point: P1
Second point: P2
Third point: P3
Fourth point: P4
```

In contrast to the *Continuous* option, the drawing process will end with the definition of the fourth point. The option *Segment* also allows the use of the *Close* function, thus enabling triangular shaped 3D faces to be created. Should you wish to add further 3D surfaces to an existing one, this can be achieved at any time by utilizing the *Append* option.

Append

The option *Append* allows you to connect new 3D-faces to existing 3D-faces. First select the edge which is to act as the base line for the connection, then proceed by defining the required corner points for the new 3D surface. The option will automatically change the mode to *Continuous*. This means that after having connected one 3D-face you can define additional pairs of points to create more surfaces.



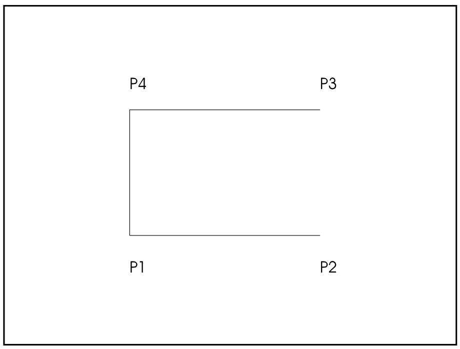

```
> 3DFACE
First point: Append
Select line or edge of 3DFace: P1
Third point: P2
Fourth point: P3
Third point: P4
Fourth point: P5
and so on.
```

Invisible Edges

The option *Invisible* is used to draw hidden lines in a 3D-face. This feature is particularly useful when dealing with complex drawings containing a large number of 3D-faces. The use of hidden lines will make the drawing more understandable. This function is available for all modes used in 3D-face drawing. The function must be activated prior to the definition of the starting coordinate of the invisible line.

```
> 3DFACE
First point: P1
Second point: Invisible
Second point: P2
Third point: P3
Fourth point: P4
```

would result in the following:



3D Polyline

3D polylines are drawing objects created from straight line segments. You can to define 3D coordinates for each individual 3D polyline control point (vertex).

You may include curved segments in 2D-polylines, as described earlier, but this is not possible with 3D-polylines. It should also be noted that the entire polyline is assigned a standard width of zero.

As opposed to a line created using the LINE command, a 3D polylines is treated as a single object. This applies when the object is being processed with modifying functions. The command to draw a 3D Polyline may be accessed by selecting the option from the menu, or by typing in the command POLY3D.

```
POLY3D
First point: P1
Next point: P2
Next point: P3
Next point: <hit Enter key>
```

While using this command you have the following choices.

Undo	Close				

Chapter 6

Modifying Drawing Objects

This chapter deals with modifying and editing commands found in the menu **Modify**.

These commands provide translation functions such as moving, rotation, scaling, mirroring, copying, and duplicating, and repeating, and combinations of these operations. They also provide a means by which to detail the drawing, providing fillets, chamfers, trimming, stretching, and other detailing aids. The commands covered by this chapter are listed below.

Move	**Expand**
Copy	**Fillet**
Rotate	**Chamfer**
Scale	**Intersect**
Flip	**Lengthen**
Mirror	**Delete partial**
Stretch	**Rejoin**
Array	**Change Entity's Geometry**
Offset	**Modify Entire Polyline**
Trim	**Modify Polyline Vertex**

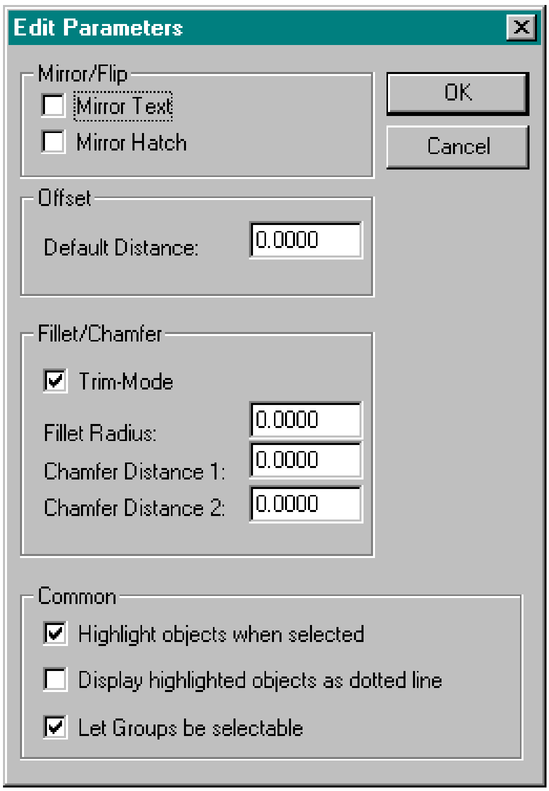
In order to apply any of these commands a drawing must be open, and an entity must have been created.

Object Selection

All modify commands will request objects be chosen for modification. The Option Bar suggests several means of selecting the objects. Multiple objects may be selected, and they will be transformed together in exactly the same relationship as they are currently placed. If more than one entity is selected, all entities will be regarded in the editing operation as a selection set, and they all will be affected by the change.

Modify Parameters...

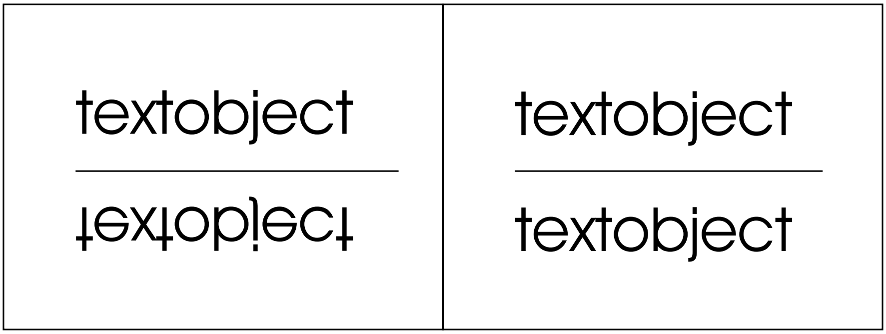
The function **Modify Parameters...** allows you to define a series of parameters, which are used as default values for some modifying and editing commands. Typing the command EDITPAR will also get this function.



Mirror / Flip

Mirror / Flip determines the position and orientation of text objects in the drawing. Activate Mirror / Flip by checking the box MIRROR TEXT. When this option is active, it will cause the text objects to be mirrored, with the original text object remaining in place. With the option de-activated, the text objects will be copied, but not mirrored, about the specified axis.

The following picture shows the influence of this selection on the display mode of text objects.

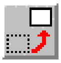


Mirroring of a text object, option **Mirror Text** activated (left) or not activated (right)

Mirror Hatching will be treated the same way as text, If highlighted.

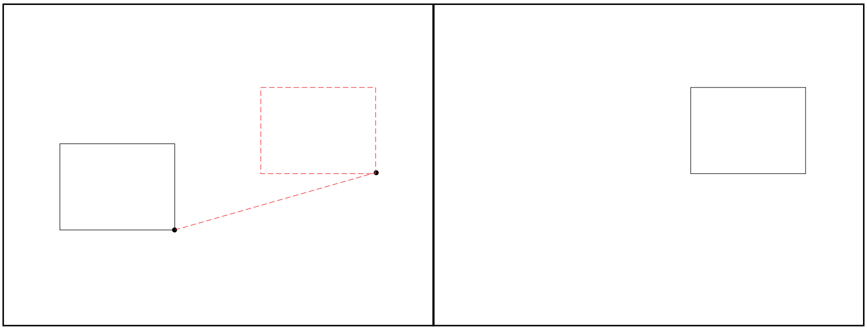
The functions **Offset** and **Fillet / Chamfer** will be discussed in detail later in this chapter. The values set for them in this Dialog Box are default values, only.

Move



Move will move drawing objects within the coordinate system.

Select the objects to be moved. Multiple objects may be selected, and they will be moved together in exactly the same relationship as they are currently placed. Confirm the selection by pressing ENTER.



Position of an object before (left) and after (right) the moving

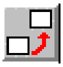
The next prompt will ask for the **Basepoint.** It is the reference point for moving objects. It denotes the point that you is choosing as the base from which to begin the command.

Basepoint	Rel.Point				

If, however, you wish to move the objects a specific distance from their present location, you may select **Rel.Point** from the option bar. The prompt will then ask for distances in the XYZ direction. Enter these values (if there is no Z value enter only the X and Y values). The move will occur when the values are verified.

The third method of accomplishing the move is to select the objects with the cursor, and “rubberband” them to the new desired location. This works if no particular precision is required, or if **OSNAP** can provide the new location snap point.

Copy



Copy is used to duplicate objects within the coordinate system. Select the objects to be copied, and confirm with ENTER. Again, the Option Bar will display the selections **Basepoint** and **Rel.Point**.

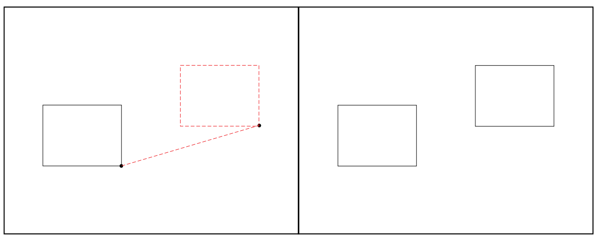
Basepoint	Rel.point				

Basepoint is usually chosen to copy. **Rel.Point** may be chosen if the copy is to be placed in a position relative to the current image of the object.

If **Basepoint** is the selection, enter the Basepoint, then, as prompted, the target point (the position to which the copy is to move). The copy will be made in the position specified.

If **Rel.Point** is chosen, the prompt will ask for the relative points by specifying the move in the XYZ directions. If the move is to be made in the X and Y directions only, simply do not specify a Z delta. The copy will be placed in the specified position.

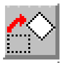
The copies created by this command are exact copies of the selected entities. All entity attributes such as layers, line type, color, etc. are copied as well as the outline of the objects.



Position of objects before (left) and after (right) the copying

```
> COPY
Select objects: P1
Select objects: <hit Enter key>
Basepoint: P2
Target point: P3

> COPY
Select objects: P1
Select objects: <hit Enter key>
Basepoint: Rel.Point
Relative point dX,dY,dZ: 2,-3
```

Rotate

 **Rotate** will rotate entities through an angle, about an axis. Select the objects to be rotated.

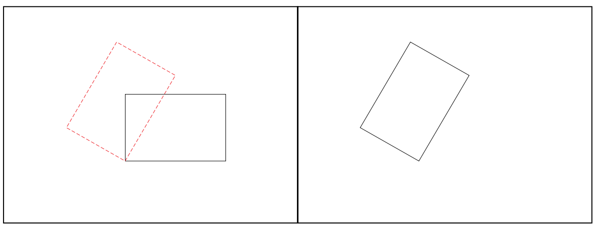
The command will prompt for the **Basepoint**. The Basepoint is the point about which the rotation will occur. It is handy to pick a corner or a center of an drawing object, but any point may be picked that will give the desired result, whether on or off of the geometry of the part(s).

The next prompt will ask for a rotation **Angle**. Enter that angle, and rotation will take place to the specified angle.

You may also select a **Reference** angle.

Angle	Reference				

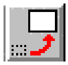
The **Reference** angle option will ask for the angle at which the object is now placed, and then prompt for a new angle to which the object will be rotated. Rotation will be counter-clockwise. As before, rotation will occur about the **Basepoint**.



```
> ROTATE
Select objects: P1
Select objects: <hit Enter key>
Basepoint: P2
Rotation angle: 60
```

```
> ROTATE
Select objects: P1
Select objects: <hit Enter key>
Basepoint: P2
Rotation angle: Ref.Angle
Reference angle <0>: 44
New angle: 45
```

Scale

 **Scale** permits you to change the scale of selected objects. Select the objects to be scaled.

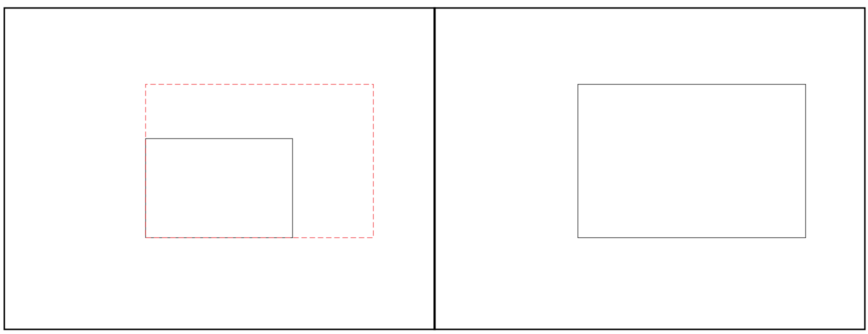
The next prompt will be to identify the **Basepoint,** and that should be done as it was in the two previous commands. Note that the **Basepoint**, if is a part of the selected object, will hold its original position during the scaling, while all the other points will be scaled by the same relative values in direction of the X- and Y-axis.

As in other cases, however, you may select any point, whether on or off the geometry of the object(s). In this case, the distance from the **Basepoint** to all of the other points that comprise the object will be changed by the same scale factor.

Then the program will bring up the options bar shown below:

Sc.Factor	Reference				

The option **Scale Factor** is the default option for scaling objects.



Enter a scale factor. The use of a scaling factor bigger than 1 results in enlarging the selected objects by this factor. A scaling factor between 0 and 1 results in reducing the objects by that factor.

For instance, the factor 3 enlarges the selected objects three times, while the factor 0.25 results in reducing the objects to one-quarter of their original size.

```
> SCALE
Select objects: P1
Select objects: <hit Enter key>
Basepoint: P2
Scale factor: 1.5
```

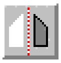
A scale factor can also be specified by indicating a second point. To do this, move the cursor after the fixing the **Basepoint**. The distance of the cursor from the **Basepoint**, measured in drawing units, will be interpreted as the scaling factor. This has the advantage of letting you see the effect of the scaling while moving the cursor, and the disadvantage of lack of precision.

Using a Reference Scale

This variant of the scaling command allows an exact change of the size of an object to a particular scale. For instance, if you were to enlarge an object with a side length of 3.45 drawing units up to 6.90 drawing units, you would first have to calculate the scaling factor. **Reference** facilitates this calculation by the input of a reference length (here 3.45) a precise changing of the size up to the desired scale (here 6.90), without the need of having to calculate a scaling factor. To use this variation of scaling, select the option **Reference** from the option bar. Enter the reference length, and then enter the new length that is desired.

```
> SCALE
Select objects: P1
Select objects: <hit Enter key>
Basepoint: P2
Scale factor: Reference
Reference length <1>: 0.95
New length: 1.0
```

Flip



Flip contrasts to MIRROR in that the original object does not remain in its original position; it is simply “flipped” about a specified axis.

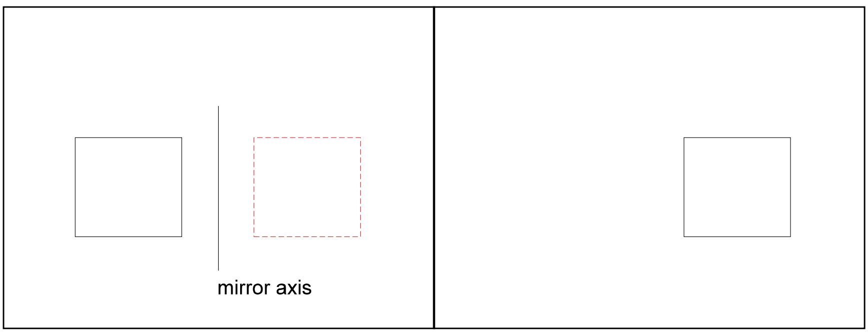
The selected objects have, after “flipping”, the same relative position to the symmetry axis as they had prior to “flipping”, but are now “flipped” in a new direction. The “flip” axis can be defined freely, or parallel to the X-axis or the Y-axis

Select the objects, then select an option for choosing the axis from the Option Bar.

Axis	Horizontal	Vertical			

Axis

Axis allows you to define the flip axis. It is the default for axis. Define the position of the axis by the indicating two points or by the input of their coordinates. The axis can have any alignment toward either the objects or the axes of the coordinate system.



If the mirror axis will run through any of the points of existing geometry of the object(s), you should specify **OSNAP** functions to define the points precisely.

```
> FLIP
Select objects: P1
Select objects: <hit Enter key>
First point of mirror axis: P2
Second point: P3
```

Horizontal Reflection

To flip the marked object about a horizontal axis choose the option **Horizontal**. This option creates a flip axis which runs parallel to the Y-axis of the coordinate system. A single point is required to define this axis.

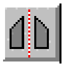
```
> FLIP
Select objects: P1
Select objects: <hit Enter key>
First point of mirror axis: Horizontal
Horizontal reflection point: P2
```

Vertical Reflection

To flip the marked object about a vertical axis choose the option **Vertical.** This option creates a flip axis which runs parallel to the X-axis of the coordinate system. A single point is required to define this axis.

```
> FLIP
Select objects: P1
Select objects: <hit Enter key>
First point of mirror axis: Vertical
Vertical reflection point: P2
```

Mirror

 **Mirror** may be entered from the menu **Modify** or by typing MIRROR.

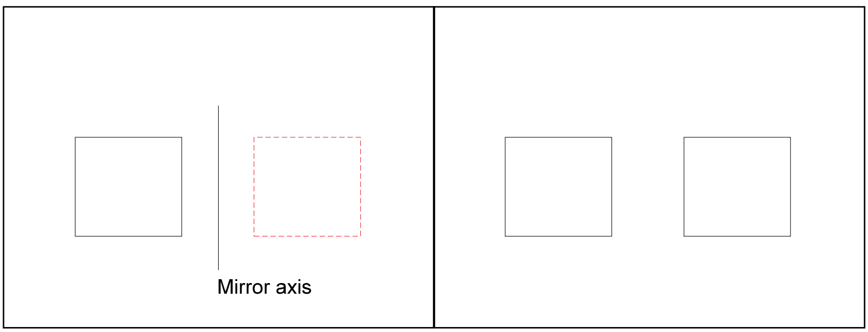
Select the object or objects, remembering that the selection of multiple objects will result in the treatment of the objects as one. The next prompt will ask for the definition of the mirroring axis.

Axis	Horizontal	Vertical			

Mirroring With a Free Axis

Axis allows you to define the mirror axis. It is the default for mirroring objects. Select the option, then define the position of the mirror axis by indicating two points or by inputting their coordinates. The mirror axis can have any alignment with regard to the objects to be mirrored as well as with regard to the axes of the coordinate system.

If the mirror axis runs through points of an existing geometry (for instance, along a solid edge), remember to employ the **OSNAP** options for precise definition.




```
> MIRROR
Select objects: P1
Select objects: <hit Enter key>
First point of mirror axis: P2
Second point: P3
```

Horizontal Reflection Axis

To choose a horizontal mirroring axis, select the option **Horizontal**. This option creates a mirror axis which runs parallel to the Y-axis of the coordinate system. It is indicated by a single point.

```
> MIRROR
Select objects: P1
Select objects: <hit Enter key>
First point of mirror axis: Horizontal
Horizontal reflection point: P2
```

Vertical Reflection Axis

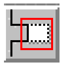
To choose a vertical mirroring axis, select the option **Vertical.** This option creates a mirror axis which runs parallel to the X-axis of the coordinate system. Its position is indicated by a single point.

```
> MIRROR
Select objects: P1
Select objects: <hit Enter key>
First point of mirror axis: Vertical
Vertical reflection point: P2
```

Mirroring Text

If there is text associated with the selected objects, it is likely that you do not want to mirror that text. In order to mirror the text it is necessary to activate an option called Mirror Text. Activating this option will result in the text also being mirrored. That option is available from the Dialog Box Edit Parameters, reached by selecting Modify Parameters from the pull-down menu Modify, or by typing EDITPAR.

Stretch

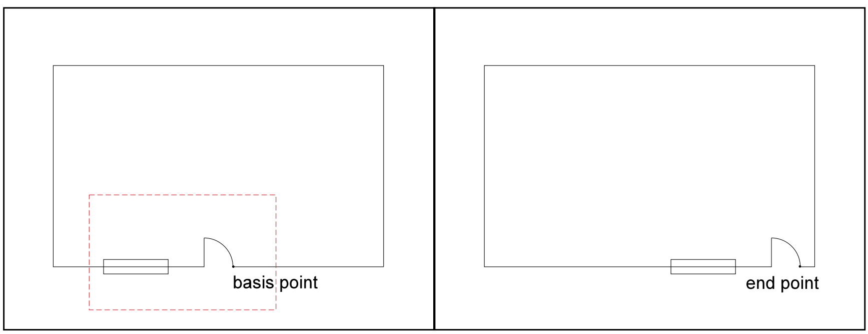


Stretch appears to be a combination of the functions MOVE and LENGTHEN. It facilitates a change of position of selected entities, but allows them to hold their relation to other entities of an object. Using this command, entities forming the borders of the object may be stretched.

In drawing practice, stretching is useful for enlargement or reduction of objects in one direction (in contrast to scaling) or for moving single entities inside of one contour.

Like all the commands and functions for editing, the selection of entities is prerequisite. Careful planning must be done in the selection of entities for stretching. During stretching, the method of selection has an influence on the question of which entities are moved and which entities are stretched.

- All entities included totally in the selecting windows will be moved;
- All entities crossing the selecting window will be stretched (lengthened); and
- All other drawing entities will retain their position and shape.



Crossing	Cpolygon	Remove			

After having completed object selection using the above options, choose an option for stretching.

Basepoint	Displacem.				

Basepoint

Basepoint is the default option for moving entities. It is useful for moving selected drawing entities to a particular position, and for stretching or moving entities on a border without changing the relationship to the border. Select the Basepoint for the stretch. Enter the coordinates or pick the Basepoint.

Next, determine the end point of the stretch. The Basepoint for the stretch will be moved to the end point.

```
> STRETCH
Select objects: P1
Select objects: <hit Enter key>
Basepoint: P2
Second point of displacement: P3
```

Displacement

Displacement stretches selected entities by a straight line (a vector). Enter the value for the move, a straight line, in the form X,Y, (Z).

```
> STRETCH
Select objects: P1
Select objects: <hit Enter key>
Basepoint: Displace.
Displacement dX,dY,dZ: 2,-3
```

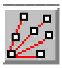
Array

Array represents a variant of COPY, except that it will create multiple copies, and they can be arranged at random, or in rows, in vertical order, on a circle, or on an arc.

The entities created by the command are exact copies of the selected entities in that all entity attributes such as layers, line types and color are copied. After entering the command, select an option:

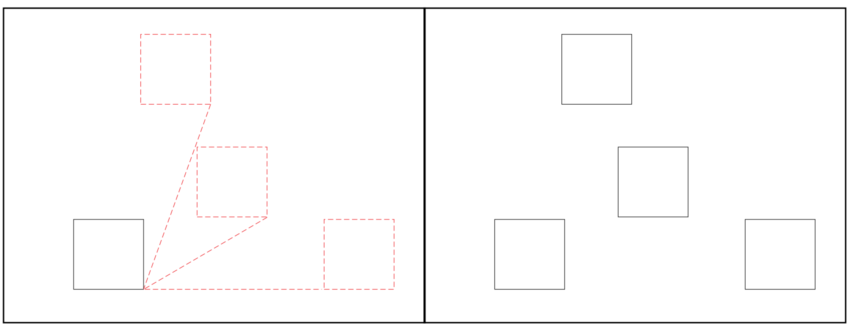
Individual	Rectangular	Circ-NRot	Ci-Rot		

Individual



This mode permits copying the selected objects as many times as is required in one sequence, and placing them at random on the drawing area. Select the entities to be copied using the selection modes in the option bar.

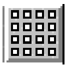
Determine the Basepoint for the copying. A particularly discreet point in the geometry or in one of the selected objects (edges, corners, intersecting points etc.) will be useful. Either input the coordinates of the target point(s) or pick the points for placement of the array. The program will create a copy of the array after each definition of a target point.



Until the command is terminated, you can produce and place as many copies as required. Terminate the function with ESC or ENTER.

```
> ARRAY Individual
Select objects: P1
Select objects: <hit Enter key>
Basepoint: P2
Target point: P3
Target point: P4
...
```

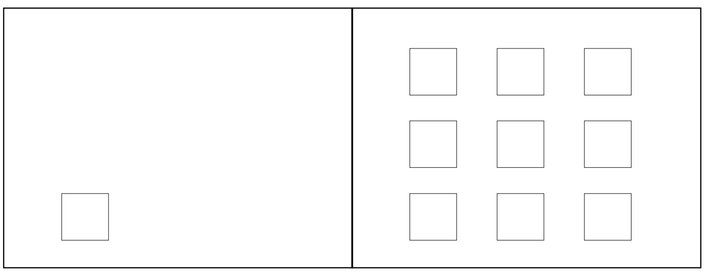
Rectangular



This option will arrange the array copies in rows or columns. The use of this function is the same as the use of the **Individual** mode.

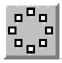
Determine the number of the columns and rows in which the Array copies should be arranged. Define the distance between the columns and the rows.

Note that, when planning the arrangement, the rows and columns must be in the positive X or Y-axis. This infers that the original Array is situated in the lower left hand corner of the quadrant plus-XY. If it is not located there, it may be useful to move the original Array before attempting to begin defining the copying positions. Use the command **Move** to do this.



```
➢ ARRAY Individual
  Select objects: P1
  Select objects: RETURN
  Basepoint: P2
  Target point: P3
  Target point: P4
  ...
```

Circular



The **Array** can also be reproduced in a circular path. The circle (arc) itself is not displayed on the screen. The reproduction of the array is described as follows:

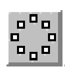
- the center of the circle (arc)
- the number of times the Array is to be reproduced;
- the included angle of the arc (360 if the path is to be a full circle) to be filled;

When it is reproducing Arrays on a circle or arc, the program selects a reference point which is positioned on the circumference of the arc (circle) for each reproduction. This reference point will differ according to the shape of the entity in this manner:

Entity	Reference point
Line	Starting- or ending point
Circle, Arc	Center
Rectangular	Starting point
Point	Inserting point
Part	inserting point
Text	inserting point

The program also offers the option **Basepoint** in the Option Bar. Using this option, the basepoint of an Array can be defined for the reproduction. Remember to use OSNAP with this option.

There are two ways to reproduce the Array on a circular path.

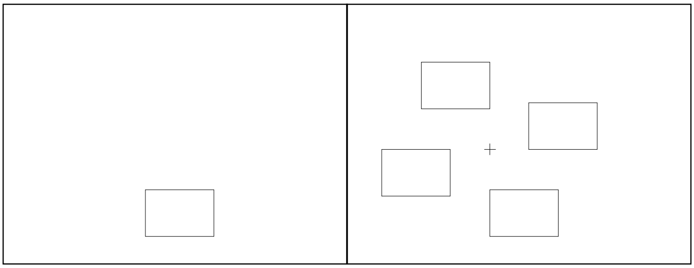
Placing the Arrays Without Rotation 

This option will arrange the reproductions on the circle (arc) in the same alignment to the principal axes as the original Array. Select the objects for the Array, choose the option, and define the center of the circular path. Then determine **either;**

- the number of the entities to be arrayed (including the original); and
- the included angle of the circle (arc),

or

- the included angle of the circle (arc); and
- the included angle between the Arrays to be reproduced (measured **between** the reference points of the objects).



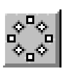
Note that planning the reproduction of the Array on a circle, the circle (arc) runs **counter-clockwise** from the starting point. For a **clockwise rotation** enter negative values for the angle inputs.

```
> ARRAY Circle-NRot
Select objects: P1
Select objects: <hit Enter key>
Center: P2
Number of entities: 4
Angle to fulfill <360>: <hit Enter key>
```

or

```
> ARRAY Circle-NRot
Select objects: P1
Select objects: <hit Enter key>
Center: P2
Number of entities: 4
Angle to fulfill <360>: <hit Enter key>
Angle between entities: 60
```

Placing the Arrays With Rotation



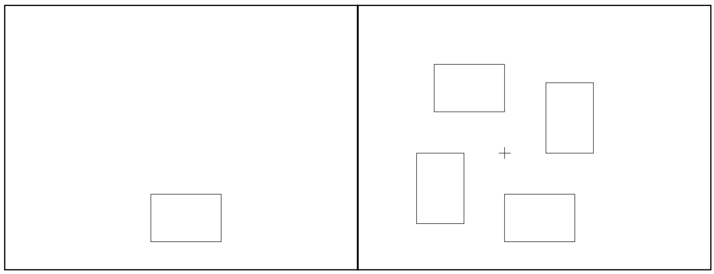
This option will rotate the copies when placing them on the circular path. The copied objects will be rotated in the same direction as the circle arc that describes the path of reproduction. The rotation angle will correspond to the angle between the copies (measured between the reference points of the arrays).

After selecting the command, choosing the objects in the Array, and choosing the option **Ci.Rot** from the Option Bar, define the center of the circular path and then determine **either;**

- the number of the entities to be arrayed (including the original); and
- the included angle of the circular arc,

or

- the included angle of the circular arc; and
- the included angle between the entities to be arrayed (measured between the reference points of the objects).



The default value is 360 degrees. Either confirm, or choose the value.

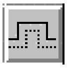
Note that, during the planning of the arraying on a circle of multiple copies, the circle (arc) runs from the starting point, that is the original object anti-clockwise. For clockwise rotation enter negative values for the angle inputs.

```
> ARRAY Circle-NRot
Select objects: P1
Select objects: <hit Enter key>
Center: P2
Number of entities: 4
Angle to fulfill <360>: <hit Enter key>
```

or

```
ARRAY Circle-NRot
Select objects: P1
Select objects: <hit Enter key>
Center: P2
Number of entities: 4
Angle to fulfill <360>: <hit Enter key>
Angle between entities: 60
```

Offset

 **Offset** creates congruent pictures of lines, circles, arcs and 2D-polylines. This command copies a selected entity(s) and places it a specific distance from the original entity. The original entity(s) stays in position.

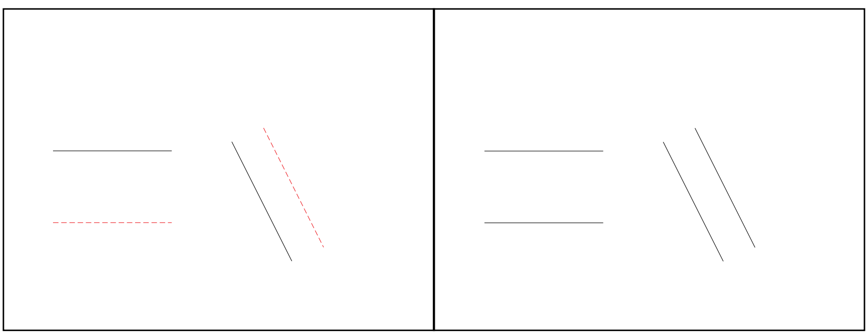
The operating mode of the command depends on the type of drawing entity:

- For lines and polylines, identical copies are created and moved in the direction of the X or Y-axis; while
- For circles and arcs copies are also created, but, in addition, they will be scaled by a selected factor. As a basepoint for scaling the center of the original circle or arc is used. In this way, concentric circles or arcs are created.

Offset for Lines/ 2D-Polylines

Enter the command, then select the line(s) or polyline(s) that are to be offset by picking with the cursor. Either select the distance for the offset by picking, or select **Distance** from the Option Bar, and enter the offset distance. Then answer the prompt **Side to Offset?** with a point on the desired side.

```
> OFFSET
Select objects (line, circle, arc, 2D-polyline): P1
Offset by point: P2
Select objects (line, circle, arc, 2D-polyline): <hit
Enter key>
```

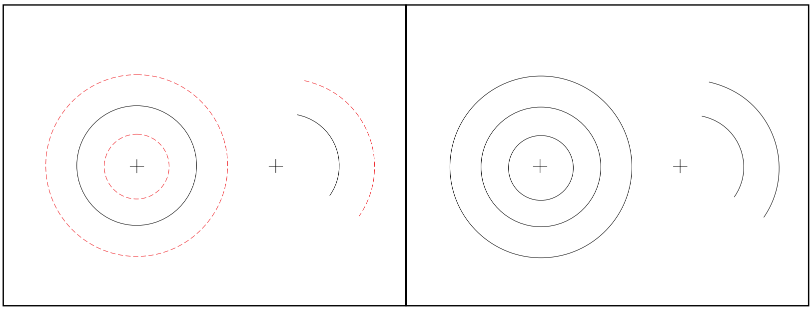


Offset for Concentric Circles/ Arcs

Concentric circles or arcs can be created with a radius or diameter smaller or larger than the original entity.

After having selected the objects to be offset, the position of the offset is determined as follows:

- by indicating, or picking, a point;
- by input of coordinates of a point on the circle or arc; or
- by using the option **Distance** from the Option Bar (see below).



```
OFFSET
Select objects (line, circle, arc, 2D-polyline): P1
Offset by point: P2
Select objects (line, circle, arc, 2D-polyline): <hit
Enter key>
```

Distance

Distance allows the offset to be created a specific distance from the original entity. Select this option from the option bar.

Distance					

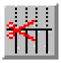
Determine the distance between the original entity and the offset by either a numeric value input (in drawing units) or by cursor selection. Then, point (and click) with the cursor to the side of the original entity on which the offset is to occur.

```
> OFFSET
Selection of an entity
(Line, circle, arc, 2D-Polyline): P1,
Offset by point: Distance
Distance <5> 10
Offset to side ? P2
Selection of an entity
(Line, circle, arc, 2D-Polyline): RETURN
```

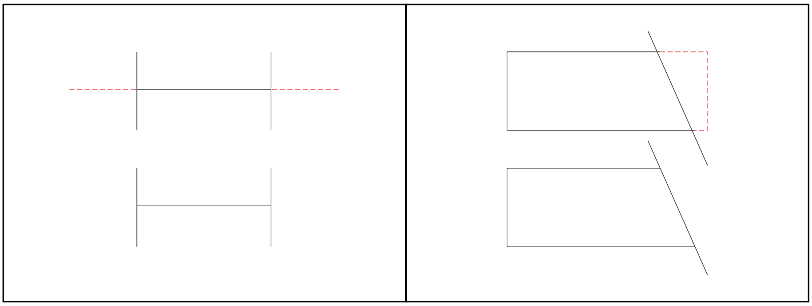
or

```
> OFFSET
Selection of an entity
(Line, circle, arc, 2D-Polyline): P1,
Offset by point: Distance
Distance <5>: 10
Second point P3
Offset to side ? P4
Selection of an entity
(Line, circle, arc, 2D-Polyline): RETURN
```

Trim

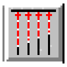
 **Trim** will a request the identify of the entities to make up the "cutting edge". These entities will be the ones that will result in the "trimmed" intersection of lines. Pick these entities with the cursor. Verify the selection by pressing ENTER.

The next prompt will ask for the entity to trim. That selection should be the portion of the cutting edge entities that you wish to dispose of. As the entities are selected, they will be trimmed, and the trimmed portion will disappear from the drawing.



```
> TRIM
Select entities for intersecting edges
(Line, circle, arc, 2D-Polyline): P1, P2
Select entities for intersecting edges
(Line, circle, arc, 2D-Polyline): RETURN
Select entity for trimming
(Line, circle, arc, 2D-Polyline): P3, P4
...
Select entity for trimming
(Line, circle, arc, 2D-Polyline): ESC
```

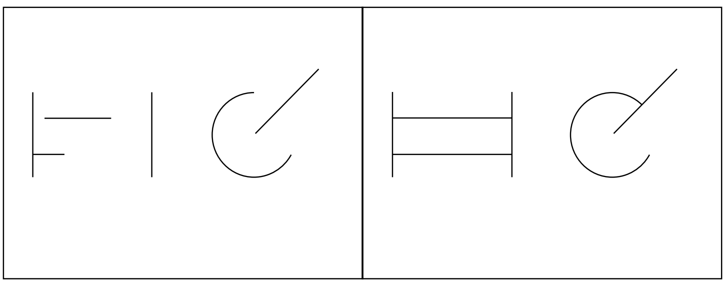
Expand

 **Expand** allows you to modify objects which should have common boundary edges. Parts of objects can be expanded to the intended boundary edge. This command makes it possible to “close” polygons, and to extend entities to the proper boundary edge.

During the execution of the command the drawing entities to be expanded are extended. They do not change direction. A line will remain a line, an arc an arc. Select the entities which are to serve as boundary edges for the entities to be expanded. These entities will remain unchanged. There are selection functions in the option bar.

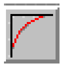
Then select the entities to be expanded. In doing this, make sure that the selecting window of the cursor is on the part to be trimmed.

The expansion of selected entities is always executed from the endpoint which lies nearest to the cursor when it selected the entity. If more then one boundary edge has been selected, the entity is expanded to the first boundary edge first. Picking a second time will result in the expansion being continued to the second boundary edge. The following illustration describes the command EXPAND.



```
> EXPAND
Select entities with boundary edges (line, circle, arc,
2D-polyline): P1, P2
Select entities with boundary edges (line, circle, arc,
2D-polyline): RETURN
Select entity for expanding
(line, circle, arc, 2D-polyline): P3
...
Select entity for expanding
(line, circle, arc, 2D-polyline): RETURN
```

Fillet

 Often two solid edges do not intersect in acute or right angles. **Fillet** enables solid edges to be filleted quickly and precisely.

You will first be prompted to select two lines or edges whose intersection is to be filleted. These can also be lines of an object such as a rectangle.

```
> FILLET
Select first line: P1
Second line: P2
```

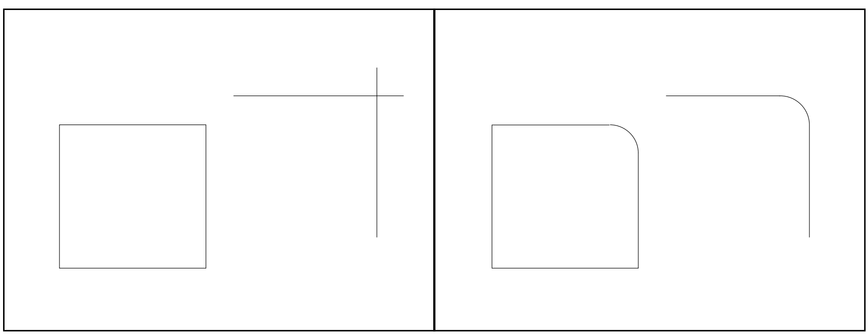
The following options will be shown in the options bar.

Polyline	Radius	No_Trim			

Radius

Radius will define the radius for the fillet. FelixCAD will always offer the value of the last radius used in the current command operation.

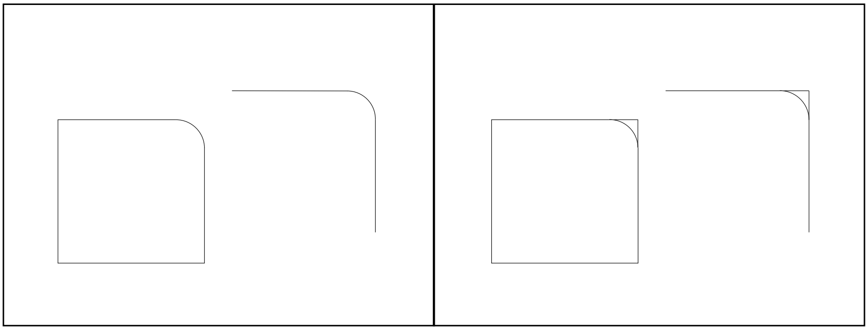
```
> Fillet
Select first line: Radius
Filleting radius <1.0000>: <hit Enter key>
Select first line: P1
Second line: P2
```



No_Trim

Trim functions as a switch. It decides whether the parts of lines which lie beyond the fillet are erased or not. If the switch in the Options Bar displays the term **Trim**, you can **switch off** the function **Trim** by clicking the switch. It will then change to **No_Trim.**

The following picture shows again the effect of the switch **No_Trim.**



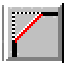
Filleting: Option Trim switched on (left) and Option Trim switched off (right)

Polyline

Polyline allows you to use the fillet function for polylines, which normally are treated as **a single** entity. This command will fillet both sides of the polylines (if the polyline has a width > 0).

```
> FILLET Polyline
Select 2D-polyline: P1
```

Chamfer

 **Chamfer** creates a beveled corner between two intersecting lines.

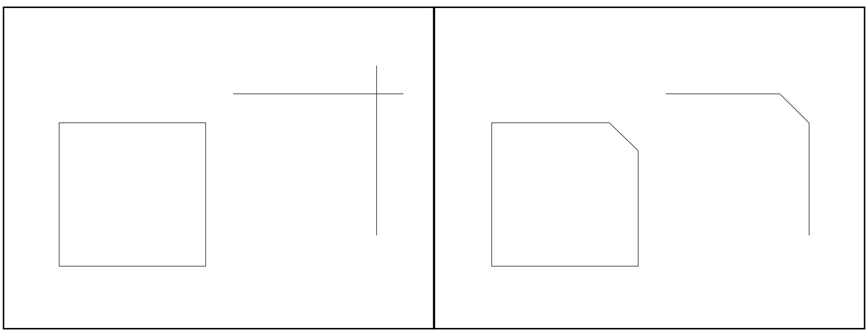
Select the entities to be chamfered by picking them with the cursor. In the Options Bar, the option **Distance** will appear. There are two choices: either select, and enter, a distance for the **Chamfer**, or pick the distance for the **Chamfer**. The distance for the **Chamfer** must be specified for both of the intersecting entities, since there are often non-symmetrical chamfers.

Polyline	Distance	No_Trim			

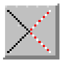
The option **No_Trim**, or **Trim**, functions in exactly the same manner as described with the command FILLET, above, as it does with the option *Polyline*.

```
> CHAMFER
Select first line: P1
Second line: P2
```

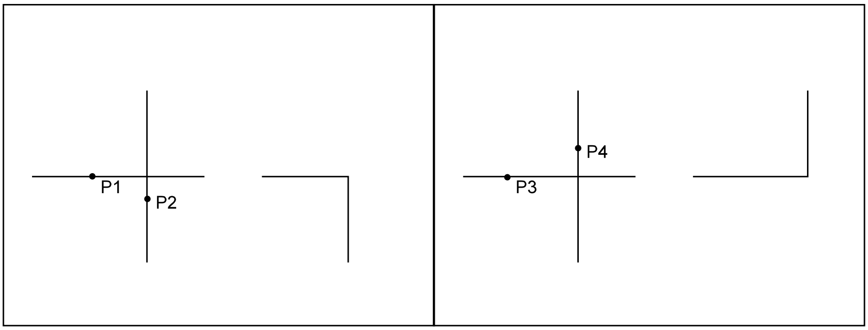
```
> CHAMFER
Select first line: Distance
First chamfering distance <1.0000>: <hit Enter key>
Second chamfering distance <0.5000>: 1.000
Select first line: P1
Second line: P2
```



Intersect

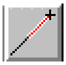
 **Intersect** covers a special case of trimming. It causes intersecting lines to be trimmed following the intersection..

Select the two lines to be trimmed. Make the selection by picking, and be sure to make the pick on the segment of each line that is to remain.




```
> INTERSECT
Select first line: P1
Second line: P2
```

Lengthen

 **Lengthen** allows you to change the length of existing lines or arcs. Unlike the command EXPAND, LENGTHEN extends an entity to a specific value, or in a particular ratio (percentage value), of the original size.

Entities will be only lengthened in their existing direction. All entities will always be lengthened on the end, which lies closest to the point where the cursor was positioned during the selection.

Angle

To lengthen arcs, use the option **Angle**. This option functions like a switch. Selecting it from the option bar has the effect of causing all following selected options are to be related to the included angle of the arc.

Dynamic

The command LENGTHEN normally works in the **Dynamic** mode**.** This mode is active after the selection of a drawing entity and needn't be selected from the option bar. After any lengthening operation the mode will return to **Dynamic,** no matter which of the options has been used.

In the **Dynamic** mode the line or the arc will be lengthened to a point or to an angle indicated by the cursor. A preview will show the result of the command during this operation.

```
> LENGTHEN
Select entity (line, arc): P1
Current length <100.00>: Dynamic P2
Select entity (Line, Arc): <hit Enter key>
```

In addition to the standard mode **Dynamic,** the following modes are available for lengthening lines and arcs:

Total	Percent	Relative			

Total

Total allows the entity to be lengthened to a dimension that is defined. The dimension will apply to either the length of a line or to the circumference of an arc. To assist you the current length of the selected entity is displayed in parentheses.

```
> LENGTHEN
Select entity (line, arc): P1
Current length <100.00>: Total
Total: 200.00
...
```

Percent

Percent will permit the entity to be lengthened by some ratio of the original size. This percentage value relates to either line length or the circumference of an arc. For instance, the input of the percent value 200 doubles the length of the line or the arc, the value 50 diminishes it to the half of the original value.

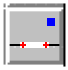
```
> LENGTHEN
Select entity (line, arc): P1
Current length <100.00>: Percent
Percent: 60
...
```

Relative

Relative lengthens a line or an arc by a value which is defined by indicating a distance between two points. This is an advantage if the entity is to be lengthened by a straight line. If the option **Angle** has been selected, determine the angle by which the arc is lengthened by indicating two points in the XY-plane.

```
> LENGTHEN
Select entity (Line, Arc): P1
Current length <10.00>: Relative
Relative: P2
Second point: P3
...
```

Delete Partial

 DELPARTIAL (**Delete Partial**) will cause a portion of a line, circle, or arc to be deleted..

Entities can also be divided by this command. The result of such a division will be two independent entities which can be edited or operated on separately.

The end points for the part to be deleted or divided can be placed freehand on the desired line or polyline, circle or arc.

Deleting Part of a Line

Deleting part of a line is the standard option for this command. To perform this deletion, determine the end points of the part of the line to be erased. The line will be divided at this point, and the portion of the line between the two points will be deleted.

If the entity is a circle**,** that part is separated and deleted which lies clockwise between the first and the second dividing point. A straight segment of a polyline with a width greater than zero will be separated at a right angle to the line.

```
> DELPARTIAL
Select object (line, circle, arc, 2D-polyline): P1
First point: P2
Second point: P3
```

Separating Drawing Entities

To separate or break an entity choose the option **1 Point** from the option bar.

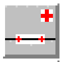
2 Points	1 Point				

Determine the point on the entity where the separation is to occur. It may be wise to use **OSNAP** to increase the precision of the input. The edited object will be cut into two parts which can be edited or operated upon separately.

```
> DELPARTIAL
Selection of an entity
(Line, circle, arc, 2D-Polyline): P1
First point: P2
Break point: P3
```

A circle can not be divided or broken by the input of a point. In this case the program will display a message stating that.

Rejoin

 **Rejoin** will unite separated line segments which remain aligned in one direction, or arc segments which share the same center and the same radius. This function will also reunite separated line or arc segments that have been divided by the command **Delete partial**.

Select the command, then pick the entities to be rejoined.

```
> REJOIN
Select first entity (line, arc): P1
Select second entity (line, arc): P2
```

Change

Change serves to edit entities including lines, circles, arcs, text, attributes and parts with regard to their geometrical character or their attributes.

Note:
To change the properties layer, color, or linetype the function **Modify Properties...** is provided (command PROPEDIT), found in the **Edit** menu.

Select the entity to be edited using the functions shown in the options bar. During this operation you may wish to be supported by the preview provided by **Dragmode**. Select this mode from the options bar. The attributes or geometry that can be changed depends on the entity. In the following paragraphs is a summary of the possible operations, listed by the type of entity.

Dragmode					

Line

One of the two ending points of the line may be moved to a new position, while the length of the line remains the same. If the end point of a line is moved without using **Dragmode**, the point to be moved will be the point nearer the selection point.

If **Dragmode** is activated, the point which lies nearer to the cursor position will be moved. By changing the cursor position the end point of the line to be moved can be changed.

```
> CHANGE
Select entity (Line, circle, arc): P1
Select entity (Line, circle, arc): <hit Enter key>
Modification point: P2
```

Circle

Change reference geometry can only alter the diameter or radius of a **Circle**. Select the object, then determine a point either by picking it or by coordinate input. The diameter or radius will be enlarged or reduced.

```
> CHANGE
Select entity (Line, circle, arc): P1
Select entity (Line, circle, arc): <hit Enter key>
Modification point: P2
```

Arc

Arcs may be changed as were circles.

```
> CHANGE
Select entity (Line, circle, arc,
Text, attribute definition, part): P1
Select entity (Line, circle, arc,
Text, attribute definition, part): RETURN
Modification point: P2
```

Text

These text attributes, or parameters, may be changed. Existing values are displayed as parameters and can be taken over by pressing ENTER.

- Insertion point of the text object
- After the request Modification point:, indicate by picking or entering the coordinates of the block of text.
- Text style
- Enter the name of the new text style or select a text style from the selecting window which will be opened after selecting TextStyle from the options bar.

TextStyle					

- **Text height**
  Define the new height by inputting a numeric value or by “picking” a new height. The current text height will be displayed in the command line area. **Dragmode** is automatically activated during the change of this attribute.
- **Rotation angle**
  Define a changed **Rotation angle** by pointing or by numeric input. **Dragmode** is automatically activated during the change of this attribute.
- **Text string** (character sequence)
  Enter the changed sequence of characters. The current text string is displayed as parameter.

```
> CHANGE
Select entity (Line, circle, arc,
Text, attribute definition, part): P1
 1 selected
Select entity (Line, circle, arc,
Text, attribute definition, part): RETURN
 1 selected
Modification point: P2
Text style <STANDARD> Normal
Height <2>: 1
Rotation angle <0>: 45
Text <first floor>: 1st floor
```

Attribute definition

These attributes can be changed;

- Inserting point;
- Text style;
- Text height; and
- Rotation angle

These are specific characters of those attributes that may lend themselves to change:

- **Name:** Determines the name of the attribute. Enter a new or changed name.
- **Request:** When changing variable attribute values a request is made for a value to be entered. Enter a new or changed sequence of characters.
- **Parameter value:** The constant or variable (parameter) value of an attribute can be changed. The **Value** is that part of the attribute which is

written, together with the part, into the drawing. It may either name the part or contain information about the part. Enter new values (sequence of characters or numeric input).

You may find detailed information to the characters of attributes in Chapter 8, Parts and Attributes.

```
> CHANGE
Select entity (Line, circle, arc, text, attribute
definition, part): P1
Select entity (Line, circle, arc, text, attribute
definition, part): RETURN
Modification point: P2
Text style <STANDARD> Normal
Height <2>: 1
Rotation angle <0>: 45
Text <first floor>: 1st floor
Name <Att003>: RETURN
Request <Index No.>: current number
Parameter value <1.01>: 2.01
```

Part

You may change the insertion point and/or the rotation angle of a part.

The existing values are displayed as parameters and can be repeated by pressing ENTER.

- **Insertion point of a part:** After **Modification point***:* indicate the new insertion point or enter its coordinates.
- **Rotation angle:** Define the new rotation angle by pointing or numeric input.

Dragmode is automatically activated during the change of this attribute.

```
> CHANGE
Select entity (Line, circle, arc, text, attribute
definition, part): P1
Select entity (Line, circle, arc, text, attribute
definition, part): RETURN
Modification point: P2
Rotation angle <0>: 90
```

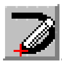
Polyline Editing

Polylines may be modified in the following fashion:

- editing a Polyline as a whole;
- filleting a polyline;
- chamfering a polyline; and
- editing individual polyline segments by editing their control points.

The command may be entered from the menu **Modify**, or may be keyboarded by typing POLYEDIT. If the vertices only are to be edited, that option may be picked from the sub-menu, or keyboarded by typing VTXEDIT.

Edit Entire Polyline



Select the option from the *Detail* menu **Modify Polyline**, or type POLYEDIT. The command allows you to

- open or close the polyline;
- convert lines or arcs into polylines;
- join the polyline with other drawing entities such as line, arc or polyline;
- change the width of the whole polyline; or
- undo the last step.

If you have selected a line or arc, you will be asked if you wish to transform it into a polyline.

Yes	No				

Select the desired option by clicking the Option Bar or entering the first character of the desired option.

Open	Join	Width			

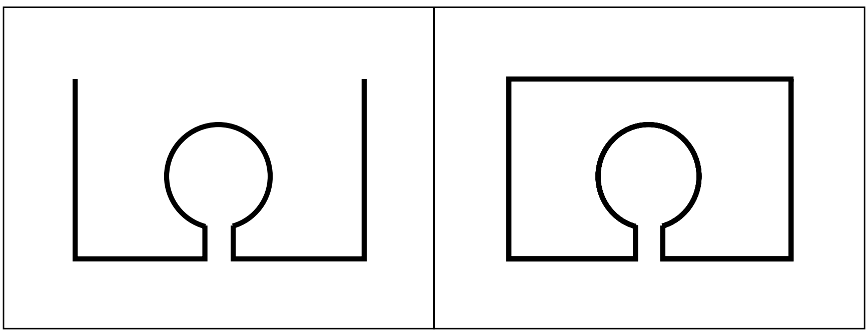
The first option panel will display the Option **Open** or **Close,** depending on whether the selected polyline is currently opened or closed. Note that the option **Back** is only available after the execution of the first operation under the command.

Close	Join	Width	Back		

Open / Close

Close is an option already available during the drawing of a polyline. This option joins the start and end points of the polyline with a straight polyline segment. The width of this segment equals the ending width of the end point of the last drawn polyline.

Open is a possible option if the polyline to be edited has already been closed during drawing, or in a successive working step. Select the Open option. The polyline segment between the start and end points of the polyline will be deleted.



Open (left) and closed polyline

```
> POLYEDIT
Selection of a 2D-Polyline: P1
Input: Close
Input: RETURN
```

Join

Join allows lines, arcs and other polylines to be joined with the selected polyline. The prerequisite to this is that the polyline to be edited and the object to be joined meet in one ending point.

Indicate, after calling the option **Join,** the entities to be joined.

```
> POLYEDIT
Select 2D-Polyline: P1
Input: Join
Selection of an entity (line, arc, 2D-polyline): P2
Selection of an entity (line, arc, 2D-polyline): RETURN
```

The polylines chosen are now joined as one entity.

The width of the new polyline segment that has been created depends on the character of the entity which has been joined to the polyline. If lines or arcs are joined to the polyline, the new polyline segment will have the same width as the polyline segment at the end at which the line or the arc has been joined.

If polylines are joined, the ending width of the polyline segment added will be the starting width of the “parent” segment. If the polyline to be joined is a polyline with a uniform width the ending width of the first selected polyline will be the new width of the segment being joined.

Width

Width allows you the opportunity to change the width of the polyline. Polylines which have different start and end widths as single segments can be made a uniform width for all segments.

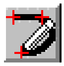
Define the desired width of the polylines by numeric input or indicating a width with the cursor. As a “help” parameter for this input, the value used last for the width of the polyline segment to be joined will be displayed. This value may be used by pressing ENTER.

```
> POLYEDIT
Select a 2D-polyline: P1
Input: Width
New width of the polyline <1.5000>: 2
Input: RETURN
```

Undo

Undo will remove the last operation. The function **Modify Polyline** is continued, and you will see the request **Input:**.

Vertex Editing



Editing single segments of a polyline is done through the command **Vertex Editing** (command VTXEDIT). This command allows you to change the vertices, the start and end points, of the polyline. The vertices of a polyline segment can be:

- inserted;
- deleted;
- moved; and
- the width of the polyline segments can be changed.

Select the function by selecting the option **Vertex Editing** in the sub-menu **Modify Polyline** of the menu **Edit** or by typing the command VTXEDIT.

Selecting the Vertex

Select the polyline containing the vertex to be edited. The selection of the polyline will also cause the selection of the vertex containing the start point as the first vertex to be edited.

The vertex selection may be changed by stepping through the vertices using the **Next** and **Previous**. You may want to experiment with these selections to understand thoroughly how they work. The selected control point will always be marked by a rubberband line joining the selected point with the current cursor position.

After selecting the control point to be edited, select one of the editing options from the options bar:

Next	Previous	Insert	Move	Delete	Width

Insert

Insert allows the addition of vertices, and thereby of further segments, to an existing polyline.

Select the option, then define the position of the additional vertex.

The width of this new segment will correspond to the values for the width of the polyline at the bordering control points before the addition of the new segment.

After inserting a segment the command VTXEDIT stays active. You may select another operation or complete the command by pressing ESC.

```
> VTXEDIT
Select a 2D Polyline: P1
Next: <hit Enter key>
Next: Insert
New control point: P2
Next: <ESC>
```

Move

Move allows you to relocate a vertex of the polyline to a new position.

First determine the vertex to be relocated using the same technique as above, then define the new position of the chosen vertex. The vertex will be moved to this position. Other segments of the polyline will remain unchanged.

The command will remain active. Finish editing the polyline segments by pressing ESC.

```
> VTXEDIT
Select a 2D Polyline: P1
Next: <hit Enter key>
Next: Move
New Position: P2
Next: <ESC>
```

Delete

Delete erases a selected control point from the selected polyline.

Define the vertex to be deleted , then select the option **Delete** from the options bar.

The segment which has its starting point in the selected vertex will be deleted. The previous (in order of drawing) segment will be lengthened to the next control point.

The command VTXEDIT remains active. Select another option or finish editing the polyline segments by pressing ESC.

```
> VTXEDIT
Selection of a 2D Polyline: P1
Next: <hit Enter key>
Next: Delete
Next: <ESC>
```

Width

Width allows the width of single segments of a polyline group to be changed.

Define the vertex to be modified, then select the option **Width** from the Options Bar. Input or select with the cursor the desired width of the polyline segment. The width of the selected segment will be changed according to the inputs.

The command VTXEDIT remains active. Select a another operation or finish editing the polyline segments by pressing ESC.

```
> VTXEDIT
Select a 2D Polyline: P1
Next: <hit Enter key>
Next: Width
Starting width <0.50>: 1
Ending width <1.00>: 2
Next: <ESC>
```

Polyline to B-Spline

This function fits a B-splined curve to the chosen polyline. Choose it from the Detail Menu - Modify Polyline or enter SPLINE. The curve is not forced to pass exactly through the polyline vertices as in the Curved Polyline described below, but will produce a smoother curve fit.

```
>SPLINE
Select polyline to alter to B-spline:P1
Number of segments for approximation (2 to 1024) <8>:
```

8					

You can turn the Polyline back into its original form using the **Decurve B-Spline** command on the same menu or enter SPLINE D.

2-D Polyline to Curved Polyline

This function fits a curve to the chosen polyline. Choose it from the Detail Menu - Modify Polyline or enter PCURVE. The curve is forced to pass exactly through the polyline vertexes, and the degree of fit can be made smoother with the number of steps and tension options. The tension is similar to pulling on the ends of a string constrained to pass through the polyline vertices.

```
>PCURVE
Select 2D-polyline to alter to curved polyline:P1
Number of steps for interpolation points (0...6) <3>:
Tension (0.0 ... 1.0) <0.5000>:
```

You can turn the Polyline back into its original form using the **Decurve Curved Polyline** command on the same menu or enter PCURVE and select the Decurve option from the bar or by typing D.

Decurve					

Chapter 7

Text Objects

The functions described in this chapter are used for inserting text into drawings, as well as for changing text entities. They provide labeling, create legends, descriptive notes, and other text elements in a drawing. The commands are found in the *Detail* menu.

Properties of a Text Entity

A text entity is defined by the following properties. These properties are assigned during the creation of the text entity. Any of them may be changed after the creation of the original entity. They are:

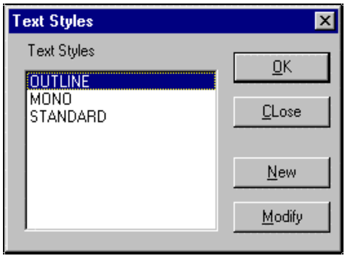
- a **character string**, which represents the content of a text object. All characters or special characters may be used for this string;
- the **font** (type face) - the program contains a series of standard fonts, that can be used in creating text objects. Font files are marked by the extension **.fsh**;
- the **text size** (height of characters) defined in drawing;
- the **alignment** of the text objects - determines the horizontal and vertical position of the text string relative to the insertion point of the text object. The character string can run right or left from the insertion point, or the insertion point may be situated centered. The vertical alignment of the text objects may be above, center, base line, or below. Any text object can be rotated during insertion as well; and
- the **insertion point** of the object. The insertion point (and the alignment) of a text object determines its position in the drawing. The insertion point can be defined by coordinate input or by selecting the point with the cursor.

All of the parameters discussed above can be set in the text dialog box.

Fonts

The procedures for managing and defining the properties of fonts are described in this section. Procedures are described for new fonts in the font selection list, editing and saving as font styles including the height, width factor and inclination angle, as well as alignment, of individual text fonts.

Fonts... allows you to load fonts from the font files and customize them to the requirements of the drawing. Call the option **Fonts...** from the **Detail** menu or type FONT at the command line. The Dialog Box will display available font types, and offer a **New** option so that fonts may be added.

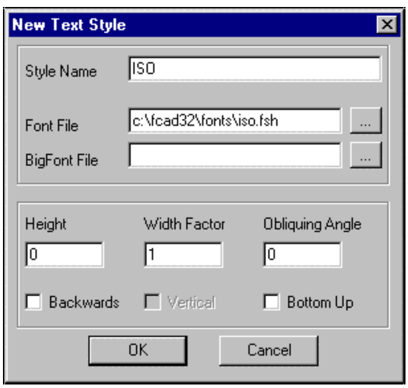


Adding or Changing Fonts

Selecting an existing font and making changes in it or adding a new font are done in the same way and are, therefore, discussed as one operation.

New Font

To add a new typeface open the **Text Style** dialog box and select **New.** The **New Text Style** dialog box will open. Define the new font style, and confirm with OK. If the font is located in another file, select the **Find...** button from dialog box. The dialog box called **Open Font File** will open. Enter the name of the desired font file in the input field Font File or select a file using the Find... function.



Changing Properties of the Font

After the new font has been selected its properties can be edited using the **New Text Style** dialog box. If the option **Modify** was selected after calling FONT, these additional editing steps may be used for existing font.

Note
that some type fonts do not support all of these options. For instance, some fonts can be aligned vertically. On others, there is no change in aspect ratio permitted.

- **Height** (text size) is expressed in drawing units. Entering a numeric value for *Height* gives the font a **fixed** height which can no longer be changed when creating a text object using this font. If change in the font height may be required, you should keep the height parameter at a 0 value. The font height may then be defined individually in the *Text* dialog box.
- **Width factor** controls the width of the characters in relation to the height. This ratio is often called the Aspect Ratio.
- **Obliquing angle** defines the inclination of the characters in relation to a horizontal (X axis) line. Input values between 0 and 180 result in an inclination to the right (forward), values above 180 to an inclination to the left (backward). Practically, values between 0 and 30 or between 180 and 210 are meaningful.
- **Backward**, when selected, will produce a mirror-image display of the text.
- **Vertical**, when selected, will product characters aligned vertically.
- **Bottom Up**, when activated, will produce characters displayed upside down.

It is possible for you to combine these settings, although in some cases the combination of properties may interfere with each other.Fonts added or changed in this manner can now be used for creating or editing text objects. Defined fonts are saved with the drawing file and are available any time that the drawing is opened.

Font Conversion Table

A file FONTS.TAB in the configuration directory allows you to create a font association table for opening files with the OPEN command.

The following is an example of FONTS.TAB file

```
[FONTTAB]
complex=apcomplx
romanc=apcomplx
iso=apnorm
helvetica=c:\felixcad\fonts\apswiss
```

Currently [FONTTAB] is the only recognized section in the FONTS.TAB file. Any line in the FONTTAB section has the following syntax:

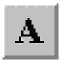
source_font_filename=*apply_fcad_font_filename*

The *source_font_filename* must always be specified without filename extension.

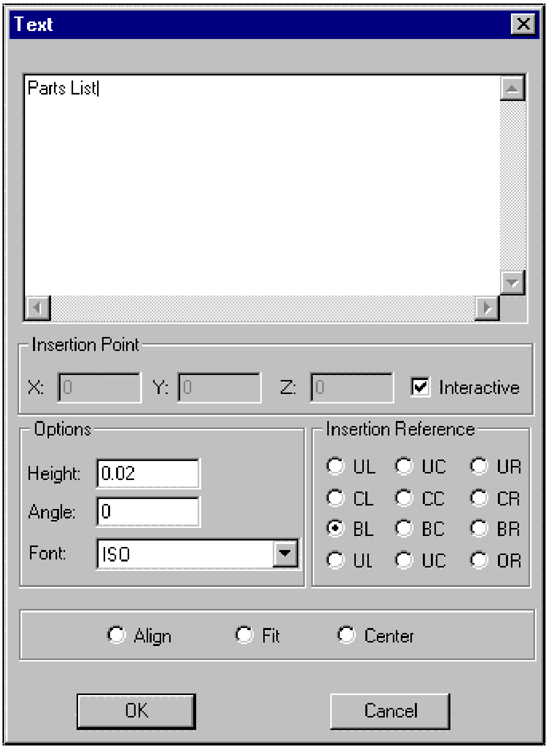
The *apply_fcad_font_filename* must always be specified without filename extension, but can have a path specifiction.

The program will search for an FSH file specified by *apply_fcad_font_filename* in the application directories. If that FSH file is found, it will be applied automatically; otherwise the program will request you to specify a FSH file to use for replacing a non-corresponding font file.

Placing Text



The command TEXT allows you to add information to the drawing. The following dialog box will be opened.



Text Input

Text is entered into the edit box in the upper area of the dialog box. The cursor will be positioned in this field upon the opening of the box, so you may start the input of the text immediately.

The text editor in this dialog box offers the basic functions for editing text. Incorrectly entered text can be marked and erased just as with a word processor.

When a line or paragraph of text is complete, press ENTER.

Insertion point

There are two methods of defining the insertion point. The insertion point may be entered using (absolute) coordinates (edit boxes are available for inputting the X-Y- and Z-coordinates). The second option is to define the insertion point interactively. The option **Interactive** must be checked to activate this option.

Options

These options will aid you in determining the height (point size), the rotation and the font (typeface) of the text object to be inserted.

Height

Define the text height in drawing units using the input field **Height**.

Angle

Define the insertion angle of the text with this input field**.** Any positive or negative value between 0 and 360 is possible. This will also align the text to vertical or sloping objects. The default value is a rotation angle of 0 degrees: the text object is inserted horizontally.

Font

A drop-down list allows you to set the desired font used for the text. The list of available fonts is displayed in this drop-down list.

Alignment

The alignment of the text object is specified by using the options Align, Fit and Center.

Position

Defining an **insertion reference** point determines the position and the alignment of the text object in relation to the insertion point of the text object. A reference point on the text object is defined. During insertion of the text object this reference point is coordinated with the insertion point.

If the reference point is, for instance, situated at the lower left corner of the text object, the text will run to the right.

In total, there are twelve reference points available for your choice. They are represented by the initials of the locating corner (or center) of the text block. UL, for instance, represents the Upper Left corner of the block. Select the position for the reference point by filling the circle.

Align

Align allows text objects to be rotated and scaled during insertion. The rotation angle and the scaling factor are defined by indicating two points on the drawing. These are the procedures for using **Align**:

- Activate the option by clicking the **Align** button at the lower left side of the Text Dialog Box. The options **insertion point**, **text height**, **angle** and **position** will no longer be available. The parameters required for alignment result from the definition of two points during the insertion of the text object. The reference point of the text object is positioned left justified, lower corner;
- Confirm **Align** by clicking OK;
- Enter the first point for insertion of the text object. This point is interpreted by the program as both the insertion point and the first point for the rotation angle and scaling factor;
- Enter the second point, which completes the definition of the rotation angle and the scaling factor, and sets the text height.

Fit

Fit enables the simultaneous rotation and expansion of a text object during the insertion. The text object is not scaled, that is, the text height remains unchanged, but the string is expanded or compressed to fit. The procedure is:

- activate **Fit** by filling the circle at the lower left side of the Dialog Box;
- confirm **Fit** after inputting the text string by clicking OK;
- the prompt will be for the first point - it may be specified or picked - do this; then
- the prompt will be for the second point - specify or pick this point.

The text will be fit between the two points, rotated so that it follows the line between the points.

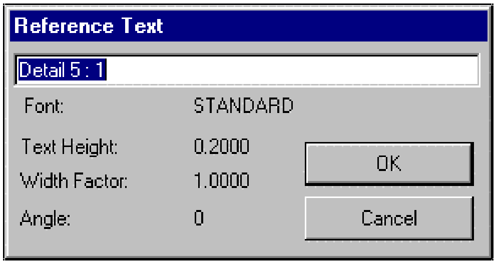
Center

The option **Center** enables you to insert a text with both vertical and horizontal alignment centered. Text height and rotation angle may be chosen.

Reference Text

The **Reference Text** utility (command RTEXT) enters new text in a dialog box so that it matches the style and attributes of previously entered text.

Simply pick a text entity and overwrite the string in the dialog box.

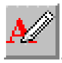


Then specify the location of the new text entity in the drawing.

The new text will have the same font, height, width factor, rotation angle and layer as displayed in the dialog:

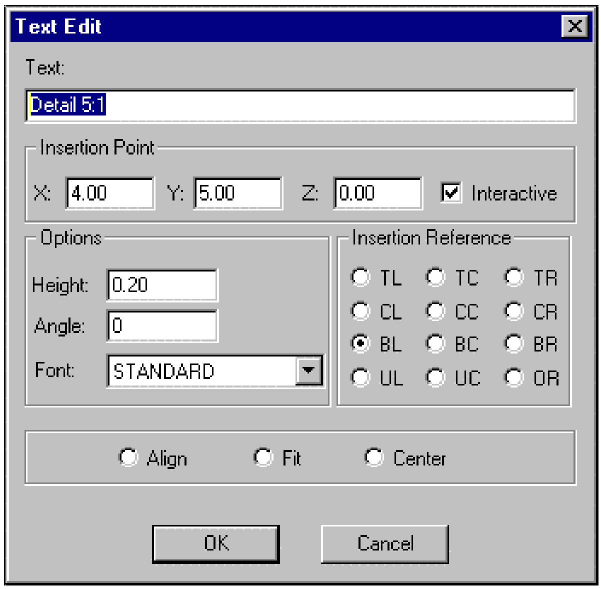
```
> RTEXT
Select reference text: P1
<Dialog box to edit text>
Insertion point: P2
```

Modify Text

 You can modify an existing text line in the drawing by selecting **Modify Text …** from the **Detail Menu** or entering the TEXTEDIT command.

Select an text entity in the drawing. Immediately, the dialog box displayed below will appear. The dialog box should be filled out with the parameters required by the new entity

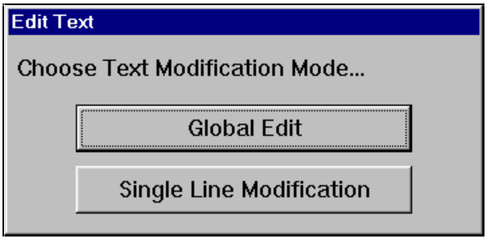
If the check box labeled **Interactive** is marked, you will be prompted to relocate the text when exiting the dialog. Otherwise, the insertion point coordinates in the dialog box may be changed if desired.



Correct Text

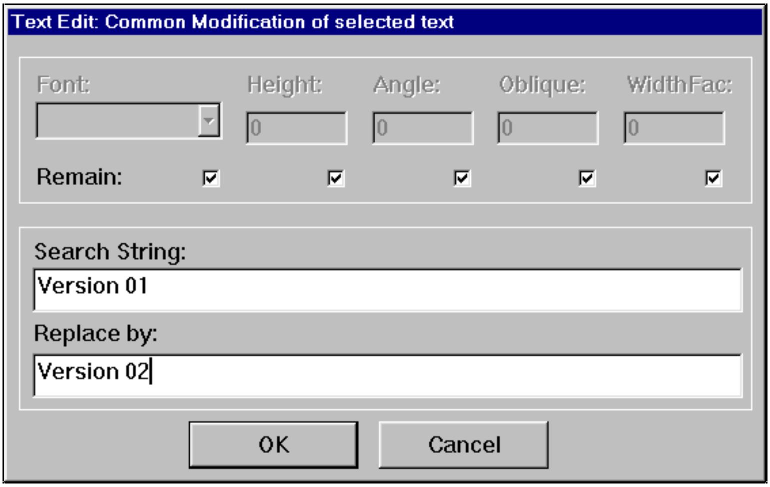
Correct Text allows you to edit text entities and attribute definitions. You may correct texts or text attributes as well as changing the font or the point size. Call the function by entering TCORRECT or select **Correct Text..**. from the **Detail** menu.

The first prompt will request you to select the text lines or attribute definitions. Use the usual object select functions. Finish the object selection by ENTER. Incorporate the desired corrections in all selected line globally, or line by line. Choose an option from this Dialog Box:



Global Edit

The Dialog Box for global editing of selected texts and attribute definitions is the following:



In the upper area all of the text attributes which can be changed are listed. Using the marking array **Remain,** enter, for any text attribute, whether this attribute is to be changed globally or should remain unchanged. If you switch off the box below the Text Height, for instance, you may enter a new value for the text height. All selected text lines will be changed to this text height.

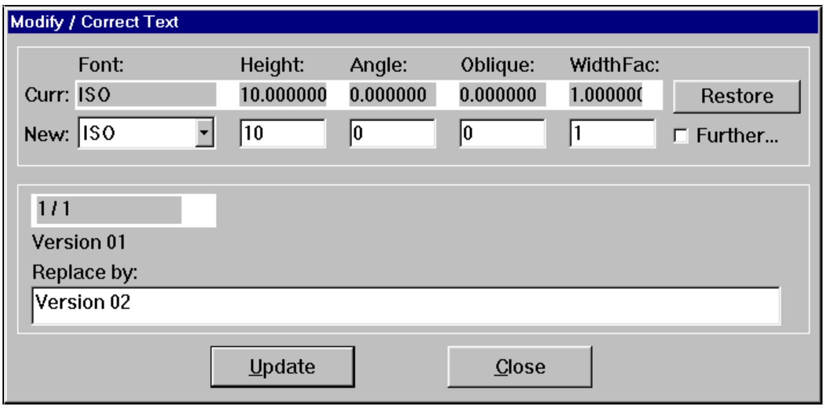
The lower part is a search function. Enter a string in the line **Search-String**, indicating what name should be located and what string should replace it. The program will then replace that Search-String by the new string.

Single Line Modification

Single Line Modification enables you to change the same text attributes.

In the line **Curr:** actual values are displayed. Enter the new values in the corresponding fields of the line New. Checking the box **Further...** will input the new values you have selected. The **Restore** button restores the previous text string.

When you leave the dialog box by clicking to **Update**, the changes are executed. The button **Close** ignores all changes made, and asks whether you want to end the text correction or to continue with the next line.



Chapter 8

Parts and Attributes / Groups

Parts and Attributes are explained in detail in this chapter. The chapter is divided into six basic sections, each dealing with a particular feature of **Parts and Attributes**.

This chapter covers definition, creating, inserting, removing, and managing **Parts**. It covers creating and editing **Attributes**.

Drawing entities can be combined in **Groups**. Grouping entities allows the user to manipulate all entities within the Group in a single operation (Moving, Rotating, Mirroring, or Scaling, for example). Single entities of a Group may still be *edited* individually (for example, intersecting or stretching). Entities may be removed or added to a Group at any time.

The main differences between ‘Parts’ and ‘Groups’ are:

A Part (a block) has its own insertion point. A Group has none. A Part can occur multiple times within the drawing. If a block is redefined, all of its occurrences are updated. Groups are not treated like that. Groups can, however, be copied. A Group that originates from another Group that has been copied is a separate union of entities. Each Group has a unique name.

Parts - and the Advantage of Working With Them

Parts allows you to create a drawing using repetitive entities. These entities may contain single objects, groups of objects, partial drawings or complete drawings, among other possibilities. These entities are inserted into a drawing as a “whole” entity, and not a ‘group” of single entities.
Many times, you will find that existing parts, construction groups or aggregates have only to be designed once. Repetitive use of these parts, or entities, allows you to create a drawing quickly.

To define or insert Parts, use the menu *Part*.

Properties of Parts

A Part is a complex object which consists of several drawing entities or objects. Examples of those objects can be:

- basic entities or objects, such as lines, circles, arcs, etc.;
- drawing entities or objects with hatches, dimensioning or details;
- complete drawings; and
- drawings already defined as Parts.

After an entity or group of entities is selected and defined as a Part, it is saved under a part name and treated as **one single entity** for future operations.

Editing, Changing, and Deleting always will always effect the defined part as a whole. A special command (**Explode Complex Objects**), found in the *Edit* menu, allows a Part to be dismantled (exploded) into its original components.

A part can be inserted in an existing drawing by selecting **Insert Part...** from the menu, or by typing the command INSERT. This command will facilitate changes in the scale and the rotation angle so that the part may be “fitted” to the current drawing. When the part is inserted into the drawing, all of its defined properties (layer, color, line type) will be inserted with it.

The part will be inserted on a layer, as it was defined. If the part contains another layer (for instance, if the part is a group of parts, each on its same layer) which does not exist in the current drawing, that (those) layer(s) will be created in the current drawing during the insertion of the part. The exception to this rule occurs when entities of a part are situated on **layer 0**. These entities will automatically be inserted on the current layer.

Internal and External Parts

Internal and External Parts are treated differently with regard to storage location and with regard to relationship with the current drawing.

Internal Parts are parts which have been created and stored in the current drawing. They are *internal* to the current drawing. They may be reused in the current drawing, but are not available for use in other drawings without an additional operation.

The command **Write Part File...**, found in the menu or entered by typing PARTEXP, will allow the part(s) to be written in a separate file or put into a library.

External Parts, on the other hand, are created and stored as separate drawing files. They may be used independent of the current drawing. It is good management to “group” these parts with functionally similar parts in a library, and name that library so that it may be used repeatedly.

By building these libraries, you will facilitate her or his drawing tasks greatly.

Attributedefinitions Added to Parts

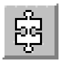
A part can be given **Attributes**. Whereas the drawing of the part may adequately describe its physical outline or dimensions, **Attributes** are used to describe other non-graphical or physical characteristics.

Attributes may be displayed, or may be hidden values. For example, a part made of 4041 steel may have any definable physical dimensions, but there is no way to display the material from which the piece is made other than on the Bill of Materials, or by a drawing note. Assignment of the material as an **Attribute** does this.

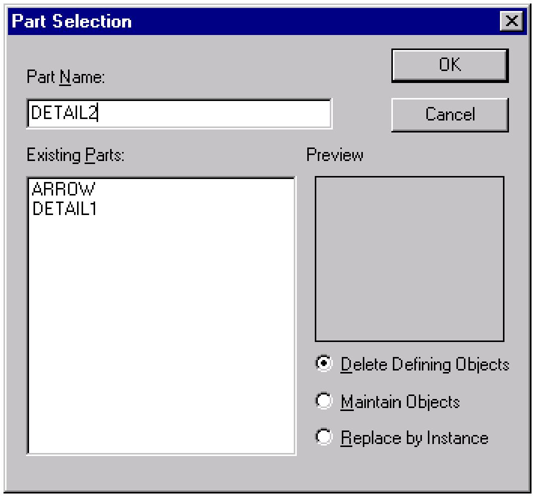
Creating Internal Parts

A **Part** is created by selecting the object, entity, or group of entities that represent the part; giving the part a name; and selecting an insertion point for the part.

Defining the Part



Select the function **Define Part...** from the menu, or enter the command PARTDEF via the keyboard. The following dialog box will open:



Part Name

Names of parts can contain up to 31 characters. In addition to letters and numbers, special characters **$**, **-** , and **_** are allowed.
Enter the name of the part in the appropriate field of the dialog box.

Existing defined parts are listed in the box labeled *Existing Parts*. If one of those parts is to be redefined, select the part from the list. An alert box will appear, warning that you are about to redefine that part. The message will give you the option of choosing a new name for the part about to be defined, or of redefining the existing part.

The radio button column in the dialog box allows you to determine how to treat the objects selected to define an internal part when the definition has been finished. There are three choices:

- Delete Defining Objects
- Maintain Objects
- Replace by Instance

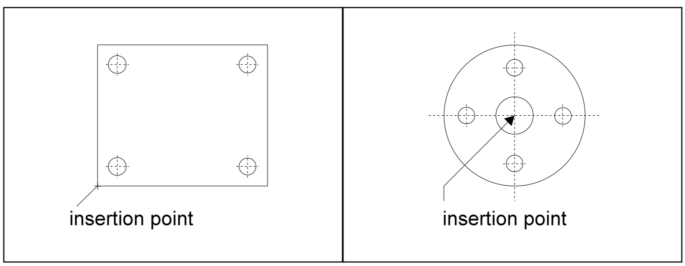
Usually, once you have created a part definition the defining objects are not used any more in the drawing, because the purpose of the internal part just created is to be inserted at different locations in the drawing. If you want the defining objects to be removed from the drawing, check the *Delete Defining Objects* radio button. This is the default setting in the dialog box. Note, that you can restore the defining objects (once the part definition has been terminated) by calling the UNDELETE command.

If you want to leave the defining objects in the current drawing as they are (for example, for further modification and subsequent definitions of similar parts with the same attribute definitions), check the *Maintain Objects* radio button.

If you wish to leave a copy of the block at the current location in the drawing, check the *Replace by Instance* radio button.

Insertion Point

Each part must have a defined insertion point. The insertion point is a reference point that enables you to position the part properly during insertion. The insertion point of a part also serves as the basepoint for changing the scale and the rotation angle during the insertion.



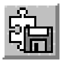
Any point on - or even off - the part may be selected as the insertion point. Typically, users will select the left lower edge, the center of an object, or some other discreet point of the geometry as the insertion point. It is a good idea to use the **Object Snap** functions for this selection to insure precision.

Selecting the Part Objects

Next, you must select the object or objects that will make up this part. The Option Bar furnishes the object selection options.

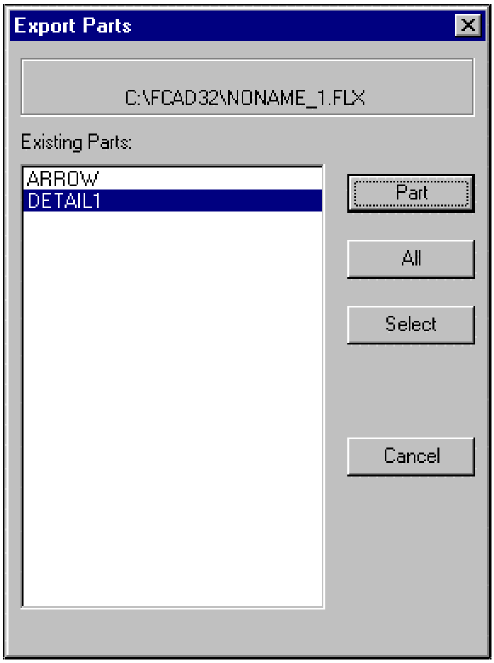
An internal part has now been created. You may wish to experiment a bit by now attempting to place the part on a drawing.

Creating Part Files, External Parts

 External Part Files are created using the export function **Write Part File...**, found in the menu, or typing the command PARTEXP. The definition of external parts is very similar to the procedure for defining internal parts.

After entering the command, the dialog box will appear. First, determine the name of the external part, its storage location (drive, directory), and confirm these inputs.

The dialog box **Export Parts** is used to define the part. An external part can consist of single entities, groups of entities, existing internal parts, or even the complete current drawing.



File Type Specification

In addition to save the part drawing in the native FLX file format, you can utilize the Part Export function to save internal blocks or selected entities as DWG or DXF file. To do so, choose from the File Type list the desired file format and specify the name of the file.

Difference to Save As: The Purge Effect

The PARTEXP command can also be used to save the entire drawing. This is similar to the SAVEAS command, but with one main difference. PARTEXP will **purge** all unused block references and unused symbol table entries (layer, linetypes, text styles, etc.) when the drawing is saved. By this, the command can help to get rid of unreferenced entries in the drawing database and free it from unneccessary ballast.

Drawing Entities as External Parts

To save single drawing entities as External Parts click **Select**. Then proceed exactly as was described with the Internal Part.

- determine the insertion point of the **Part**;
- select all of the entities which belong to the new **External Part**.

The selected drawing entities will be erased from the original drawing when the external part is saved. To avoid erasure, invoke the UNDELETE command.

To Save an Internal Part as an External Part

The list of Internal Parts is displayed in the list box of the dialog. Select the desired part to be saved, and select the button **Part**. The marked part will be saved externally, along with all of its attributes and its insertion point.

To Save the Entire Drawing as an External Part

In the dialog box *Export Parts,* select the button **All**. The complete drawing will be saved as an part file, along with all of the attributes, parts, entities, loaded fonts, linetypes, layers, and so forth.

If the complete drawing is saved as an external part, the drawing entities are **not** erased from the drawing.

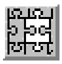
Inserting Parts

Any Part that is defined can always be inserted in a current drawing. Four commands are available to do this:

- **Insert Part...** (command INSERT); and
- **Insert Quick...** (command QINSERT).
- **Paste Internal Part (1:1)...** (command INSERTQ).
- **Merge External Part (1:1)...** (command MERGE).

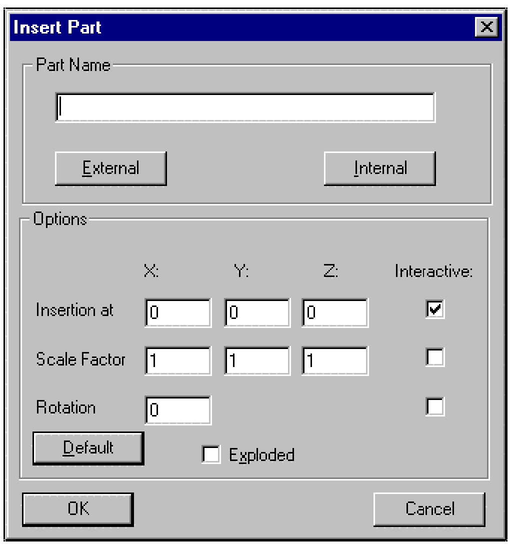
The choice between these commands is up to you. If one or the other of the Dialog boxes gives you a greater degree of comfort, that is the command that should be used.

Insert Part...


The command INSERT offers broad support for the user in its dialog box. This dialog box contains several selections that are not contained in the dialog box used by QINSERT.

The **Insert Part...** dialog box is shown below. Many selections are offered. There are selection areas for:

- selection of **external parts**;
- definition of the **insertion point**, scale factors, and rotation angle, with future choices of using coordinate/ numeric input or cursor selection;
- all parameter values may be reset; and
- the **Part** may be "exploded" into its components.

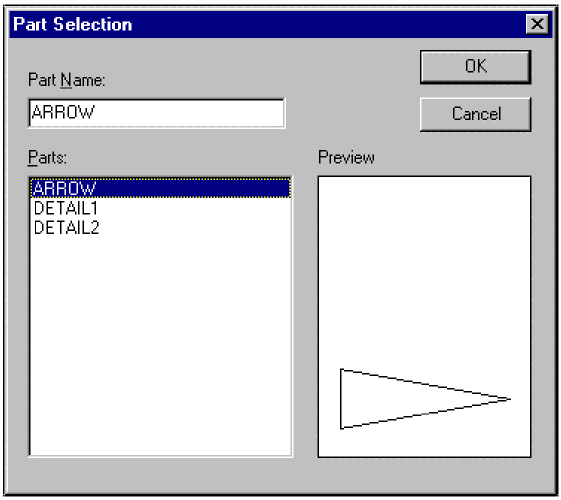


The option *External* allows you not only to insert FelixCAD drawings, but also DWG and DXF files into the current drawing. The dropdown-list *File Type* of the dialog box to insert an external file as part (block) contains beneath the FLX file format the file types

- AutoCAD drawing (*.dwg)
- DXF file (*.dxf)

Selecting a Part

To select the part to be inserted, type the name of the part in the input area *Part Name* or select an existing part from the list by using the buttons *External* and *Internal*.



Defining the Insertion Point

Define the Insertion Point either by entering the proper coordinate values, or by selecting Interactive, and “picking” the point with the cursor.

Scale Factor

Scale Factor allows you to scale the part during the insertion. Enter the Scale Factor with a discreet number value in the input box, or select, again, *Interactive*, and then define the window into which the part is to be fit.

The input areas for Scale Factor will offer the factor 1.0 as the Scale Factor. This will insert the part at its original size. Remember that a Scale Factor greater than 1.0 will increase the size of the part, and a factor less than 1.0 will reduce the size of the part.

Interactive allows you to define the size of the part during the insertion. To insert the part using the *Interactive* option, simply define the rectangle into which the part is to fit. Dragmode is active during this operation, so you will have the benefit of seeing the selection results before confirming the selection.

Rotation Angle

The *Rotation Angle* can be set from the dialog box. Enter a value for the Rotation Angle in the proper input area. The input area will always display an angle of 0 degrees as the default value. For counter-clockwise rotation enter a positive value, for clockwise rotation enter a negative value between 0 and 360 degrees.

Again, *Interactive* enables you to define the rotation angle using the cursor during the inserting operation. As with the insertion option, dragmode is active during this form of rotation.

Default

Default allows you to reset the parameters for the insertion point, the scaling factor and the rotation angle back to the original parameter values.

Exploded

Exploded allows you to “explode” the part into its component entities. Exploding will cause the loss of the characteristics that were retained as a part.

Quick Insert

Quick Insert..., or the command QINSERT, is very effective for inserting internal parts from the current drawing quickly. It offers a reduced selection of options for change, and requires knowledge of the part names and of the storage location of the parts (for external parts).

The dialog box for this selection contains the **Part** selection field with a list of the internal parts, and a field for the part name that is to be entered.

Select or enter (if an external part) the part name, and simply enter the required prompts.

The first prompt will ask for the Insertion Point. Enter the value, or pick the point with the cursor. Next, the prompt will ask for the X factor. It is asking for a scaling factor. Remember that a factor greater than 1 will enlarge, less than 1 will reduce. After the scaling Factor is entered, the prompt will request the Y Factor. Enter this as well, or press Enter to keep the X and Y factors the same.

The selection of the factors can be facilitated by selecting **Factor** from the option bar. Several options will appear.

X=Y=Z=1	Corner	XYZ			

X=Y=Z=1

This option is the default option for inserting parts. This option inserts the parts in original size (scaling factor 1).

Corner

This option allows you to accomplish a scale change by pointing to the drawing area. The part to be inserted can be easily customized, interactively, to existing drawing entities. Indicate the opposite corners of the rectangular window within which the part is to be scaled. **Dragmode** will support this operation and show the window to be defined as well as a preview of the part in the selected scale.

XYZ

This option enables you to input different scaling factors for parts. The scaling factor for the X, Y, and Z-axis are each requested.

Rotation Angle

Finally, the **Rotation Angle** is requested. Remember that a counter-clockwise rotation requires a positive value, a clockwise rotation requires a negative value between 0 and 360 degree. This angle can also be specified using the cursor to pick the value, and, once again, the benefit of **Dragmode** will accrue to the user, presenting a preview of the location as the cursor moves.

```
> QINSERT
Partname <part01>: <hit Enter key>
Insertion point: 3,3
X factor <=1>/Corner/XYZ: <hit Enter key>
Rotation angle <0>: <hit Enter key>
```

If you choose to call the command QINSERT by keyboarding, the Part Selection Dialog box will not appear. Instead, you should simply follow the series of prompts as they are offered.

External Parts may be inserted by QINSERT or by selecting **Quick Insert**. You must enter the complete path to the Part in this case: (drive\directory\subdirectory\partname including file extension).

Paste Internal Part (1:1)...

This command is useful for inserting an internal part quickly.

The dialog box for this selection contains the **Part** selection field with a list of the internal parts, and a field for the part name that is to be entered. Select or enter (if an external part) the part name.

The next prompt will ask for the Insertion Point. Enter the value, or pick the point with the cursor. It offers the following options:

Factor	Rotation				

See the instructions for Quick Insert for a description of these options.
The part is then inserted.

Merge External Part (1:1)...

This command is used for inserting an external part quickly.

The dialog box for this selection allows you to select a part file from disk. Select or enter the part name.

The next prompt will ask for the Insertion Point. Enter the value, or pick the point with the cursor. It offers the following options:

Factor	Rotation				

See the instructions for Quick Insert for a description of these options.
The part is then inserted.

Preload Parts (PINSERT command)

The command PINSERT incorporates an external block into the current drawing without creating an instance of the block in the drawing. This variant can be seen as discrete insertion.

The main purpose of this command is

- to preload parts (blocks) for later use
- to load table definitions defined in a template drawing (layers, linetypes, text styles, dimension styles, etc.)

The only request of the command is to specify the *file name* of the external part be be brought in, but not actually be inserted into the drawing. Only the symbol tables will be enhanced by those (additional) entries found in the descrete inserted drawing. This command is provided especially to be used in the programming interfaces.

Exploding Parts

Parts inserted in the drawing constitute **one single drawing object**. It is not possible to edit any single one of the entities that make up the **Part** while in this mode,. If, for any reason, there is a requirement to edit one of the entities comprising a **Part**, then the **Part** must be **“Exploded”** (broken into its individual entities). In addition to exploding parts, this command is used to explode polylines, crosshatches, and dimensioning.

Select **Explode Complex Objects** in the *Edit* menu, or enter XPLODE in the command line and a dialog box appears to select the type of objects to explode, (entering the command EXPLODE will skip this dialog and go directly to object selection of parts for exploding of all types.)

After selecting which types of objects to explode, continue with object selection. When the object selection is complete, the parts (polylines, crosshatches, or dimensions) will be exploded.

Now you may choose any of the entities for editing. It is important to realize that, if there was a variable attribute attached to the part, this attribute has been lost. Instead, an attribute name will appear.

Part/Symbol Library

 Part Libraries allow you to manage drawing files. These files may be project drawings, template drawings, part files, or symbol files.

A **library** is defined by a **parameter file**. The drawing files are assembled in a directory where the **library parameter file** is located.

Select **Part/Symbol Library** from the Part Menu, or enter the command LIBRARY, and a dialog box will be displayed allowing you to select a Part library (.PLB) file. Or enter the command PARTLIB followed by the name of a .PLB file (for example, c:\felixcad\drawing.plb.) Once a part library has been chosen, the Library command continues to use that same library until the **Set Library…** command is chosen, or you enter the command SETLIB.

The program displays a dialog box with a preview bitmap of each drawing in the library to select a drawing to be inserted. Within this dialog box you will be able to preview the drawing and information about the drawing.

There are two modes for the list box:

- In the **Info mode** the drawings are displayed together with the textual information;
- In the **Quick mode** the drawings are displayed only with the bitmap.

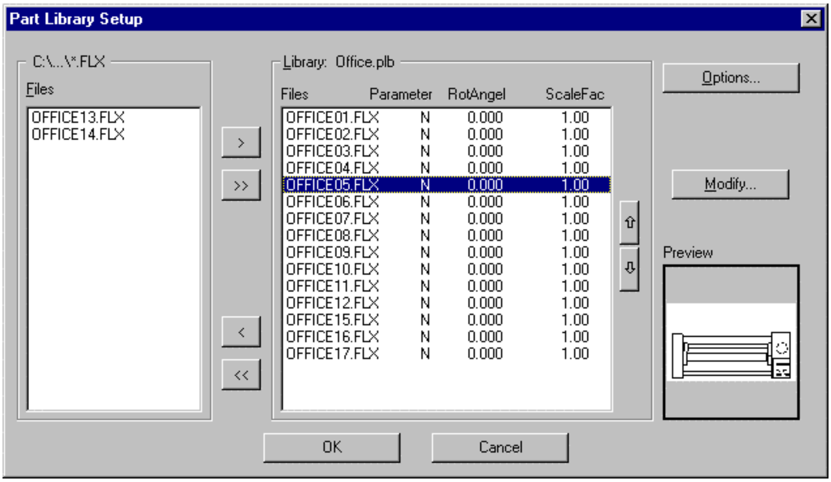
Setup and Manage Part Libraries

The command **PLBSETUP** (Part Library Setup) allows the user to setup up new part libraries and/or modify existing part libraries.

The command provides Dialog Boxes that:

- select the drawing files to be included in a library;
- set preferences for part or symbol insertions from the library; and
- set the parameters which determine the layout and options of the part library Dialog Box.

PLBSETUP creates .PLB files. These files store the parameters for a single part library. A part library is called by using the command PARTLIB, and then specifying the name of a .PLB file. No knowledge of structure or syntax of the .PLB format is required to use the PLBSETUP command.



Specifying the Drawing Files Belonging to a Part Library

The Dialog Box **Part Library Setup** allows the user to specify the drawing files to be incorporated into the individual library. He or she can only choose drawing files from one folder (directory). It is not necessary to include all of the drawings in a folder. This allows the user to maintain all of the drawings used for part or symbol insertions within one directory, and at the same time permits the creation of multiple “thematically separated’ libraries.

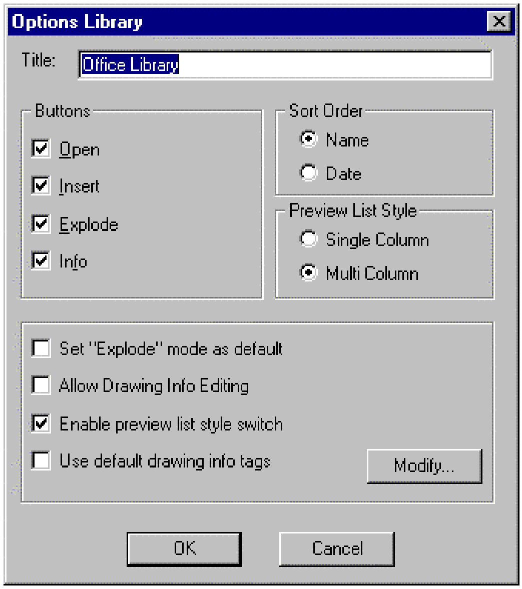
To add drawings to the current library highlight the file names in the left side file list and then click to the button > to insert them to the file list making up the library. The buttons (<, >) permit removal of drawing files from the list of part library files. When only one specific file is selected from either of the list boxes, the preview image of the drawing will be displayed in the Preview Area. The buttons >> and. << allow the user to easily move file names from one list to the other.

Insertion Parameters

It is possible to redefine insertion parameters for the files added to a library. Select one, more than one, or all entries from the list of the library files and then select ***Modify***. Next, activate the option *Activate insertion parameters* and enter the default values of the Scale Factor and the Rotation Angle for insertion.

Part Library Dialog Box Layout and Options

Using the **Options Library** Dialog Box, the user can determine the layout and the available buttons and options of the Dialog Box which comes up when calling this individual library with the PARTLIB command:



The edit box **Title** allows the user to enter a name for the Dialog Box of the part library.

Preferences for the Preview List

Preview List Style lets the user specify whether the part library Dialog Box should start up with a single column or multi column style of the list showing the drawing previews)

Sort by... helps the user determine whether the drawing preview list should be sorted by name (usually determined for part libraries) or by date (usually determined for project libraries).

Buttons assists the user in deciding options provided in the Dialog Box for a specific Part Library.

Open allows the user to open a selected drawing for modification.

Insert allows the user to insert a selected drawing into the current drawing as part.

Explode allows the user to specify whether the part should be inserted as complex object (block), or exploded (broken down to single entities).

Info allows the user to open the Drawing Information sub-dialog of the specific library.

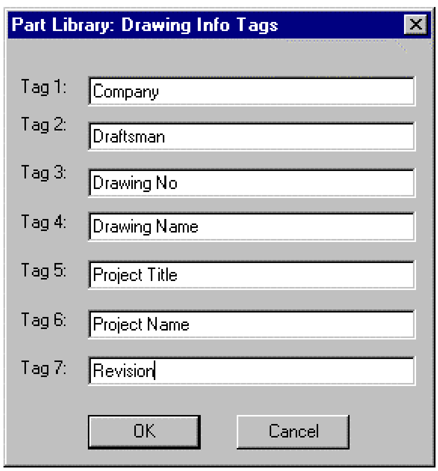
Set "Explode" mode as default sets the preference (activated/deactivated) for the mode of the button **Explode.**

Allow drawing info editing specifies whether the user may alter drawing file information stored with the drawing database.

Enable preview list style switch determines whether the library Dialog Box should contain the ***Quick Mode / Extended Mode*** button, which, in turn, allows the user to toggle between single column and multi column style in the file preview list.

Use default info tags determines whether default info tags (Project, Name, Drawing No., Draftsman, Notes) or user defined info tags are used when drawing information is displaced in the *Drawing Information* Dialog Box of the PARTLIB command.

Modify, in the preferences area, allows the user to determine individual tags for the file info fields used in the specific library (for example; manufacturer, price, etc. in a library managing purchased parts).



Attributes

It is important to understand the procedures when working with **Attributes**. Attaching **Attributes** to entities or parts can facilitate the drawing process, and help create a more meaningful drawing. It also helps set the stage for further design work using the drawing parameters. **Attributes** can facilitate strengthen of materials calculations and other engineering requirements.

General

Attributes are, in general, text objects which are related to, or joined to, **Parts**. They are treated together with the Part as a whole object. The text objects may be constant, or variable. They may be visible, or hidden. There may be a single **Attribute** with a Part, or several.

Attributes are used to define the part by adding textual information. This information may be used to label Parts in the drawing, or may be used to contribute to a data base program for analysis later in the engineering process.

An **Attribute** consists mainly of three components; its name; its request; and the value. A series of options will be offered during the creation of an **Attribute**.

The **Name** is the designation of the **Attribute** under which it is integrated in a part. If a part contains more then one **Attribute**, the name is the most important criteria for the effective usage of the **Attribute**.

The **Request** is a component of the **Attribute** which can be displayed during the insertion of the Part in the command line area. Normally, the request contains an invitation to enter a variable **Attribute** value.

The **Value** is that portion of the **Attribute** which is written, together with the Part, into the drawing. It can be constant or variable, visible or transparent.

The following parameters of an **Attribute** may be defined during the creation of the Attribute, and edited later in the design process.

- Insertion point;
- Font options, specifically Font name, font height and font angle;
- Reference point for insertion;
- Visibility, whether visible or hidden; and
- **Attribute** value, whether constant or variable.

Attribute Definition

You may define an attribute and place it as text object in a drawing without associating the attribute with a part. In that mode the attribute definition would represent only an isolated text object.

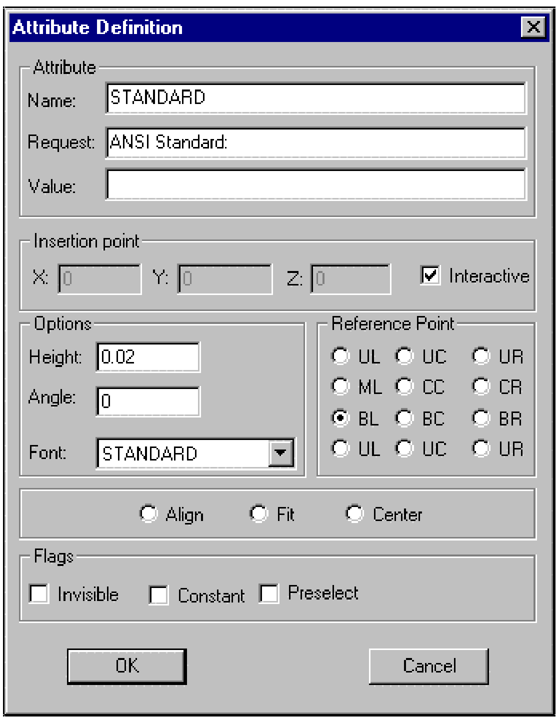
Attributes are, therefore, linked to parts. They should be created in connection with parts and be joined with them.

In the following paragraphs the assumption will be that the part has been created, and that the attributes are being joined to the part. In practice, it does not matter which is created first, the part or the attribute.

Creating an Attribute

 In order to create an **Attribute**, enter the command either through the selection of **Define Attribute...** in the menu, or by typing ATTDEF at the command line.

This dialog box will appear:



Attribute Name

Enter the name of the attribute in the input field. Use as descriptive a name as possible. An attribute name can consist of up to 31 characters, and may include special characters _ and $.

Request

Request is a textual component of the attribute which can be displayed during insertion of the Part in the command line area. Normally, **Request** invites you to enter a variable attribute value.

Request can consist of up to 256 possible characters. Blanks and special characters are allowed.

As an example, you may enter in this input field "Enter norm term". This would be a variable. A typical answer to **Request** would be "Schedule 40 Steel Pipe".

This answer is interpreted as an attribute value and is inserted in the drawing (see below).

Occasionally you may omit the answer to **Request**. If a part is using a constant attribute value, for instance, the answer would be omitted (see below).

Value

The **Value** is that component of the **Attribute** which is written, with the part, into the drawing. It designates the part and contains information to be analyzed. Its content is determined either by **Value** or by **Request** (see above). An **Attribute Value** can consist of up to 256 possible characters. Blanks and special characters are allowed.

Attribute Value - Constant / Variable

You must distinguish between **constant** and **variable** Attribute Values. A value is constant or variable depending upon the status of the box labeled **Constant**. A cross in that box indicates that the value is constant. An open box indicates that it is variable.

Constant Attribute Values are discreetly defined when entered in the input field **Value**. Any time that the associated Part is shown in a drawing, the text that will accompany it is that which is shown in that particular input field. Since the **Value** has been defined as **Constant**, the request for a variable value will be superfluous and will be ignored.

Variable Attribute Values can be entered whenever the part is inserted in a drawing. For instance, a Part may be constantly defined as “Schedule 40 steel pipe”, but, on insertion into the drawing, require a diameter and a length. In this case, the description “Schedule 40 steel pipe” is constant, while the diameter and length are variable. This Part, therefore, has at least three **Attributes**.

Insertion Point

The insertion point may be defined either of two ways: by the entry of absolute coordinate values in the X, Y, and Z axes, or by selecting **Interactive**.

Interactive lets you define the **Insertion Point** via the cursor. You can exercise this option by checking the **Interactive** control box. The advantage to **Interactive** definition is that **Dragmode** is active, offering you a preview of the insertion.

Text Options

Text options help you determine the height (font size), the rotation and the Font style of the attribute text.

Font size is defined by inputting the **Height**. Enter the desired text height in drawing units**, not** in text points.

Via the input field **Angle,** specify the insertion angle of the **Attribute** value. For an counter-clockwise rotation enter a positive value, for a clockwise rotation enter a negative value, between 0 and 360 degrees.

A selection window is available to define the **Font** style. A list of the available fonts is displayed. Select the desired **Font** style.

Reference Point

The **Reference Point** determines the position and alignment of the text object in relation to the **Insertion Point** of the **Attribute**. To this end, a **Reference Point** is defined on the text object. During insertion, this point will be coordinated with the **Insertion Point**.

For instance, if the reference point is the lower left corner of the text object, then the text will run from the **Insertion Point** to the right, and the lower edge of the text will be located at the height of the **Insertion Point**.

There are twelve **Reference Points** available. Select one of them weit cursor. The **Reference Points** represent the initials of the position on the text object. BL, for instance, represents Bottom Left, while UR represents Upper Right. Center positions are available as well.

Align

Text objects may be rotated or scaled during the process of insertion by **Align.** Determination of a rotation angle, as well as a scaling factor, is made by indicating two points in the drawing using the cursor.

Align is activated by checking t e circle marked “**Align”**. W en this opton is active, the options Insertion Point, Height, Angle, and Reference Point will disappear. **Align** requires you to pick two points on the drawing. The text object will be placed “left justified” on this baseline.

Follow t e prompts for a first point, then a second point. The first point is interpreted as the **Insertion Point**, te second point sets the angle of insertion. The second point will also detrmine a scaling fact wich will detrmine the text height.

Fit

Fit enables you to rott and expand a text object during te insertion. T e text objec not caled, so the text height emains unchanged, but e text sting is expanded or cmpressed as required by **Fit**. T e procedure for tis opton is:

- activate **Fit** by selecting the box. Te options **Insertion Point**, **Angle** and **Reference Point** will then disappear. The parameters required for **Fit** are two points which must be defined during insertion of te txt object. Te reference point of the txt ject is automatically positoned “left stied” on te base line. Click “OK” and ten;
- answer t e prompt for “first point”. This point is interpreted by the program as the **Insertion Point.** It is also the point of rotation for the rotation angle and the first point for te expansion factor; then
- indicat the secoint. This will detrmine the rottion angle and expansion factor, and set the txt height.

Center

Center allows fast insertion of a text object with the vertical and horizontal alignment centered. Text height and rotaton angle may be specified as usual.

Flags

Invisible

Invisible is a switch panel that controls the visibility of the attribute value. If **Invisible** is activated, the attribute value will not be displayed during the insertion of the Attribute in the drawing. This option should be chosen if the Attribute only contains information to be analyzed later.

It is also convenient to use this option when there are so many attributes that showing them all would make the drawing too "busy", confusing the drawing.

Constant

This function is used for switching between constant and variable Attribute values (see above).

With **Constant** active the text entered in the Value field will be written as a constant attribute value in the drawing. There will be no opportunity to insert any variable data.

With **Constant** de-activated, a request for variable attribute Values will appear at the command line.

Preselect

Preselect allows you to create attributes that accept their default values. When **Preselect** is active attribute values are not requested. **Preselect** is the most flexible mode, since it allows you to edit pre-selected **Attribute Values** after insertion using the editing commands.

Multiple Attributes

To assign multiple attributes, simply repeat these steps until all of the attributes are set and inserted in the drawing.

Part Definition With Attributes

To save a part with **Attributes,** follow the instructions earlier in this chapter for saving **Parts**. Specify the name of the part and its **Insertion Point**. Select the proper **Attributes**. Insert the **Attributes** in the Part Drawing where they belong. Finish this selecting by pressing ENTER. The Part, including the selected **Attributes,** is now saved **internally** in the drawing.

A part provided with **Attributes** can also be saved **externally.** Follow the procedure for saving a drawing externally, and add the **Attributes.**

Inserting Parts with Attributes

Inserting internal or external parts which include attributes is performed in the same manner as inserting parts without attribute (see above in this chapter: **Inserting Parts**). If variable attribute values are used the request for input of attribute text is added.

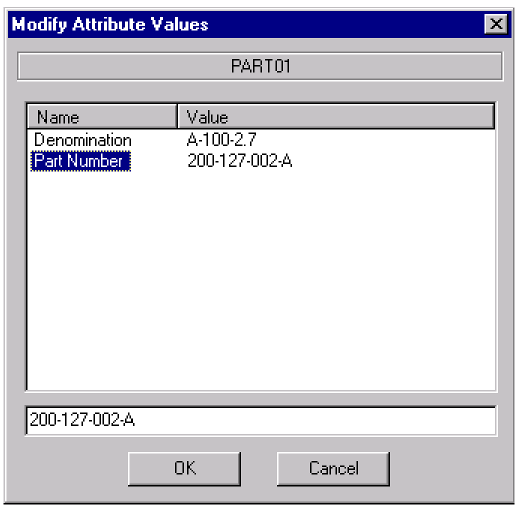
In that case, enter the desired attribute value in the command line area and confirm the input or confirm a possible parameter value by pressing ENTER. If the part contains several variable attributes, repeat the attribute value the required number of times. The part will be inserted in the specified position.

Editing Attributes

Editing attributes of parts already inserted in the drawing may be done in two different ways:

Edit Part Attributes

If only the attribute values are to be changed, (whether constant or variable) use the command ATTXEDIT or call the function **Edit Part Attributes...** from the **Part** menu. The following Dialog Box will appear:



In the text window, requests are displayed on the left side, and current values on the right. Select the attribute value to be changed. The value selected is transferred to the editing field (below) for editing.

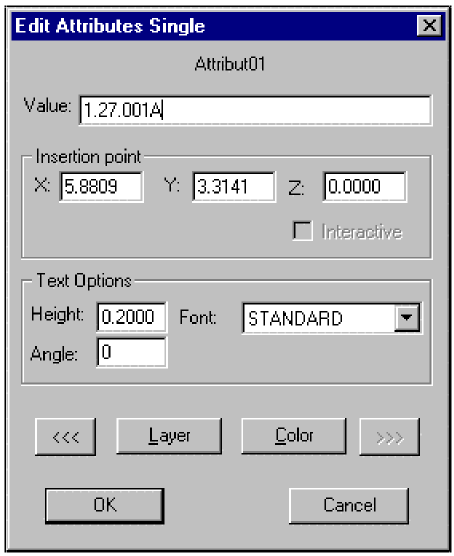
Modify Attributes


This command allows editing all aspects of the attributes associated with a part during one operation. Constant values cannot be edited like in the above command. Enter the command ATTEDIT in the command line area or select the functions **Modify Attributes...** from the *Part* menu.

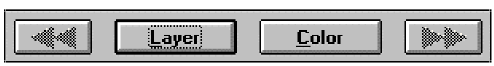
The first prompt will ask for selection of the part to be edited. Make this selection either by naming the part, or by selecting the part with the cursor. Select only one part.

After the part selection, the dialog box shown below will appear. The editing process from this point is very similar to the procedure followed in creating an Attribute, except that the name is now displayed, and cannot be changed.

If there is any question on this procedure, review the procedure for **Creating an Attribute**.



There are some new switches in this dialog box that should be explained.



If a part contains several attributes, individual attribute values can be displayed in the input field **Value**. The right double arrow will display the next attribute, while the left double arrow will display the previous attribute. The attribute that can be edited at any one time is indicated by the Name.

Attribute Utilities

The program offers several utilities to modify attribute text entities. These utilities can be chosen from the **Modify Attribute Text >** sub-menu in the menu *Parts*.

Edit Attribute Text Value

Call the utility by entering ATTVALUE or select **Edit Attribute Text Value** from the sub-menu **Modify Attribute Text**.

The command permits you to select a single attribute of a part insertion and change its value at the command line:

```
> ATTVALUE
Select attribute of part: P1
New value for attribute <47>: 48
```

Move Attribute Text

Call the utility by entering ATTMOVE or select **Move Attribute Text** from the sub-menu **Modify Attribute Text**.

Use the command ATTMOVE to select a single attribute of a part insertion and modify its location:

```
> ATTMOVE
Select attribute of part: P1
Target point: P2
```

Rotate Attribute Text

Select **Rotate Attribute Text** from the sub-menu **Modify Attribute Text** or call the utility by entering **ATTROT**.

The command ATTROT allows you to select a single attribute of a part insertion, the rotation angle, and modify it:

```
> ATTROT
Select attribute of part: P1
New angle <0>: 90
```

Display Modes for Attributes

In the dialog box for drawing modes there is an area in which the display of attributes can have the status ON, OFF or NORMAL, see the **Draw Menu - Drawing modes** command on page 115.

The following settings are available:

Display as defined	All attributes are displayed according to their definition.
All invisible	All attributes are invisible.
Display all	All attributes are visibly displayed.

Select the desired display option by clicking the proper switch.

Externally Referenced Drawings

Externally Referenced Drawings serve especially to

- create assembly drawings from different files
- insert detail drawings into a finished one
- temporarily reference another drawing in the current drawing

Often, the biggest benefit of using externally referenced drawings is given within a running project with a workgroup creating several drawings and details in a network.

Another advantage is that within the current drawing only a link to a referenced drawing is stored but not the entire entity and symbol database information of the referenced file.

In several situations, the usage of externally referenced drawings is more convenient and applicable then inserting external parts into the current drawing as blocks.

XLINK command: Control externally referenced drawings

The XLINK command allows you to create a **link** to an external drawing and to control externally referenced drawings (also called Xrefs) in the current drawing.

Differences to Part Insertions

Although the treatment of linked drawings is basically similar to those of part insertions (block insertions), the some fundamental differences are:

A Xref establishes a link to another drawing file, but it does not become a permanent part of the current drawing.

Dependent symbols

When attaching an external file, those symbol tables are loaded, into the current drawing, which are required to display the referenced coincident to its original: layers, linetypes, text styles, dimension styles, and block definitions.

The program utilizes the following sheme to name dependent symbols in the current drawing:

`Logical_Xref_Name|Dependent_Symbol_Name`

Or, another way of thinking about this;

(Original Drawing Name)|(Symbol name in the Original Drawing)

For example, the layer names of an externally referenced drawing attached with the logical name SHAFT are listed as:

(Original Drawing Name)|(Layer Name in the Original Drawing)

```
SHAFT|CONSTRUCTION
SHAFT|DETAILS
SHAFT|DIMENSION
```

Other dependent symbol table entries like linetypes or blocks will carry the same naming convention.

Note, that you cannot redefine or rename dependent symbols. Also, dependent blocks cannot be inserted and a dependent layer cannot be made the current layer.

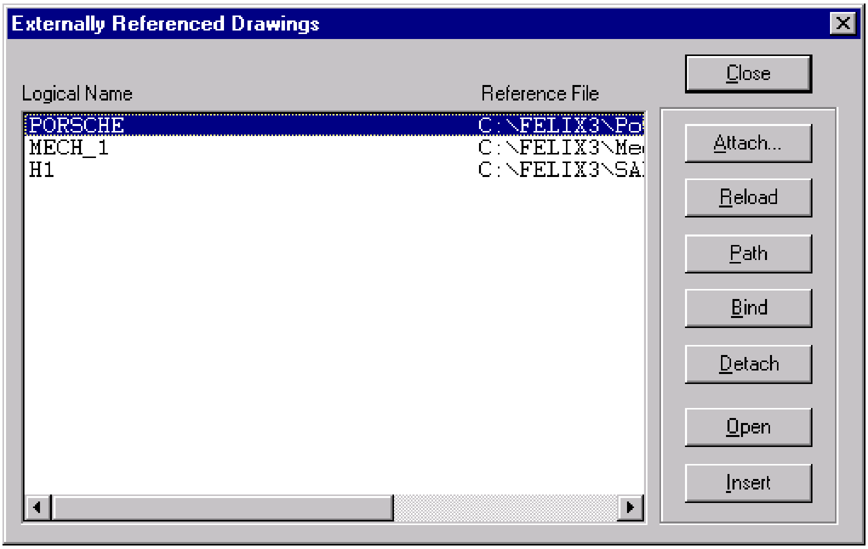
But the dependent symbol conventions allow you to control the visiblity, the colors, and the lintypes of externally referenced drawings. The layer dialog box displays the names of the dependent layers and you can apply the options On/Off, Thaw/Freeze, Linetype, and Color on them.

The logical name prefix makes it easy to distinguish the dependent layers from the layers originally defined in the current drawing.

To keep the visibility and displayed settings over subsequent sessions, make sure to set the **system variable VISRETAIN** to 1. By default VISRETAIN is set to a value of 0, which means that visibility and display modifications made to the dependent layers are not maintained when you leave the current drawing.

The dialog box Externally Referenced Drawings

The dialog box displays a list box and a column of option buttons.



The list box displays the following information on externally referenced drawing files which have been inserted into the current drawing

1. the logical name assigned to it when the file was attached, and
2. the path and filename of the referenced drawing

The options available to manage externally referenced drawings are provided in a column of buttons:

Attach Insert an external drawing file into the drawing as a block

Reload Update externally referenced drawing to display its current state

Path Relocate/redirect or alter source of the referenced file

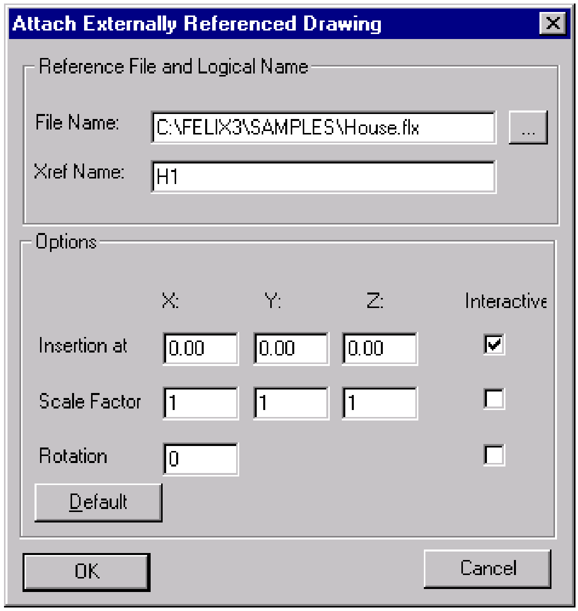
Bind Transform an externally referenced drawing to an ordinary block in the drawing (no longer an Xref)

Detach Delete an externally referenced drawing from the current drawing and erase ....

These options are described in detail in the following sections.

Attach

The *Attach* option allows you to insert an external drawing as a Xref block to the current drawing.



To attach an external drawing file (set up a link and insert the contents),

1. The standard file dialog box allows you to specify the drawing you want to attach to the current drawing.
2. Within the dialog box Attach External Drawing, a unique Logical Name (block name) needs to be determined for the associated referenced file. The program suggests to use the name of the file. However, in many cases it is recommended to use a short logical name, because the logical name will be used as a prefix for the names of dependant symbol names for layers, linetypes, etc.
3. The other options of the dialog box let you specify preferences for the insertion and are identical to those found in the dialog box of the INSERT command (insertion point, scale factor, and rotation angle), as described above in this chapter.

The drawing attached is based on the most recently saved version of the drawing file.

Reload

The option *Reload* allows you to update an external reference any time during a current drawing session. Any Reload reflects any subsequent modifications made to the external file.

Note, that when you open a drawing which contains externally referenced drawings, these are displayed automatically in their current state. When working on a project in a workgroup over a network, it might be neccessary to update one or more external references while you are working on a current drawing. Another person may be updating one of your external referenced drawings at the same time that you are using the Xref. When this person saves the work, you would need to Reload the Xref to ensure that you are seeing all of the updates this person just saved. The program will reload the specified drawing(s) in the state as they have been saved most recently.

To reload externally referenced drawings in a current drawing session,

1. select the external reference to be updated from the list box
2. click the *Reload* button

Path (Relocate or alter source of the referenced file)

Occasionally the storage location of an externally referenced drawing might get changed or the referenced file may be renamed. Also, when sending the project files including referenced drawings to someone else, the path to the referenced files may change and need to be updated.

In such a situation the option *Path* of the dialog box allows you to re-establish the path to the referenced drawing, to reload a renamed referenced file, or to replace an referenced drawing by another file.

To edit or relocate the path to a previously linked reference drawing,

1. select the external reference to be relocated from the list box
2. click the *Path* button

From now on, the link to the referenced file will be maintained as specified and the data of the referenced file will be updated from that source.

Bind

In certain situations it is useful to make an externally referenced drawing a permanent part of your drawing, for example when you need to send the drawing to someone else or if it has been finalized and is going to be archived. Binding the referenced drawing to the current drawing has the advantage that there is no need to gather an assembly of drawings within an archive or mail.

The option *Bind* of the XLINK command allows you to transform an externally referenced drawing to an ordinary block in the drawing. Once you bind an externally referenced drawing, the link to it is broken and you will no longer see any modifications made in the original referenced file.

Bind also incorporates the layers, linetype definintions, text styles, and dimension styles into the current drawing. Within each corresponding dependent symbol name, the vertical bar symbol (|) is replaced by the three characters *n*, where *n* is a number which is usually 0. For example, the layer name HOUSE|BASEMENT will be altered to HOUSE0BASEMENT. The number *n* would increase from 0 to 1 if the renaming of the symbol or layer would cause a duplication within the drawing. n can range from 0 to 9 as required.

To bind one or more external references to the current drawing,

1. Select the corresponding entries in the list box of the dialog box
2. Click the *Bind* button

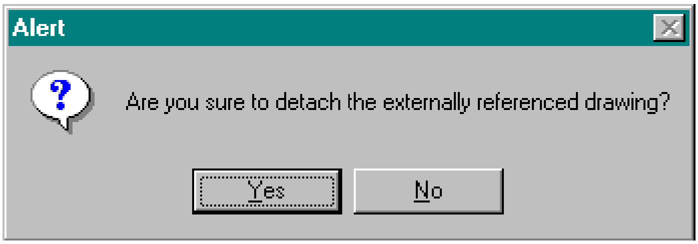
Detach

Externally referenced drawings which are not needed any longer in the current drawing can be removed from the base drawing with the *Detach* option. This will remove all of its dependent layer, linetype, and block references as well.

Note: Although you can erase an externally referenced drawing with the DELETE command, it is recommended to use the Detach option of the XLINK command, as this operation will also remove the Block definition from the drawing database.

To remove one or more external references from the current drawing,

1. Select the corresponding entries in the list box of the dialog box
2. Click the *Detach* button



Open

As the Multiple Document Interface allows you to display up to four drawings on the desktop, the option *Open* of the dialog box allows you to open an externally referenced drawing simultanously on the CAD desktop. This allows you to modify a referenced drawing, for example, if you have detected discrepancies or inconsistencies in the linked drawing which needs to be fixed.

To open a referenced drawing in another drawing window (viewport),

1. Select the external reference you want to open from the list box
2. Click to the Open button

Once you have modified a referenced drawing, save the drawing (and, if the modification has been finished, close it). To display the revised external reference in the master drawing, call the command XLINK and reload the corresponding file (see above: Reload).

Insert

The option *Insert* allows you to insert another copy of the Xref choosen from the list box into the current drawing. You may then define the insertion point, the scale factor and the rotation angle for that additional instance of the external drawing in the current drawing.

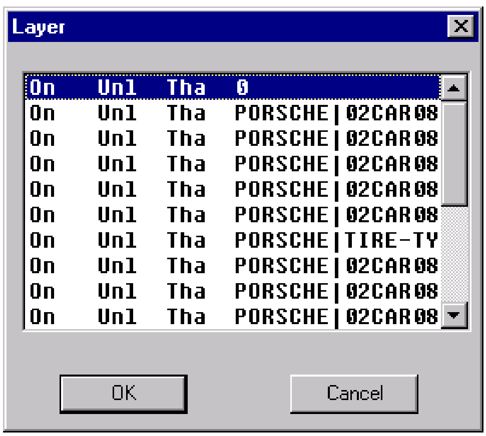
You can move, scale, rotate, and copy externally referenced drawings. But, you cannot explode an external reference.

You should not remove a Xref from the current drawing with the DELETE command, because it does not purge the Xref block definition from the drawing database and the associated symbols will be maintained. To remove an external reference correctly from the current drawing, it is recommended to use the *Detach* option of the XLINK command.

The *Insert* option, however, can serve to reinsert an external reference which accidently has been deleted.

XINSERT command:
Attach an external drawing file to the current drawing

The command XINSERT is processed similar to the XLINK command applied with the *Attach* option. The command is provided for convenience, allowing you to directly attach an external drawing to the current drawing.



Grouping Entities

Drawing entities can be combined into Groups. Grouping entities allows the user to manipulate all entities within the Group in a single operation (Moving, Rotating, Mirroring, or Scaling, for example). Single entities of a Group may still be *edited* individually (for example, intersecting or stretching). Entities may be removed or added to a Group at any time. An object may be a member of more than *one* Group. *Groups* may be contained in *other* Groups. Group definitions are stored in a distinct table of the drawing database. Groups are maintained in this fashion from one drawing session to the next.

Differences Between 'Parts' and 'Groups'

A Part (a block) has its own insertion point. A Group has none. A Part can occur multiple times within the drawing. If a block is redefined, all of its occurrences are updated. Groups are not treated like that. Groups can, however, be copied. A Group that originates from another Group that has been copied is a separate union of entities. Each Group has a unique name.

Note*:*
If an entity which is member of a Group is included in a part definition, the entity is deleted from the drawing and removed from the Group when the part is originally defined.

The Command GROUP

To Group entities, or to manage and manipulate Groups, use the command GROUP. The options within the command are:

Create	Explode	Rename	Add	Remove	Selectable	

Option	Meaning
Create	Brings entities into a newly named Group
Explode	Abolishes a Group definition
Rename	Alters the name of an existing Group, especially useful for anonymous groups
Add	Adds drawing objects to an existing Group
Remove	Removes drawing objects from a Group
Selectable	Determines whether a Group is selectable or not

Creating Groups

To create a group, first determine a unique name. The name will be required to identify the group later for a number of command options. In addition, specify whether the group should be selectable or not (see below).

To define a new group,

1. call the command GROUP with option Create,
2. specify a unique group name (max. 31 characters),
3. if desired, enter a description (max. 64 characters),
4. determine whether the group should be selectable or not (the default is Yes), and
5. select the drawing objects, which should belong to the named group.

To create a group using Command Line:

```
Enter the Command  GROUP;
Select option: Create
Group name: G1
Description: Screw
Should the group be selectable (Yes/No)? Yes
Select objects: Select the entities which should belong to the group
```

Exploding Groups

Group definitions may he abolished by using the **Explode** option of the **GROUP** command. When a group definition is dissolved the corresponding entry in the group table is deleted in the drawing database. The original drawing entities will return to their previous state within the drawing.

The only way to abolish a group definition is by abolishing its name.

To explode a group,

1. choose the command GROUP with its option *Explode*,
2. at the prompt "Group name: " enter the name of the group, to be exploded.

Because a single entity may belong to multiple groups and groups may be part of other groups, the **Explode** option does not allow the user to pick an entity within a group to explode the group.

This Explode option is not the same as the Explode command used with parts and hatching.

Renaming Groups

The name of a group may be changed by using the **GROUP** command **Rename**. It is useful to employ this method when "anonymous" groups that have been created by copy commands exist. The system will automatically apply anonymous names to copied groups (**A1, **A2, etc.) which exist through the use of one of the commands like **COPY**, **MIRROR**, or **ARRAY**.

To rename a group,

1. call the command **GROUP** with its option **Rename;**
2. at the prompt "Current group name: " enter the name of the group (for example "*A0");
3. at the prompt "New group name: " enter the name of the specified group (for example "Screw_Copy1")

Add / Remove Entities

Two other options of the command GROUP are provided to allow the user to integrate additional entities into an existing group or to remove entities which are currently in the group.

After a group name is specified, the entities which belong to the group are highlighted.

- To add entities to an existing group, simply 'pick' the entities that are to be added (they will *not* be highlighted).
- To remove entities from an existing group, 'pick' the highlighted entities that are to be removed

Note:
Should *all* entities be deleted from a group, remember that *the group will still exist as long as it has a name!*

Selectable

When a group is set as selectable that means the group is active and will be treated as a group by all editing commands.(based upon the rules shown below) Setting a group to be unselectable means you are temporarily turning the group definition off so you can apply many editing commands to the individual entities. You can switch between selectable and unselectable at any time, turning this particular group on and off. This can be handy when an entity is in multiple groups and selecting this entity would make it difficult to pick the correct group to be worked with.

Working with Groups

Editing commands will treat all entities in a group as a single object. Commands like **MOVE**, **ROTATE**, or **COPY** will act on the entire group.

If the user should select an object for one of these commands at the prompt, that selection will lead to the automatic selection of all group entities.

A typical prompt of a command like this would be:

```
Select objects:
```

The rule is:
In commands, which allow multiple selections, identifying a *single* entity of a group is a selection of *all* entities belonging to the group(s). When the user is *prompted* to select multiple objects, he or she may choose the object selection option ***Group*** from the options bar, and enter a group name to select all of the entities belonging to the group. Groups, therefore, can also be treated as named selection sets.

Other **MODIFY** commands (**Offset**, **Lengthen**, **Intersect**, **Trim**, etc.) request a single entity in one or more steps (usually to modify the geometry of drawing entities). A typical prompt for a command like this would be:

```
Select entity (Line, Circle, Arc, 2D-Polyline):
```

In this case **single selection** is being used. Any time the user is prompted to select a single entity, the group does not matter. These commands modify the geometry of single selected entities even though those entities belong to a group. The modified entities will remain members of the group. This feature allows great flexibility in working with GROUPS and provides significant advantages for the user compared to using BLOCKS.

The following table summarizes the editing commands and describes how entities which are members of a group will be processed when using each command.

DELETE	Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. When a single entity of a group is deleted, the entity is removed from the group definition.
MOVE, ROTATE, SCALE, FLIP	Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used.
MIRROR, COPY, ARRAY	Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. The system will apply anonymous names to copied groups ("*A1", "*A2", etc.) that are created by the use of these commands.
STRETCH	Those entities belonging to a group which have been selected using the object selection modes *Crossing* or *Cpolygon* will be stretched,
TRIM, EXPAND	The prompt to “select cutting edges” or “...boundary edges” will include all entities of a group, unless the Object Selection Mode *Single* has been used
OFFSET, FILLET, CHAMFER, LENGTHEN, INTERSECT	These commands always prompt the user to select a *single* entity to be modified. That an entity belongs to a group does not matter in these operations. Single entities altered by these commands will remain in the group. If *new* entities *originate from these commands* they will *not* be automatically included in the group.
DELPARTIAL, REJOIN	When an entity (line, circle, arc or 2D-Polyline) is broken into two or more entities by partial deletion, only one of the entities will remain in the group. When an entity which has been broken into two or more pieces is rejoined, the newly created entity is no longer member of any group, even if one or more of the pieces was part of a group.
PARTDEF, HATCH, HATCHEDIT	Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. This also applies to the command HATCHEDIT when the user is prompted to `select additional objects:` to be included to the set of entities for associative hatch editing.
TEXTEDIT	The text entity retains its association to its group if the command TEXTEDIT is used to modified text.

Chapter 9

Dimensioning

This chapter contains a summary of the various dimensioning options, a description of the ways in which the dimensioning functions can be controlled and detailed instruction on creating and editing dimensions.

Dimensioning is a very important element of any technical drawing, so much so that without an exact and standardized method of dimensioning a drawing looses its practical value.

For this reason a number of dimensioning functions are made available to the program user. These enable a fast, easy, precise, and standardized method of dimensioning.

Measurement Points

The basis of dimensioning is the definition and insertion of **Measurement Points** (DEFPOINTS). The program registers the coordinates of these **Measurement Points** and calculates the corresponding distance or angle between the points. Using conversion variables, the measurement is scaled to represent a dimensional value which is then inserted in to the drawing.

In coordinate dimensioning the X and Y coordinates of the **Measurement Points** are inserted into the drawing.

Measurement Points can be placed anywhere on the geometry of an object or, quite independently, anywhere on the drawing area. It is general practice, however, to place them at geometrically significant locations, such as endpoints, mid-points, or at an apex. The dimensioning function offers different methods of object selection and allows you to choose that which best suits the character of the dimensioning requirement.

Utilizing the selection function to mark the element which is to be dimensioned, it is possible to ensure that the point of measurement is placed at a relevant and realistic position on the geometry of the element.

This method is particularly useful when using the general object selection function to place individual dimensioning points on lines and angles.

Measurement Points are drawn on a separate layer of the drawing. This layer is usually allocated the name DEFPOINTS. If this layer does not yet exist in a drawing, it will be created as soon as the first dimensioning process is carried out.

Associative Dimensioning

Associative Dimensioning alters the position and placement of measurement points and drawing elements as editing commands are processed.

Associative Dimensioning will cause any dimensioning which exist to be recalculated, and the new dimension displayed, as soon as an object has been modified.

Dimensioning Elements

In addition to scaled values, dimensioning commonly includes and displays the following elements.

1. Dimension Lines with Arrowheads

Dimension lines connect the points of measurement, or the extension lines to those points of measurement, and enable a reference to be created between the dimension value and the points of measurement.

Dimension lines usually end with arrowheads which physically connect the extension lines or measurement points. These arrowheads can be replaced with an oblique or any other symbol acceptable to the drawing.

Dimension lines, with extension lines, act as a reference point for measurement text.

2. Extension Lines

Extension lines are used when the dimensioning is to be done outside of the objects perimeter. They extend the points of measurement to the dimensioning lines.

The extension lines in diameter or circumference dimensioning start at the dimension value and “point” to the outer edge of the object.

3. Dimensioning Text

This text is made up of the dimension value and can be extended to include the unit of measurement or any relevant label.

Alternative Units of Measurement and Their Designation

Alternative units of measurement can also be displayed with the dimensioning text. This allows an object to show two different units of measurement, such as a combination of metric and imperial values.

Tolerances and Fit Entries

Tolerances and Fit entries can be represented, a requirement in mechanical drafting. Appropriate values can be shown in the drawing, alongside the dimensioning text.

Types of Dimensioning

T type of dimensioning is dictated by the nature of its positioning in the drawing. FelixCAD has accommodated nearly all of the positioning variances imaginable.

1. Linear

Linear dimensioning shows the linear extension of an object. You may differentiate between the X axis (horizontal) extension, Y axis (vertical) extension, and the absolute (positioned or rotated) extension.

2. Coordinate - Horizontal, Vertical, Rotated or Aligned

This dimension represents a distance between a specific point and its origin in the coordinate system. The measurement can be taken on either the X or Y axis and inserted into the drawing. It is particularly applicable in surveying drawings.

3. Circular - Diameter, Radius, and Center

This dimension contains a variation applicable to the diameter and radius dimensioning of circular or arced objects. The calculated measurement is shown on an extension line outside of the circle or arc.

4. Angle

The angle between two separate objects, or the angle inside a specific object can be measured and calculated using angle measurement.

5. Individual

A single measurement is displayed.

6. Serial

Serial positioning is only used in linear dimensioning and follows the same principals as continuous dimensioning. It allows you to add a second dimension, or to create a chain of dimensions. All values in the chain will be shown in one row, the first dimension in the row dictating the direction in which the chain will be organized (horizontal, vertical, aligned or rotated.)

7. Continuing

A second dimension is connected or joined to an existing value. The endpoint of the first dimension acts as the starting point for the second. Both dimensions are shown in one row. The second measurement will follow the same horizontal, vertical, aligned or rotated organization determined by the initial dimensioning.

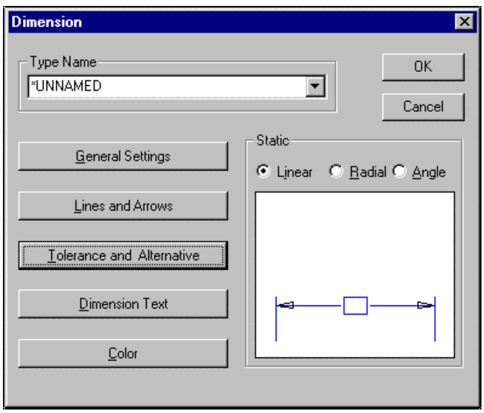
8. Baseline

Baseline dimensioning is used with linear dimensioning. It follows the direction of the existing dimension. Using the starting point of the measurement as a base, a second measurement is made. This method can include any number of dimensioned or un-dimensioned points or objects.

Defining the Dimensioning Function

Dimensioning is defined by a Dialog Box. To access the Dimensioning Dialog Box, type in the DIMTYPE command or, from the menu *Detail***,** select **DimensionType**.

Use the style name, input and selection dialog to either store the current settings as a dimensioning style, or to select a pre-defined style as the default.

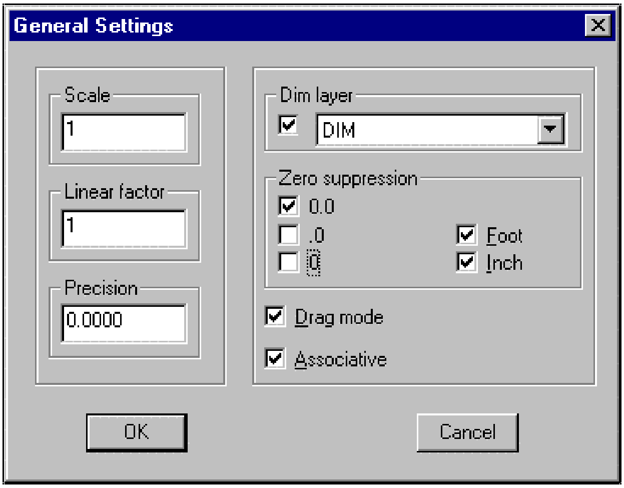


The buttons described below each open additional dialog windows in which individual dimensioning parameters can be defined.As most of the control options are self explanatory, the following explanations are kept to a minimum.

General settings

Scale

Scale value enlarges or reduces the size of all dimensioning elements (apart from the dimension value itself) and allows the dimensioning elements to be scaled to suit the aspect ratio of the drawing.



Linear factor

Linear factor converts the measured drawing units into a dimension figure. The default value is 1.

Precision

Precision defines the number of decimal places to which the measurement will be carried. This cannot be changed here, see Precision and Units Settings, page 45.

Dimension Layer

The layer in which the dimensioning values are stored is identified by the name entered in the dimensioning layer selection field. Select the layer required or enter a new layer name. It is prerequisite that the option has already been activated by clicking the appropriate control box. A check mark indicates the current status of the option.

Zero Suppression

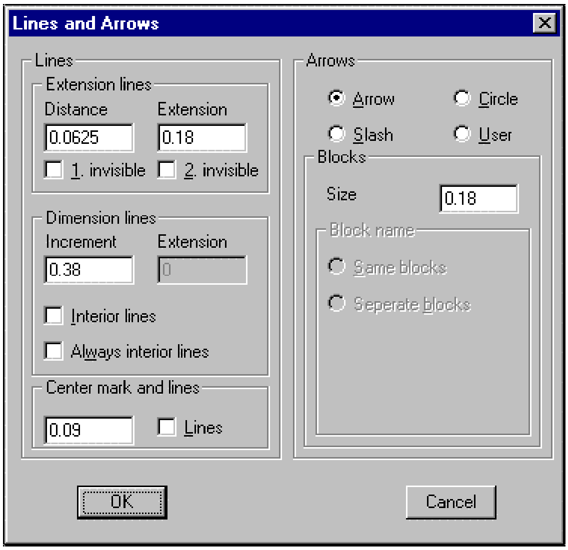
The Zero suppression field decides whether or not zeroes are displayed.

Drag Mode/ Associative

The last two control boxes activate or deactivate either the “rubberband” mode and / or the associative character of the dimensioning.

Lines and Arrows

Extension lines and the nature of the arrowheads to be used are defined in the dialog box *Lines and Arrows*.



Lines decide the characteristics of the extension and dimension lines.

Extension Lines

This area defines the spacing of the extension lines.

Distance allows you to choose the space between the object and the point at which the extension lines begin. A value = 0 indicates that the lines will start directly on the object itself.

Extension dictates how far the extension lines should extend past the dimensioning line.

The control boxes marked **Invisible** can be used to suppress either one or both of the extension lines in the drawing.

Dimension Lines

This area defines the spacing and character of the dimension lines.

Increment sets the spacing between successive dimensioning lines which use the same extension lines.

Extension allows you to enter the distance which the extension lines should run past the dimensioning lines. This value can only be entered when the **Slash** option has been selected as the dimension arrow type.

The field **Interior Lines** or **Always Interior Lines** specifies whether dimensioning lines will be allowed to extend past the extension lines.

Center Mark and Lines

This area whether or not, and what size, cross-mark indicating a circle center point will be used. It also defines whether or not dashes or strokes should be added to the center point cross or not.

Arrows

This field specifies the type of arrowhead to be used.

Blocks

The length of the arrow head is defined in this block. The size selection of this area is the only selection normally active. **Block Name** is active when the arrow style selected is **User.** The size of the **Block** to be used is defined here.

Block Name

This field decides the **Block Name,** and is active only when the arrow style selected is **User**. The name commonly in the field is ARROW.

Separate Blocks

This field allows or suppresses the use of divided or separate arrow blocks. When it is activated, it is also possible to define which elements should be used at either end of the dimensioning line.

Tolerance and Alternative Dimensioning

Tolerance Dimensioning

Tolerance Dimensions specify the tolerance, the maximum and minimum values, allowed in a dimension. The display of these elements is controlled from this area. Values for the positive or negative tolerance are entered in the field marked **Values**. These fields also enable you to type in a line of text (String).

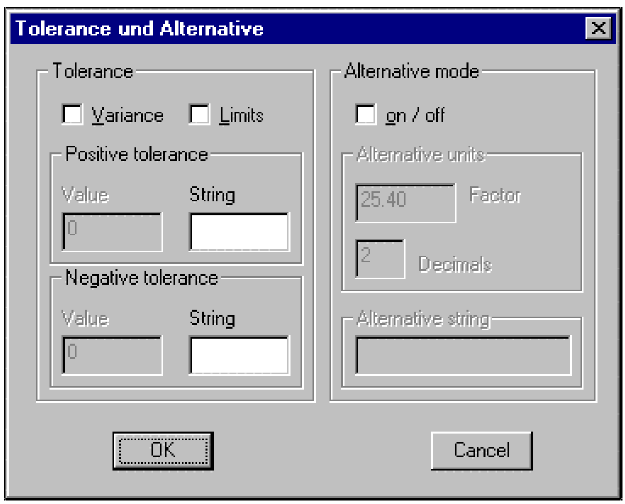
Alternative Dimensioning

Alternative mode options control the use of a second alternative dimensioning method. When activated, this mode will allow dual dimensioning in metric and imperial values.

Factor: The conversion factor between the two units of measurement is entered in this field.

Decimals: The number of decimal places selected is input in the decimal display.

Alternative string should contain a description or title of the second unit of measurement.



Dimension text

These fields determine the position and look (size, style, spacing) of the dimensioning text.

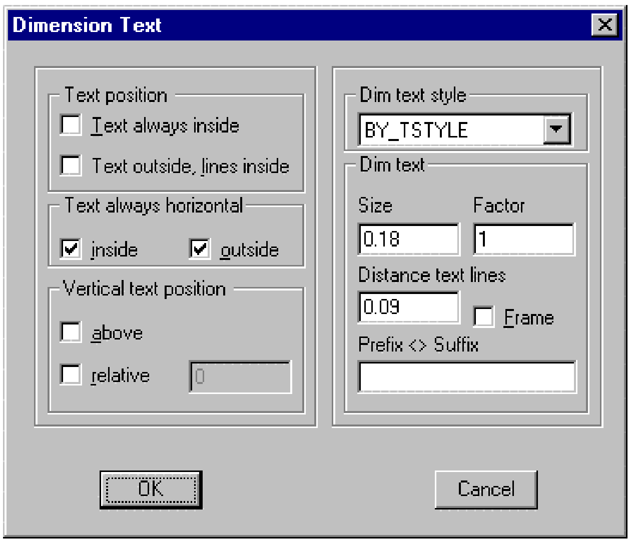
Text position, Text always horizontal, and Vertical text position fields serve to define the position of the text with respect to the dimension lines.

Dimension Text Style, Dim Text

These fields define the style and size of the text, and the distance separating text lines.

The prefix and suffix fields enable you to type in a character string which will appear before or after the dimensioning text. For example, this field may be used to display the unit of measurement.

The < > character decides whether the character string will appear before or after the dimensioning text. For example, **approx. <> ft.** entered in this field would produce the following dimensioning: ***approx. 12 ft.***.



Colors

The Color Dialog Box permits colors to be assigned to the dimensioning lines, extension lines and dimensioning text (dimension value, alternative dimensioning, tolerances, limit values, unit of measurement etc.)

Seven standard colors are available in addition to the BYLAYER and BYBLOCK options.

Creating Dimensioning

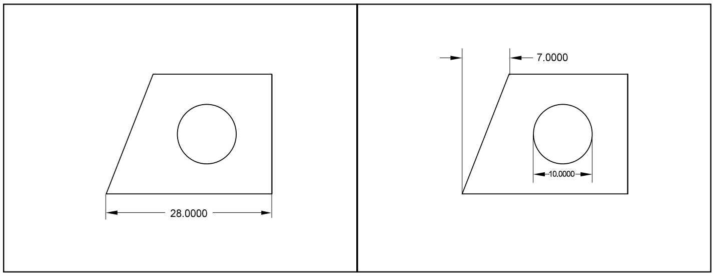
Function Selection

The individual dimensioning functions can either be selected using the option **Create Dimensions** in the *Detail* menu, or with the symbols in the DIMENSIONING palette, or by typing in the appropriate command abbreviations. A list of the command abbreviations is shown below.

Function	Command
Horizontal	DIMHOR
Vertical	DIMVER
Aligned	DIMALI
Rotated	DIMROT
Coordinate	DIMORD
Diameter	DIMDIA
Radius	DIMRAD
Mid point	DIMCEN
Angle 3 Point	DIMA3P
Angle 4 Point	DIMA4P
Continuous	DIMCON
Series	DIMSER
Baseline	DIMBAS

Horizontal (dimensioning)

Horizontal dimensioning is linear dimensioning that measures the distance between two points or the linear extension of an object parallel to the X axis.



Define two points of measurement (DEFPOINTS) for this command.

You may also select a specific dimensioning object function from the Options Bar by choosing the **Select** option. Lines, polylines, circles and arcs may now be selected. Point to the desired object with the cursor. The start and end points, or the diameter of the object, will be interpreted automatically and used for the measurement calculations.

Once the points of measurement have been defined, you will be asked to choose a position for the dimensioning line.

The distance between the two points will be measured along the x axis and displayed.

Pressing the ENTER key will verify the calculated value and insert it into the drawing.

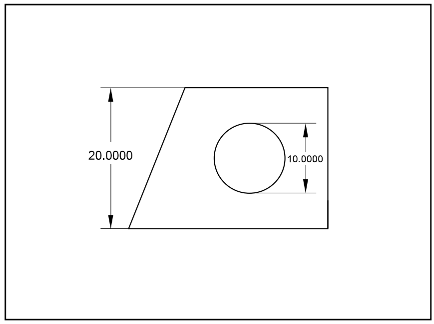
```
> DIMHOR
First Point: P1
Second Point: P2
Dimension line positioning: P3
Dimension text <2.50>: <hit Enter key>
```

or

```
> DIMHOR
First Point: SELECT
Select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.5000>: <hit Enter key>
```

Vertical (dimensioning)

Vertical dimensioning measures the distance between two points or the linear extension of an object parallel to the Y axis.



Define two **points of measurement** (DEFPOINTS).

You may also select a specific dimensioning object function from the Options Bar by choosing the **Select** option. This allows lines, polylines, circles and arcs to be selected. Select the desired object with the cursor. The start and end points, or the diameter of the object, will be interpreted automatically and used for the measurement calculations.

Once the points of measurement have been chosen, you will be asked to define a position for the dimensioning line. Do this by entering the relevant coordinates, or with the cursor.

The distance between the two points will be measured along the Y axis and displayed accordingly. Pressing the ENTER key will verify the calculated value.

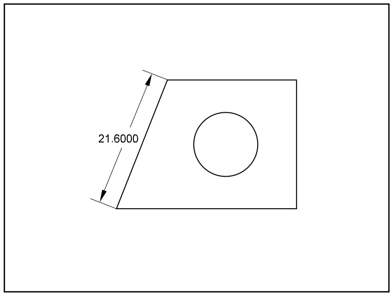
```
> DIMVER
1. Point: P1
2. Point: P2
Dimension line positioning: P3
Dimension text <2.50>: <hit Enter key>
```

or

```
> DIMVER
1st Point: SELECT
select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.50>: Return
```

Aligned (dimensioning)

Aligned dimensioning is a type of linear dimensioning which measures the absolute distance between two points (or the linear extension of an object), regardless of the relative position of the axes. This makes it possible to dimension lines, edges and spaces etc. which are not located on the principal axes of the coordinate system.



Define two points of measurement (DEFPOINTS) or select a specific dimensioning object function from the options bar by choosing the **Select** option.

The first point of a circle will be defined as a point situated along the perimeter of the circle. The second point will be obtained by drawing a straight line from the first point, through the center point of the circle, to the opposite side of the perimeter of the circle.

Once the points of measurement have been chosen, you will be asked to define a position for the dimensioning line. You may either enter the relevant coordinates, or select with the cursor.

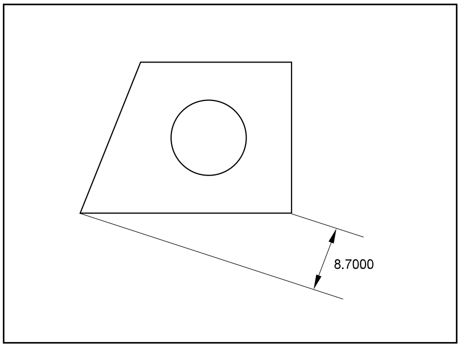
The absolute measurement between the two points will be displayed. Pressing the ENTER key will accept the calculated value. The dimension will be inserted into the drawing and shown on a horizontal plain.

```
DIMALI
1st Point: P1
2nd Point: P2
Dimension line positioning: P3
Dimension text <2.50>: Return

or

DIMALI
1st Point: SELECT <CR>
select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.5000>: Return
```

Rotated (dimensioning)

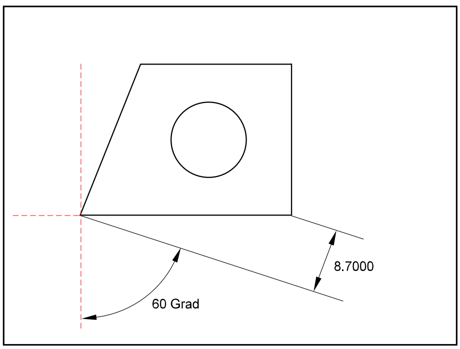
Rotated dimensioning is also a type of linear dimensioning which measures the distance between two points, or the linear extension of an object, at a angle which is user-specified.

When dimensioning the distance between two points (or an object with linear extension) at an angle, the space allocated to the dimension will always be smaller than the absolute dimension measurement. This procedure is equivalent to looking at an object from a specific angle and establishing the visible length from a different view point.

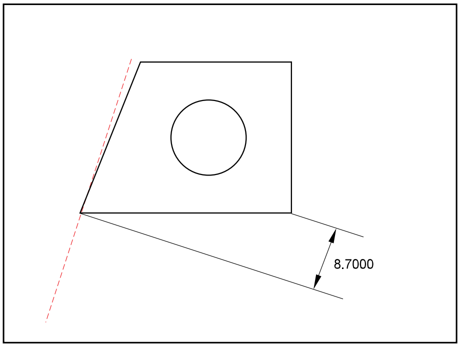


You will be asked to define the angle at which the dimensioning is to appear in relation to the axial alignment of the coordinate system. Do this by entering the coordinates at the keyboard, or by using the cursor.

The first figure entered defines the angle between the direction of the negative Y axis and the first dimension line as seen in a clockwise direction. The apex of this angle is determined by the position of the first point of measurement.



If you define the angle at which the dimensioning is to take place by selecting two separate points, then a perpendicular line will be drawn at 90° to the extension lines.



After having determined the dimensioning angle, you will be prompted to define two points of measurement (DEFPOINTS). You may also select a specific dimensioning object function from the Options Bar by choosing the **Select** option

The start and end points of a line, polyline or arc will be interpreted automatically and used for the measurement calculation.

The first point of a circle will be defined by the program as the next point situated along the circle perimeter. The second point will be obtained by drawing a straight line from the first point, through the center intersection of the circle and on to the opposite side of the circle perimeter.

In contrast to objects which have a linear extension, this method will always measure and display the exact diameter of a circle. Only the position of the extension lines will be affected when using rotated dimensioning.

Once the points of measurement have been chosen, you will be prompted to define a position for the dimensioning line.

The distance between the two points will be displayed at the designated angle orientation. Pressing the ENTER key will accept the calculated value and cause it to be inserted horizontally into the drawing.

```
> DIMROT
Angle of rotation <0>: 60
First point: P1
Second point: P2
Dimension line positioning: P3
Dimension text <2.5000>: Return
```

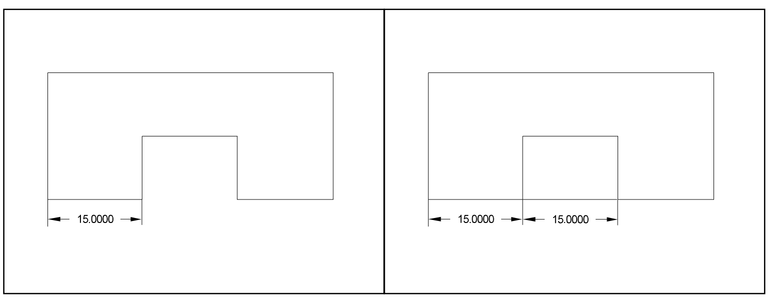
or

```
> DIMROT
Angle of rotation <0>: 60
1st Point: SELECT
Select line, polyline, circle or arc: P1
Dimension line positioning: P2
Dimension text <2.5000>: Return
```

Continuing (dimensioning)

Continuing dimensioning is a variation of linear dimensioning in which existing dimensioning is continued or extended. A second dimension is linked to an existing dimension so as to create a dimensioning chain. Continuing dimensions can only be created when a drawing already contains an existing linear dimension.

A continuing dimension always inherits the same properties as the existing dimensioning. Therefore, it is only possible to connect dimensioning of the same type. For instance, a new horizontal dimension can only be added or linked to an existing horizontal dimension.



A message will appear in the text window requesting the selection of an existing dimension. When selecting the desired dimension remember to place the selection cursor as close as possible to the point at which the continuing dimension is to be linked.

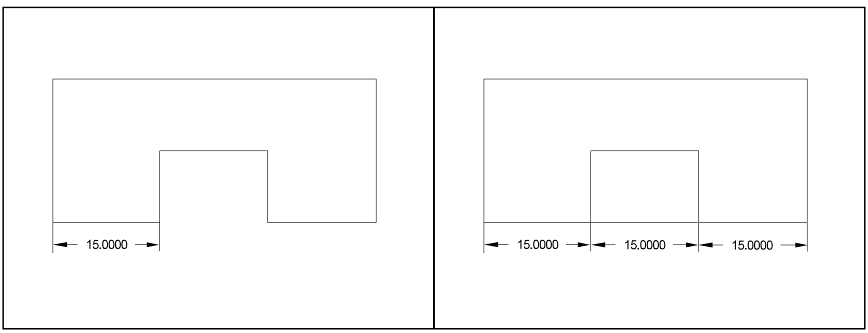
The dimensioning is based on two points of measurement. The starting point for the continuing dimension will be the point which lies closest to the selection point at the time the object is selected. Choose the second point by entering coordinates or by selecting with the cursor. Whenever possible, employ the object snap function.

The program will display the calculated dimension. This value can either be altered, or confirmed with ENTER. You will not be asked to position the dimensioning line, as this will automatically be placed in line with the existing dimensioning.

```
> DIMCON
Select a dimension: P1
2nd Point: P2
Dimension text <2.5000>: RETURN
```

Serial (dimensioning)

Serial dimensioning is done in the same way as continuing dimensioning. Where only one additional dimension can be added to an existing dimension with continuing dimensioning, serial dimensioning enables you to link as many dimensions as required. One dimension must already exist to be used as a basis for the rest.



Only dimensioning of the same type can be added, since the additional dimensioning will inherit the properties of the dimensioning used as the base.

You must select the dimensioning that is to act as the basis for the series, taking care to place the selection cursor as close as possible to the point at which the continuing dimension is to be linked.

The dimensioning is based on two points of measurement. The starting point for the series dimensioning will be the end point which lies closest to the selection point of the base dimension.

In addition to **Select** and **End**, which are the same for serial dimensioning as for continuing, two other selections may be made from within the **Series Dimensioning** function bar: **Continue** and **Baseline**.

Continue allows you to link one additional dimension and should be selected when creating serial dimensioning. This is the default option and only needs to be confirmed.

Baseline allows you to create baseline dimensioning parallel to the existing linear dimensioning. This function is described in the next section.

Select the second point by typing in the coordinates or selecting with the cursor. Use **OSNAP** whenever possible.

The program will display the calculated dimension. This value can either be altered, or confirmed with ENTER. You will not be asked to position the dimensioning line. It will automatically be placed in line with the existing dimensioning.

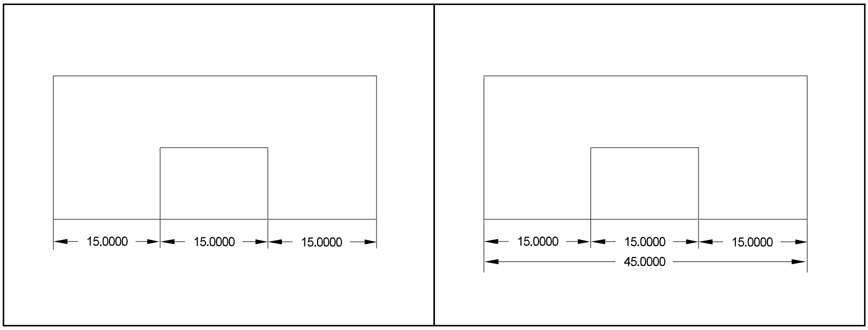
```
> DIMSER
Select a dimension: P1
Series <continuous> RETURN
2nd Point: P2
Dimension text <2.5000>: RETURN
Series <continuous>: RETURN
2nd Point: P2
Dimension text <2.5000>: RETURN
…
ESC
```

You may repeat the above steps as many times as dimensions are required by re-selecting the **Continue** option. Additional dimensioning can be terminated by pressing the ESC key.

Baseline (dimensioning)

Baseline dimensioning adds a second measurement line to the drawing parallel to the existing dimensioning lines. Using this function, you may read the cumulative dimension between two end points that are covered by a number of individual dimensions within a chain.

Baseline Dimensioning assumes that a form of linear dimensioning already exists. The appendixes dimension will assume the same properties as the existing measurements.



Mark the starting point of the dimensioning chain, taking care to utilize the same direction of measurement as laid down during the creation of the individual dimensions within the chain.

Select the second point to be used for the baseline dimension by entering the coordinates at the keyboard or with the cursor. Whenever possible, use **OSNAP** for accuracy..

The program will display the calculated dimension. This value can either be altered, or confirmed with ENTER. You will not be asked to position the dimensioning line. The second line will be drawn at a given distance from the existing dimensioning lines.

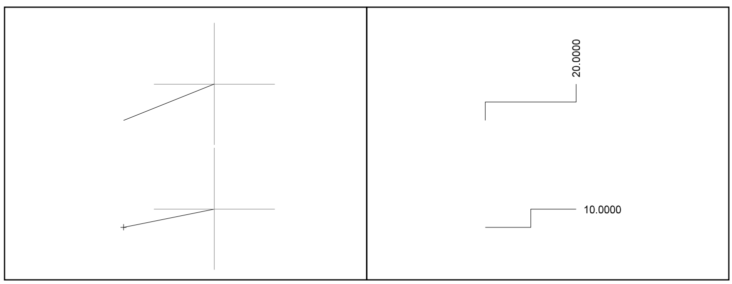
```
DIMBAS
Select a dimensioning line P1
2nd Point: P2
Dimensioning text <20.50>: Return
```

Ordinate (dimensioning)

Ordinate dimensioning is used to determine the X and Y coordinates of a point for entry into the drawing. The X and Y options will be displayed within the options bar. By selecting one of the options you can determine whether the X or Y coordinate value is to be defined. The X-value is the default and will be used if no other option is selected.

You will be prompted to select the point to be measured. Type in the coordinate values, or select with the cursor. If the location represents a specific point on an entity, use **OSNAP**. The program will ask for LEADER LENGTH. The user should enter the extension line position.

Use the cursor to select the position at which the measurement value is to be displayed. This figure can be positioned at any alignment and distance from the dimensioning line. To help, the program draws a line from the selected coordinates to the current cursor position. This illustrates the distance and alignment between the two positions. Select the desired position with the cursor.



Example of coordinate dimensioning X-axis (top) and Y-axis (bottom)

If the dimension is not placed horizontally or vertically in relation to the measurement point, then the extension line will appear in orthographic mode, and not as a straight line. An example of this is shown in the above diagrams.

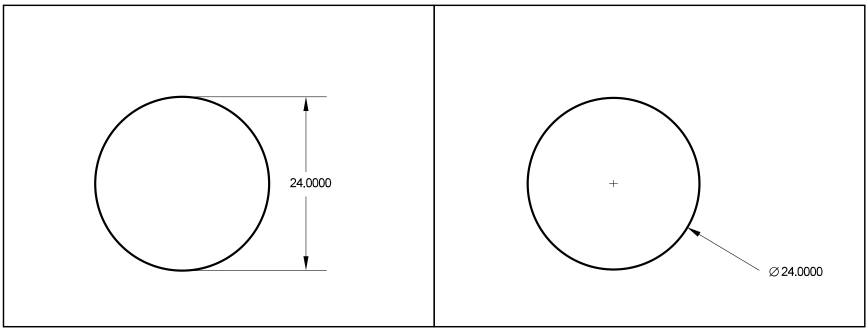
The program will display the calculated coordinate. The value can be confirmed by pressing the ENTER key, or amended as required. You may type in any alpha-numeric sequence, thus enabling the point to be titled as required.

```
> DIMORD
Coordinate point: X
Coordinate point: P1
Extension length: P2
Dimensioning text <20.00>: <hit Enter key>
```

Diameter (dimensioning)

Diameter Dimensioning is used to display dimensions of circles and arcs. Unlike the horizontal or vertical dimensioning functions, no measurement lines are shown. Instead, a single extension line is used to show the relationship between the value and the object. The diameter symbol indicates a special dimensioning feature used only in circle and arc dimensioning.

The first prompt will ask for the specific circle or arc. Select with the cursor. The point at which you selected the object will be defined as the point at which the dimensioning extension line will connect to the element.



QUADRANT object snap helps select specific points on the circle or arc. After the selection, the program will display the corresponding diameter measurement. This can be confirmed with the ENTER key or amended as required.

The second prompt will ask for the position of the extension line. Select a coordinate at the required distance from the object.

The alignment of the extension line is fixed by the position of the two selected points and cannot be altered. The extension line will be drawn along an imaginary straight line which extends from the center of the object to the measurement point.

The extension line and measurement text will be displayed and a cross (+) will appear at the center of the circle of arc.

```
> DIMDIA
Select circle or arc: P1
Dimensioning text <24.00>: <hit Enter key>
Extension length: P2
```

Radius (dimensioning)

Radius dimensioning is done the same way as diameter dimensioning except that a radius dimension is calculated instead of a diameter measurement.

```
> DIMRAD
Select circle or arc: P1
Dimensioning text <24.00>: <hit Enter key>
Extension length: P2
```

Center (dimensioning)

Center dimensioning is used with circles and arcs. It is used to position a mark (+) at the center of a given object.

The function is carried out in a single step. You will be asked to select the desired circle or arc. Select the required element.

If a circle or arc is correctly selected the center mark will appear. If it is not correctly selected you will be asked to select the object again.

```
> DIMCEN
Select circle or arc: P1
```

Angle (dimensioning)

FelixCAD contains functions which can be used to achieve fast and accurate angular dimensioning. These functions are **Angular3P** and **Angular4P**.

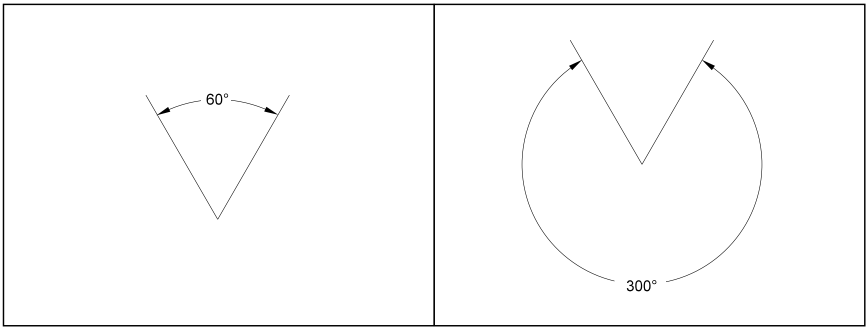
Angular 3P

This function uses 3 measurement points which are placed at the apex and legs of an angle. This function not only allows you to calculate the interior angle between the two legs, but also the exterior angle.

Select the vertex position. Use OSNAP whenever possible.

Define a point on each of the legs of the angle.

Define the location of the dimensioning arc by selection with the cursor. The dynamic preview function will assist you in predicting the result. The position chosen for the dimensioning arc also decides which angle, (interior or exterior) is to be dimensioned.



After confirmation, the program will display the measured value. Confirm with the *Enter* key, or change as required.

```
> DIMA3P
Apex: end
of: P1
1st Point: P2
2nd Point: P3
Position measurement curve: P4
Dimensioning text <60°>: RETURN
```

Angular 4P

The difference between these two functions is that the **Angular 4P** dimension is based on four points of measurement. These points could be the end points of two lines, or the tangent of the end points of an arc.

Both the interior and exterior angles can be measured and dimensioned.

The first prompt will ask for the objects (lines, polylines, arcs or circles.)

```
DIMA4P
Select line, polyline or arc: P1
Select line, polyline or arc: P2
Dimensioning text <60°>: RETURN
```

Modifying Dimensions

The program contains a number of functions which allow you to edit existing dimensions.

Several palettes contain many of these functions. All of these palettes may be reached from the **Detail** menu.

Create Dimensions

This palette may be selected by choosing **Dimensioning** from the *Detail* menu. The palette contains most of the common dimensioning methods, and makes the work of dimensioning a drawing much easier.

Modify Dimensions

This palette allows you to move the dimension values on the dimension line in a number of ways, each illustrated by an icon. There is also an icon to allow you to reach the **Dimension** detail Dialog Box, and one to bring up a detail editing palette from within the main palette.

Modify Dimension Text

There is a palette available that helps you choose these modifications. Select that palette by opening the sub-menu, and choosing **Palette Dim Edit**.

These functions may also be reached by typing the commands. The following table shows a list of some of these functions and their corresponding commands.

FUNCTION	COMMAND
Update Dimension	DIMUPD
Rotate dimensioning text	DIMTROT
Modify dimension text value	DIMTNEW
Move Dimension Text	DIMTMOVE
Undo Repositioning	DIMTPOS
Modify Dim. Orientation	DIMOBL

Update Dimension

After changing the current dimensioning style, all new dimensions will be created in line with the new criteria. The **Update Dimension** will update all of the existing dimensions so that they comply with the new dimensioning style settings.

To do this, mark or select the dimensions which are to be updated using one of the object selection functions found in the Options Bar.

You may update individual, group or all dimensions contained within a drawing at one time. Confirm the selection process by pressing ENTER. The marked dimensions will then be updated to conform to the new dimensioning style. The program will display an **"x dimensions found"** notice telling you how many dimensions have been updated.

These dialog examples will help to make the definition of dimensions more clear and also clarify the selection process.

```
> DIMUPD
Select Dimension(s): Fence
First Fence point: P1
Endpoint of the line: P2
Endpoint of the line: P3
RETURN
Select Dimension(s): All
2 Dimensions found

or

DIMUPD
Select Dimensions(s): All
Select Dimensions(s) RETURN
3 Dimensions found!
```

Storing a Dimensioning Style

The control functions which are to be used with the dimensioning function were described earlier in this chapter. Using these commands, the user may define the size, font and height, alignment and positioning of the dimensioning text, the look of the dimensioning arrows and measurement lines, as well as a number of other parameters.

The **DIMSAVE** command (Save Dimension Style) allows you to create a template of the individual parameters and store them as a named dimensioning style. This facility enables you to store and recall a number of different styles.

Architectural dimensioning, tolerance dimensioning or country and language specific options are examples of this (such as for individual units of measurement or labeling).

It should be noted that dimensioning styles can only be stored together with the current drawing. Therefore, they are only available for use from within that drawing. In order to use pre-defined dimensioning styles in other drawings, they should be created and stored in one of the template drawings, or loaded from a pre-defined prototype drawing. The use of templates is described in Chapter 1, **Getting Started: The Basics of Drawing**.

Procedure for Saving

Select the command **Modify Dimension** from the menu **Detail**. From the palette, select the icon representing the dialog box **Dimension.** Enter the proper settings and values from this dialog box. In the first field in this box, give the dimensioning template a descriptive name.

Before entering the name of a dimensioning style, you may list the file names which have already been allocated by typing a question mark (?) in the input field and pressing ENTER.

Dimensioning style names may be up to 31 characters long, and may consist of any alpha-numeric input except special symbols.. No differentiation is made between upper and lower case characters.

Confirming the selected name will store the dimensioning style.

In order to achieve a clearer overview, it is possible to list all of the file names which begin with a particular character chain by entering the chain at the input field. The standard DOS default (*) can also be used to substitute a complete character chain. When it is used in a search routine, (*) will list all of the existing file names. If you wish to search for specific dimensioning style files, type in the desired character chain followed by the (*) character. For example, <ar*> would result in a list of all files beginning with AR.

```
> DIMSAVE
Name of the dimensioning style: List
List of dimensioning styles <*>: ar*
ARCHITECT01
ARCHITECT02
```

Listing the existing dimensioning styles will complete the save dimensioning style function. Pressing ENTER will restart the option and save the current dimensioning settings.

```
> DIMSAVE
Name of the dimensioning style: ARCHITECTURE03
```

Reinstating a Dimensioning Style

This function allows you to reinstate a stored style file so that it can be reinstated as the current style. All of the stored style settings will be retrieved and used by the system for future dimensioning. Any dimensions which were created prior to the definition of the new style can also be updated so that they, too, adhere to the conventions of the new style settings. This is done by using the DIMREST command (Restore Dimension Style).

The program will then display the name of the current dimensioning style in the text window. You will be requested to choose the dimensioning style to be set as the current default. The program options bar offers you two methods by which to select the required dimensioning style.

- either type in the name of the stored dimensioning style, remembering that you can list all of the dimensioning styles already contained within the particular drawing; or
- use the cursor to select the desired dimensioning style.

Chapter 10

Hatching

This chapter discusses the process of cross-hatching an area, section of a drawing, or cross-section of a part. Hatching (or cross-hatching) drawing objects adds meaning to the drawing, and helps to differentiate the materials, the areas, or any of a number of other unique attributes.

Applying hatch marks or area fills using distinctive patterns increases the clarity and legibility of a drawing. In a number of specialized applications there are norms and specifications which demand the use of specific hatch markings within a construction drawing.

Hatch... enables you to use pre-defined hatch patterns to fill selected areas of the drawing. With the aid of a dialog box, you are able to select the desired hatch pattern from a list and display this example in the preview window. The preview function simplifies the creation of new patterns and makes it easy to alter existing ones.

Experienced users can choose to bypass this option completely and use the keyboard to input the **Hatch...** pattern procedures directly.

Associative Character of Hatch Patterns

A hatch pattern is described as having an *associative character* when it is positioned so that it adjusts to changes in location, size and design of the hatched area.

In order to use the associative character, the hatch pattern must be attached to objects of the drawing.

Hatch Pattern Definitions

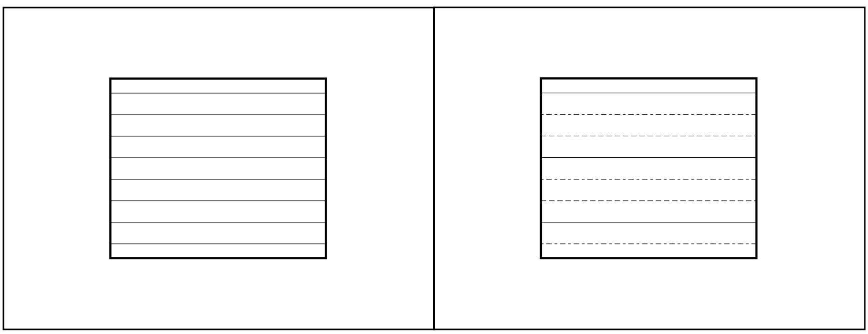
A hatch pattern is comprised of a number of elements. FelixCAD already contains a number of typical hatch patterns used in various standard fields of application. You may also choose to use other hatch patterns for specialized circumstances.

Hatch patterns are stored using the **.pat** file extension name.

Hatch Pattern Elements

Hatch lines

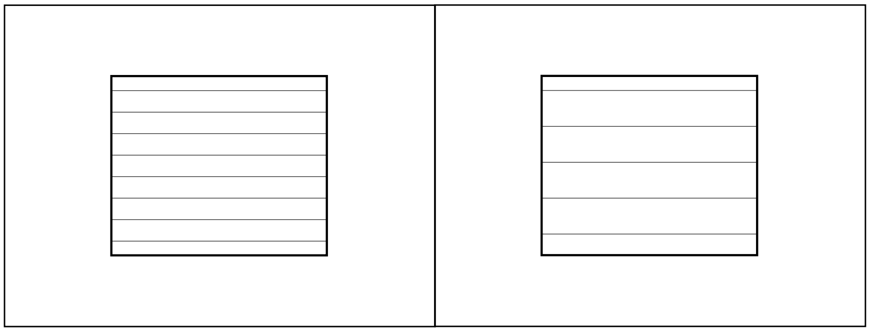
A hatch pattern can contain a number of diverse line types:



Diverse lines in a hatch pattern

Line Spacing

The spacing between the lines of a hatch pattern is measured in drawing units. The default line spacing is set to one (1) drawing unit. Altering the line spacing is possible to adjust a pre-defined hatch pattern to suit the proportions of any given drawing element.

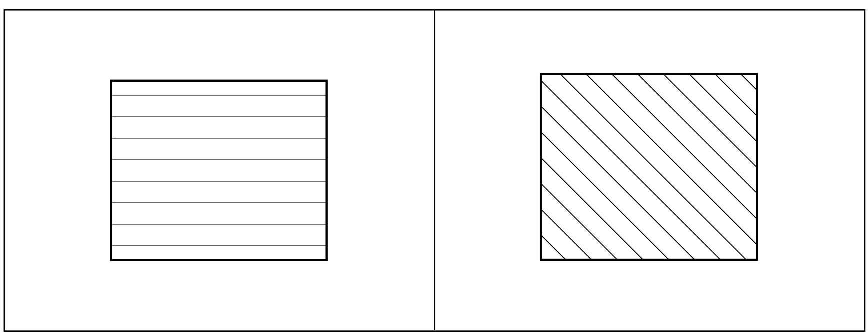


Hatch patterns with different line spacing.

Hatching Angle

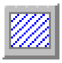
The lines of a hatch pattern can be drawn at any chosen angle. Standard hatch patterns are often either made up of horizontal or vertical lines, but may be drawn at any angle.

A hatch angle of 0° defines a horizontal line. A positive angle value will cause the lines to slant in a counter-clockwise direction. A negative value will cause a slant in a clockwise direction.

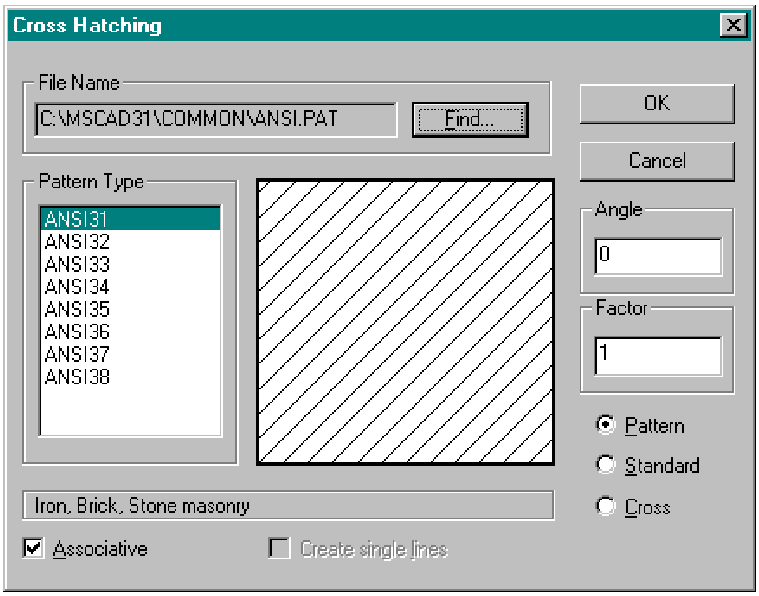


Hatch pattern drawn at a 0° angle (left) and one at a -45° angle (right)

HATCH Command: Adding Hatch Marks to an Object

 Select **Hatch...** from the *Detail* menu, or by typing HATCH at the command line.

The first prompt is to select the required object. The Option Bar contains a list of object selection functions. Select, and then terminate object selection process with ENTER.



The Hatch command will open a dialog box which allows you to select a pre-defined hatch pattern, change the parameters of a hatch pattern, or to create your own pattern. The preview function offers visual support.

To accept a hatch pattern, select the pattern and confirm your selection with OK.

If the hatch pattern is not stored in the current hatch file, then type the correct file name and path into the *Filename* input field, or click the **Browse** button located to the right of the input field. A dialog box which can be used to find and open the required **.pat** hatch pattern description file will open, displaying the various **.pat** files.

To alter an existing hatch pattern to suit the requirements of the specific construction job, select the appropriate pattern and then alter the angle and factor values (spacing of the hatch lines measured in drawing units) as required.

To create an individual hatch pattern select the **Quick** option button. This option will display a preview of a standard pattern consisting of continuous horizontal lines spaced at a distance of one drawing unit. Change this standard pattern using the angle and factor parameters until the design is complete.

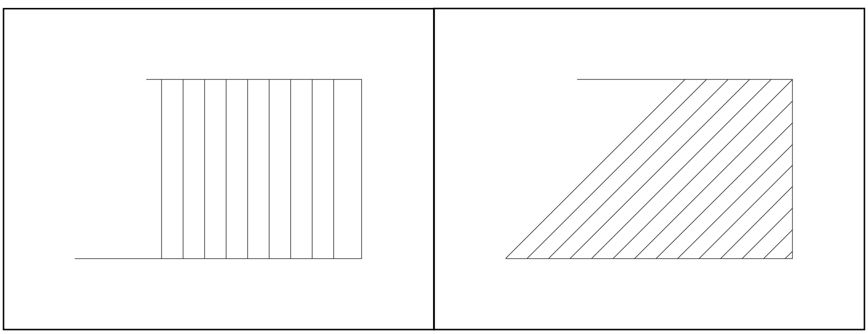
In addition, you can choose to activate the **Cross** button and add lines at right angles to the existing lines.

The Associative button allows the hatch pattern to update automatically when the boundary of the hatch is stretched.

To apply the designed hatch pattern to your drawing, exit the dialog box with OK.

Note:
Hatching Open Areas (Area Fill)

You can hatch patterns (area fill) to contours which are not fully enclosed. The hatch pattern will only be drawn to the extent of the selected contour line area. The following diagram illustrates this procedure.



Modify Hatching…

You can change the hatch patterns within an area by choosing this command or entering HPEDIT. It allows you to add additional objects to the set of enities to be hatched, as well as change the hatch pattern.

```
> HPEDIT
Select an associative hatch:
Select additional elements? <N>:
```

The routine then displays the hatching dialog box to select the type of hatching desired.

Chapter 11

Inquiry and Information Programs

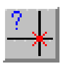
The chapter describes programs that provide you with information about the physical dimensions and properties of drawing objects, and about the current drawing parameters. These Info functions are available from the menu *Edit*.

Area and Circumference programs, coordinate identification programs, and distance and angle calculations between two points are provided.

The commands discussed in this chapter are:

Function	Command
Identify Coordinate	‘ID
Distance / Angle	‘DIST
Area / Circumference	AREA
Entity Information...	EINFO
Drawing Database Tables...	TABLES

Identify Coordinate

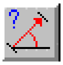


The transparent command ’ID (identify coordinate) determines the coordinates of a point to be identified.

Pick the point to be determined with the cursor. If it is a geometrically significant point of a drawing object (for instance,. corner point, center of a circle, intersecting point of two lines etc.) use the object snap functions. The coordinates will be displayed in the command line area.

```
> ’ID
Point: P1
X: 1.27  Y: 4.45  Z: 0.0
```

Distance / Angle

 Running the **Distance/Angle** command from the *Edit* menu (command DIST) will calculate the distance and angle between two points. The program displays the following information on the command line:

- the absolute distance;
- the distance on the X-, Y- and Z-axis;
- the Angle on the XY-plane; or
- the Angle from the XY-plane.

The points may be arbitrary points which do not have to be part of a drawing object.

The function operates as follows:

- determine the two points required by selecting them with the Cursor, using the Object Snap functions or by input of the coordinates; then
- the distance and angle between the two points will be displayed.

The term **Distance** is understood to mean the absolute distance between points.

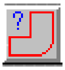
The terms **Delta X**, **Delta Y** and **Delta Z** designate the distance between the two points in the direction of the respective axis of the coordinate system. All length values are given in drawing units.

The **Angle in the XY-plane** measures the angle to the second point starting from the X-axis. It always displays the lesser-valued angle, no matter what the rotation direction.

The **Angle from the XY-plane** measures the angle between the first and the second point from the XY-plane to the Z-axis. It is assumed that the first point lies **on** the XY-plane.

```
> 'DIST
First point: P1
Second point: P2
Distance: 57.45
Angle in the XY-plane: 45 | Angle from the XY-plane: 10
Delta X: 40.00 | Delta Y: 40.00 | Delta Z: 10.00
```

Area / Circumference

 Select **Area / Circumference** or type the command AREA.

This function defines the circumference and the area of one or more drawing objects and displays the values. For objects that do not have width, the length instead of the circumference will be displayed.

The areas of single objects can be added or subtracted.

Using Area / Circumference

To determine the circumference and the area of one or more objects proceed as follows:

- Define the drawing object to be examined. Determining the circumference (or the length) and the area of an object is done by defining points that describe the geometry of the object. The preferred method is by using the cursor and object snap programs to select objects, but you can also input the coordinates of the points;
- Press ENTER at the request *Next Point:* to finish the selection. Example:

```
> AREA
First point: P1
Next point: P2
Next point: P3
Next point: P4
Next point: <hit Enter key>
Area: 0.75 | Length: 3.66
First point: <hit Enter key>
```

- You can compute another area by selecting points or objects, or you can finish the command by pressing ENTER.

Regular Areas

“Regular” geometric areas (circles, arcs, polylines, rectangles) may be selected by picking the option *Object*.

Object	AddArea	SubArea		

Select the desired object and confirm the selection by picking OK.

The determined values for the circumference and the area will appear in the text window.

You may continue computing the circumference and area of objects, or terminate the command by pressing ENTER.

```
> AREA
First point: Object
Selection of an entity (circle, arc, polyline): P1
Area: 0.87 | Circumference: 3.3
First point: ENTER
```

Add or Subtract Areas

The Area command also allows you to add or subtract areas from (or to) an area.

Select the first partial area or pick the points that describe the area geometry. The calculated values will be displayed in the text window.

Select the option *AddArea* to add a partial area, or *SubArea* to subtract a partial area, from the total area.

Object	AddArea	SubArea		

The values (circumference /length and area) for the total area will be displayed in the text window.

When partial areas are added or subtracted from an area, the total area after the addition or subtraction is regarded as the new parameter value unless you explicitly select a different option from the options bar.

The options bar will display only the options that are not active at the moment.

For example, if the *AddArea* option is active, the options bar will display:

Object	SubArea			

You should repeat the preceding steps if necessary to continue computing areas. When you have finished computing, press ENTER to end the command.

The following example clarifies the function:

```
> AREA
First point: Object
Selection of an entity (circle, arc, polyline): P1
Area: 0.87 | Circumference: 3.3
First point: AddArea
(ADD) First point: P1
Next point: P2
Next point: P3
Next point: P4
Next point: ENTER
Area: 0.75 | Length: 3.66
Total area: 1.62

(ADD) First point: SubArea
(SUB) First point: P1
Next point: P2
Next point: P3
Next point: P4
Next point: ENTER
Area: 0.75 | Length: 3.66 | Total area: 0.87

(SUB) First point: ENTER
```

The last determined value for the area is stored in the system variable AREA and can be retrieved via the LISP expression (getvar "AREA") at the command line.

Entity Info

 Call the command by entering EINFO or selecting **Entity Info…** from the menu *Edit*. This command lists information about the data stored for the selected objects.

The option *Clipboard* in the dialog box will copy the displayed information to clipboard.

Drawing Database Tables

 Call the command by entering TABLES or selecting **Drawing Database Tables** from the menu *Edit*.

These overviews can be requested in the dialog box:

Draw and Display Parameters
Layer
Line Types
Fonts
Part Definitions
Referenced Parts
Dimensioning Types
Hatching Parameters
Named Views
User Coordinate Systems

The option *Clipboard* in the dialog box will copy the displayed information to the Windows clipboard.

Chapter 12

Export and Import

One of the principle features of *FelixCAD* is its ability to interchange information with other CAD programs and other Windows programs.

FelixCAD allows you to export working files in a variety of formats to allow editing the drawings using other programs. Other drawing formats can also be imported with *FelixCAD*.

It is of prime importance to import and export the data without losing any attributes, textual information, or any other part of the data, either graphic or non-graphic.

This chapter contains the instructions for importing and exporting data.

First, this chapter addresses the procedure used to save and load drawings in .DWG and .DXF formats. This procedure allows interchange of information between *FelixCAD* and nearly any other Windows or DOS based CAD program.

Import and export functions are provided for:

- internal and external parts;
- attributes; and
- for text entities.

Each of these procedures are explained in this chapter.

Save Drawing As DXF / DWG

There are two formats available which are a de facto standard in the CAD world. These formats are the file formats **DXF** and **DWG**.

If a drawing with the format **.dwg** or **.dxf** needs to be exported it should saved using the **Save as...** command. Either keyboard SAVEAS or select the option **Save as...** from the *File* menu.

After entering the command a windows common dialog box will open allowing you to specify the drive, path, directory (folder in Windows 95) and file name. Execute these steps in the dialog box *Save File As*: Define the export format in the *File Type* list. Select the exact storing location for the export file from the *Directories* list.

WMF Export

A drawing can also be exported in the **WMF** format. WMF (Windows-MetaFile) is a vector graphics format which serves mainly to exchange graphics among Windows applications.

Call the command either by entering WMFOUT or by selecting **Copy Region to Windows Metafile** from the *Edit* menu.

The following options allow the region to be copied into a bitmap:

Window

Define a window by picking two points in the drawing.

View

The complete current view will be exported as WMF.

Extents

The region to be copied is defined by the extent of the drawing.

Finally, you may choose whether the picture should be written into a file or copied to the Windows clipboard.

Note:
The objects remain in the clipboard until you clear it, or place new objects on it with the COPYSELECT, BMPOUT, WMFOUT or other FelixCAD command, or Cut or Copy in other Windows applications, or exit Windows.

Bitmap Export

Execute the command either by entering **BMPOUT** or by selecting **Copy Region to Bitmap** from the *Edit* menu.

The following options allow the region to be copied into a bitmap:

Window

Define a window by picking two points in the drawing.

View

The complete current view will be exported into a bitmap.

Extents

The region to be copied is defined by the extent of the drawing.

Specify the Width and Height (10...2000 pixels) of the bitmap.

Finally, you may choose whether the bitmap should be written into a file or copied to clipboard of Windows.

Note:
The objects remain in the clipboard until you clear it, or place new objects on it with the COPYSELECT, BMPOUT, WMFOUT or other FCAD command, or Cut or Copy in other Windows applications, or exit Windows.

Copy Selection to Clipboard

The command COPYSELECT copies selected objects from the drawing and places them on the clipboard. The selected objects remain in the drawing. The command is used to copy objects from one drawing to other drawings. COPYSELECT places these objects onto the clipboard. Use the PASTE command to paste them from the clipboard into other drawings.

Call the function by entering COPYSELECT or choose **Copy Selection to Clipboard** from the *Edit* menu. To copy objects onto the clipboard: First specify the base point for later insertion of the objects and then select the objects to be copied to the clipboard. Close the object selection by pressing ENTER.

```
> COPYSELECT
Insertion base point: P1
Select objects: ...
```

Note:
Do not confuse this command with the COPY command found on the *Modify* Menu which make copies of objects in the current drawing. The objects selected with COPYSELECT remain in the clipboard, ready to be placed in another FelixCAD drawing.

They will remain on the clipboard until you clear it, place new objects on it with the COPYSELECT, BMPOUT, WMFOUT or other *FelixCAD* command, or Cut or Copy in other Windows applications, or exit Windows.

If you are copying information to another open drawing, it is recommended that you enter 0,0 as the base point. This point should also be entered when you paste the information into the destination file.

Import a DXF or DWG drawing

If a drawing is to be imported in **.dwg** or **.dxf**, you should select **Open**. Either Pick the open icon, select the command from the file menu, or enter the command OPEN. Then follow these steps:

1. Select the import format from the field ***"List files of type:"***
2. Enter the name of the file to be imported, or select a file from the existing file list, in the input field ***filename***.
3. If necessary, enter the exact path of the import file in the selecting window Directories. Remember, if no path is entered, the existing Directory will be chosen.
4. Confirm the selection by OK or ENTER.

Paste from Clipboard

The command **PASTE** copies objects from the clipboard to a user-defined location in the current drawing. Use the command to copy graphic data from other FelixCAD drawings into your current drawing.

You can copy objects from a drawing to the clipboard with the COPYSELECT command (see above), then paste copies into other open drawings. To copy objects within the current drawing, use the COPY or ARRAY command found in the *Modify* menu. Call the function by entering PASTE or choose Paste from the pulldown menu Edit.

To paste objects from the clipboard into the drawing:

- Choose the PASTE command.
- Specify the insertion point.
- The objects of the clipboard contents appear highlighted, attached to the cursor. Drag the highlighted objects to position them in the drawing.
- Specify the scale factor for insertion.
- Specify the rotation angle for insertion.

Example:

```
> PASTE
Insertion point: P1
Scale factor <1>: <hit Enter key>
Rotation angle <0>: <hit Enter key>
```

When inserting the objects from the clipboard you can also specify the scale factor and/or rotation angle before specifying the insertion point by choosing the corresponding option from the context bar when prompted for the insertion point:

```
> PASTE
Insertion point: Factor
Scale factor <1>: .5
Insertion point: Rotation
Rotation angle <0>: 90
Insertion point: <hit Enter key>
```

Note:
The objects selected with COPYSELECT remain in the clipboard, ready to be placed in another FelixCAD drawing.

They will remain on the clipboard until you clear it, place new objects on it with the COPYSELECT, BMPOUT, WMFOUT or other FelixCAD command, or Cut or Copy in other Windows applications, or exit Windows.

Export / Import Parts

Write Part File ...

Write Part File... works identically to the function **Part Definition ...** (command PARTDEF) but is used to export parts.

Both functions serve to define entities of a drawing or to define the whole drawing as a part. The part names, the insertion point of the part and/ or the drawing entities are entered or selected.

You will find more detailed information about the use of the function **Part Definition...** in Chapter 8, *Parts and Attributes*.

Import...

To import a part select one of these options from the *Part* menu:

Insert Quick,
Merge External Part
Insert Part

You will find detailed information about the inserting of internal and external parts in the Chapter 8, *Parts and Attributes*.

Export Attributes

 With this function, you can extract attributes for parts or entities from the drawing and save these in a Comma Delimited format (CDF) file. This is important when transferring and analyzing information from the drawing to or from programs for further processing.

Proceed as follows:

- Enter the command ATTEXP, or enter the function by selecting the menu point **Export Attributes Information...** in the *Part* menu and then select the option.
- Select the objects whose attributes are to be analyzed and the information which to be exported. The usual object select functions in the Options Bar are available. Confirm the object selection by OK;
- The dialog box shown below will open;
  Those parts which contain attribute values will be selected from the entities selected by the user. The number of the parts will be displayed in the upper area. You can allow this object filtering to be executed for the whole drawing by selecting **Search all parts**; then
- Select the attributes to be exported from the proper selection area.

Select Attributes

The left-most table lists the attributes which have been selected for export. An **N** indicates that it is a numeric value, and a **C** indicates that a character string is to be exported.

The next number displays the length of the field input for the attribute. For numeric values, the number of the decimal digits is also displayed.

Switches for **Adding** and **Deleting** attributes to the selection list are placed underneath the selection table. To delete attributes from the list, mark one or more lines in the list and select the switch **Delete**.

When the **Add** switch is selected, a selecting box with all the attributes existing in the drawing is shown. Select one or more of the listed attributes to add to the list.

Next, determine the field type. The field types may be ordered numerically, or they may not. Enter the field length in characters and define the number of the decimal digits for numeric fields. Note that the definition of the field type is valid for all selected attributes. Confirm the selection of the attributes to be added by OK.

Select Part Properties for Export

You can select which part properties are to be included in the CDF-file in a manner similar to adding attributes.

Select the switch **Adding / Changing** below the display and selection field for the parts and properties.

All of the properties which can be exported will be shown in the selection area. Select a property by selecting the marked field. Enter the desired number of characters for the field width and, if necessary, edit the number of decimal digits.

Change Field Separators

In the CDF file all fields are separated by ***Field separators***. Non-numeric fields are included in ***text separators***. These values can also be changed in this box.

View and Write CDF Files

When you have selected all of the information desired, determine the name of the output file. A list will appear in the field **CDF file**. The switch **'...'** on the right side next to the input field for the file name displays a file selection box.

The switches **Display** and **OK** allow you to search the drawing for the parts with the selected attributes, and the output in the CDF format is prepared. If you select the switch **Display**, a file will not be written, but will be displayed in a tabular Dialog Window.

From this window individual lines, or all lines, can be copied to intermediate storage (Clipboard) so that they can be easily processed by other Windows programs.

The CDF-file is written when you leave the main dialog by confirming your choice by OK. You will get a message if errors have been committed. You can also order display of error descriptions.

Save Settings

A field marked **Save settings** is also available in the Dialog Box. All user inputs there are saved in a parameter file and are at your disposal during the next call for the command ATTEXP.

Chapter 13

Printing and Plotting Using the Layout Mode

This chapter describes how to output a drawing on a printer or plotter.

FelixCAD uses the Windows drivers to print or plot. Before you can successfully print or plot from FelixCAD you must make sure that you have the correct printer or plotter drivers installed.

The commands described in this chapter are:

Function	Command
Printer Setup...	PSETUP
Print / Plot	PRINT
Toggle between Model and Paper Space	TLMODE
Create and Control Viewports...	VIEWPORT

To achieve a sufficient print quality of circles, the program temporarily sets the **circle resolution to 256** when printing if the current value of this setting is less. Otherwise the program uses the value as set in the CIRCLERES system variable.

Printer Setup

The command PSETUP (Printer/plotter Setup) or the selection of the option **Printer Setup...** from the *File* menu allows you to select the desired printer or plotter. This is the standard procedure for selecting output devices in Windows.

You will find detailed instructions regarding the operating mode of the specific options for the printer/plotter installation and for installing and connecting a printer or plotter in the Windows Manual or in the output device manual. Only the steps related to FelixCAD use of the printer/plotter are reviewed in this manual.

Printer

In the field **Printer** specify the printer or plotter to be used. Output devices which are available are displayed in this dialog. To see the complete list, select the arrow at the right edge. From the list of the installed output devices select the desired printer.

To achieve a sufficient print quality of circles, the program temporarily sets the **circle resolution to 256** when printing if the current value of this setting is less. Otherwise the program uses the value as set in the CIRCLERES system variable.

Orientation

Select either ***Portrait*** or ***Landscape*** for the desired paper **orientation**.

Paper Size

The size of the paper as well as the feed from a particular printer is controlled from the field **Paper**. Select the proper size and feed.

Options / Properties

The **Options** or **Properties** buttons open another dialog box which contain parameters that are mainly a function of the capability of the device to be connected. You will find more exact information regarding options and installation possibilities in the user manual of the device.

Print or Plot a Drawing

Start the output of a drawing on a printer or plotter by selecting the option **Print...** on the *File* Menu or by entering PRINT. First, select one of the options available in the context bar.

View	Extent	Window	NView	

These options allow you to determine the size and position of the print zone or the area of the drawing to be output.

View

The currently visible window of the drawing will be copied on the print.

Extent

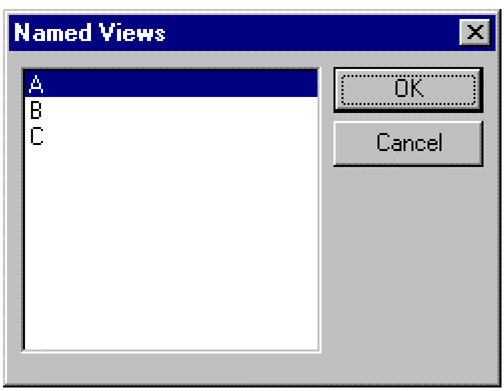
This option determines the print area so that all objects included in the drawing are covered. Note that before using this option the drawing should be regenerated (Window Menu or enter command REGEN).

Window

Window enables you to determine the print area by selecting a window in the drawing. Pick or enter the coordinates of two diagonal opposite corner points inside the drawing.

Named View

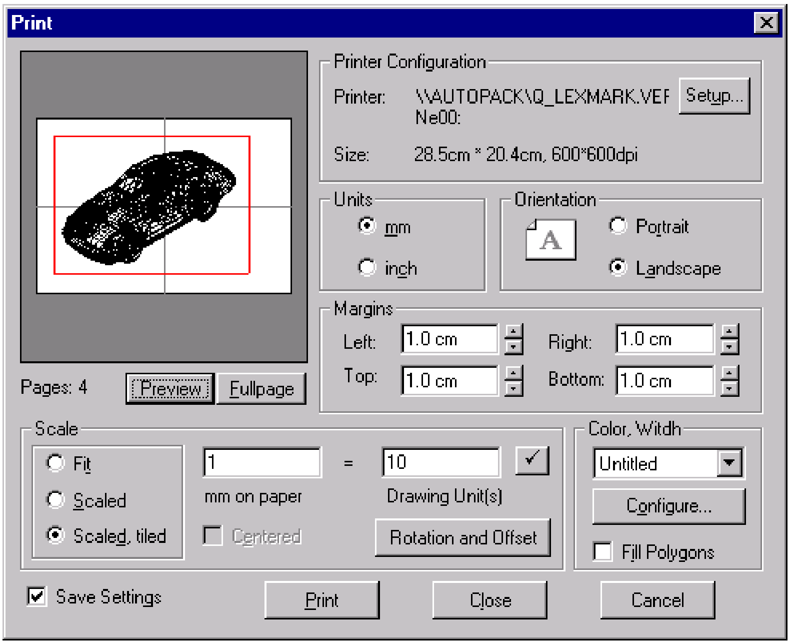
The *NView* option allows you specify a previously saved view from the dialog box *Named View*. It is a good idea to create named views which are saved with the drawing for subsequent print output.



Dialog Box Print/Plot

After defining the size of the print, the dialog box *Print/ Plot* will open, and additional print parameters may be set. The dialog box is divided into the following areas: Configuration, Units, Orientation, Margins, Scale, and Color/Width. It also contains a preview window and buttons for Preview, Print, Close, Cancel, Configure, and Setup.

The **preview window** will assist you in controlling the effects of setting changes in Layout and Scaling.



Printer Configuration Area

The Printer/Plotter configuration area contains the options for the current Printer/Plotter and the desired paper size and print resolution (in dpi).

Setup

Selection of the **Setup** button allows you to control such parameters as the paper feed (printer/ plotter drawer), paper size, format position, resolutions and the number of the copies to be run.

Note that in the Setup dialog box only the settings that your device supports can be edited.

Margins

Controls the area that is not to be printed at the sides of the page. Enter the desired values in the proper window or select with the Cursor at the right edge of the fields and set the values for margin width.

Fit to Margins

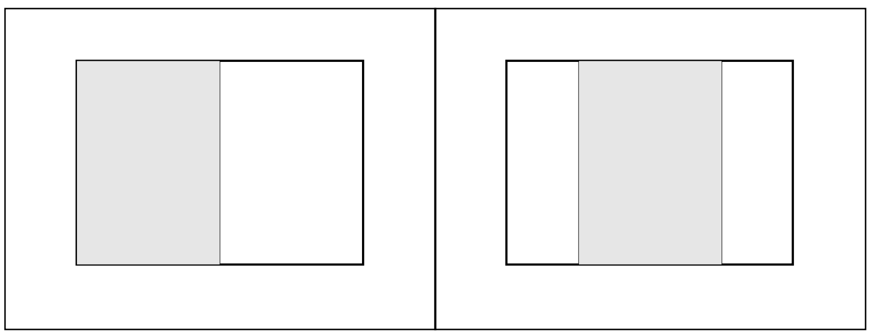
Selecting the option **Fit to Margins** will result in a full picture of the object in the print area.

This option is activated by default during any print or plot output. To de-activate, click the control field. A cross in the control field displays the active status of the option.

Note that when the option **Fit to Margins** is active, it is not possible to set the options for scaling.

Centered

Centered has a very close relation to the option *Fit to Margins*. Because the aspect ratio of the drawing to the paper remains unchanged during *Fit to Margins*, a full picture may only be output either of the length **or** width of the picture, but not with both.



Position of the printed area before (left) and after (right) turning on Center option.

Centered results in a centered placement of the print area on the paper in relation to the side that could **not be fitted** to the margins of the page.

Note that **Centered** is only available when using the **Fit to Margins** option.

Scale Settings

Scale Settings allows you to define the exact scale of the drawing. Note that Scaling is only available when the **Fit to Margins** option has been **de-activated**.

The setup of drawing plot scale is done in two steps:

1. Define the size of a drawing unit. Choose either mm or inches.
2. In the Scale input field enter the scale for the print. For example, if you are working in metric, enter the size in plotted units versus the plan size. If you are preparing a surveying drawing where the plan units are mm and the drawing units are meters, and the plot scale is 1:250, then you would enter 1 = 0.25 or 1 mm = 0.25 m. If you are working in Imperial units with a plan scale of 1 inch = 20 feet, you would enter 1 = 20. The print dialog box shown on a previous page shows this example.

You can see scale setting effects in the preview window when you pick the small check mark to the right of the drawing units input edit field.

Pages

The number of pages that will be printed is shown immediately below the drawing preview window.

Multiple pages will result when you have selected a drawing scale that results in a printout larger than the selected page size (minus the defined margins).

Fill Polygons

Fill Polygons will cause filled polygon areas to be displayed with fill in the print output. If the box is not activated (the default), only the outlines of filled areas will be output.

Color / Width

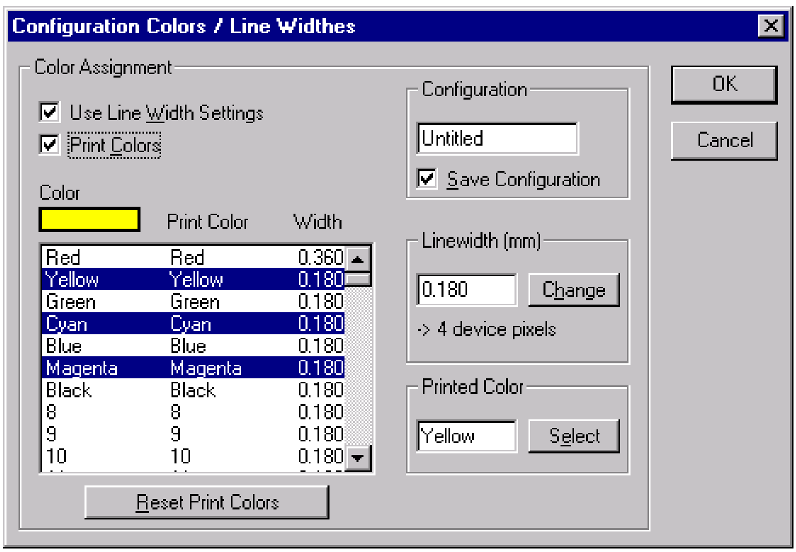
The area **Color/Width** defines the line widths output on the printer/plotter. This choice of line widths allows for the structuring of a drawing to improve its legibility.

The setup of line widths is made by associating the line widths and colors or color identification numbers. The current setting of line widths can be saved under a configuration name and then, like a parameter, re-used for future printing operations.

Every object has been assigned a color that is defined by a color identification number (also called color index). For detailed instructions on the definition of colors for layers or for objects, see Chapter 4, *Layer and Object Properties*, of this manual.

During the preparation of the print out, any color can be given a stroke width defined in mm or inches..

Select the **Configure** button in the **Color/Width** area of the Dialog Box and the **Color/Width Configuration** dialog will appear:



Switching Line Width Selection On/Off

The check box **Use Thickness Settings** determines whether Line Widths will be used.

There is a second check box for **Print Colors** used to switch the color printing On or Off. Of course, the output device must have color capability to print in color.

Attach Stroke Widths

Use the selection area for coordinating color and width.

1. Select the desired color or its color index (color identification number). In the selecting window the color identification number and the present stroke width is displayed. In the preview window **Color Index** you can preview the selected color.
2. Enter the desired value in the input field **Line Width**. Attach the entered stroke width to the previously selected color by clicking the button **Change**.

Save Settings

You can save the correlation between line widths and colors. To do this, enter a name in the area *Configuration* for the current setting of the stroke widths in the input field there. Activate the function **Save Configuration** by selecting that check box.

Using saved stroke width-attachments

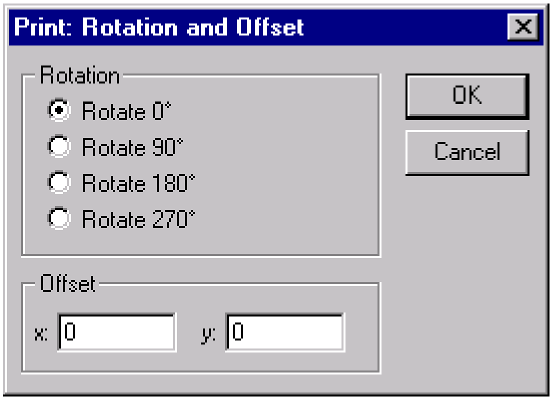
Using saved stroke widths facilitates outputting drawings and assures constant settings.

In the **Color/ Width** area in the **Print** dialog box the current stroke width will be displayed. Selecting the arrow at the right side of the window will display a list of all saved configurations. Select the desired configuration.

Print Rotation and Offset

In the sub-dialog box *Print Rotation and Offset ...*

- ... the Rotation buttons allow you to specify a rotation of the output area on the sheet about 90, 180 or 270 degrees. Rotation is applied clockwise.
- ... the Offset option allows to move the plot origin from the lower left corner of the paper (the normal position) to another location on the paper. To do so, specify the x, y values in the text boxes in paper units (either mm or inch, as determined in the main dialog box).



The settings are reflected in the preview of the dialog box respectively in the fullpage preview

Layout Mode: Process Drawings for Paper Output and Presentations

One main advantage of working with CAD is, that any plan and construction can be created in real world units. However, most jobs will eventually need to be printed on paper. Usually this means, that it needs to be plotted in a certain scale.

For many reasons throughout a job, you may need to print out a drawing that incorporates many different scales for different parts of the drawing, such as;

- to add drawing title blocks,
- to display different views and regions,
- to create details of the model,
- lay out part lists, legends, and annotations and instructions on the drawing sheet.

To serve to solve these tasks in a flexible manner, *FelixCAD* provides two modes to handle a drawing: **Model Mode** and **Layout Mode**. In other words, there are two separate spaces to work on a drawing: Model Space and Paper Space (or Print Space).

The drawing objects in model space constitute the geometry of the model. The objects of paper space reproduce the model in different views or scales and allow you to annotate the drawing.

Note:
Layout mode is not always the easiest way of finishing a drawing. If you create a pure 2D drawing, often it is easier to insert (on separate layers) title blocks, part lists, and details in model space, which are scaled to a certain factor to fit to a certain paper size. Some drawings are created easier at true scale.

When using layout mode you should always remember that this feature is provided to arrange and layout a job for presentation and paper output.

Toggle between Model Mode and Layout Mode

To switch from model mode to layout mode and vice versa, type in the command **TLMODE** (Toggle Layout Mode).

The tiled viewports of the current drawing (displaying the model in different views) are replaced by a single window. The title bar of this window shows the caption ***Layout:*** and the drawing name.

When you switch from model space to paper space the first time, the Layout window is an empty window, which represents a blank drawing sheet. This print space is not blank, if your template drawing already contains paper space objects (in this case paper space usually contains a company title block).

The layout mode made active serves essentially to perform two types of operations:

1. determination of the sheet size for printing/plotting particularly by insertion of a title block;
2. to allow creation of viewports which display individual views of the model.

You can draw in paper space as you used to in model space. All the draw and edit commands are available. However, as paper space is designated to lay out drawing output, a number of 3D commands are disabled in layout mode: 3DVIEW, PLANVIEW, RENDER, HIDE, FHIDE, LIGHT, LIGHTEDIT, SETVIEWDIR, WOPEN, and QWOPEN. As described in the following sections, paper space allows you to arrange individual 3D views of the model by creating and modifying viewports.

The Viewport Command

The command VIEWPORT allows to create rectangular areas displaying the model in print space and to control the view direction or display portion of the model in those viewports. The options to operate on viewports are:

New	ON	OFF	Layer	View		

Next, these functions are described in detail.

Creating Viewports

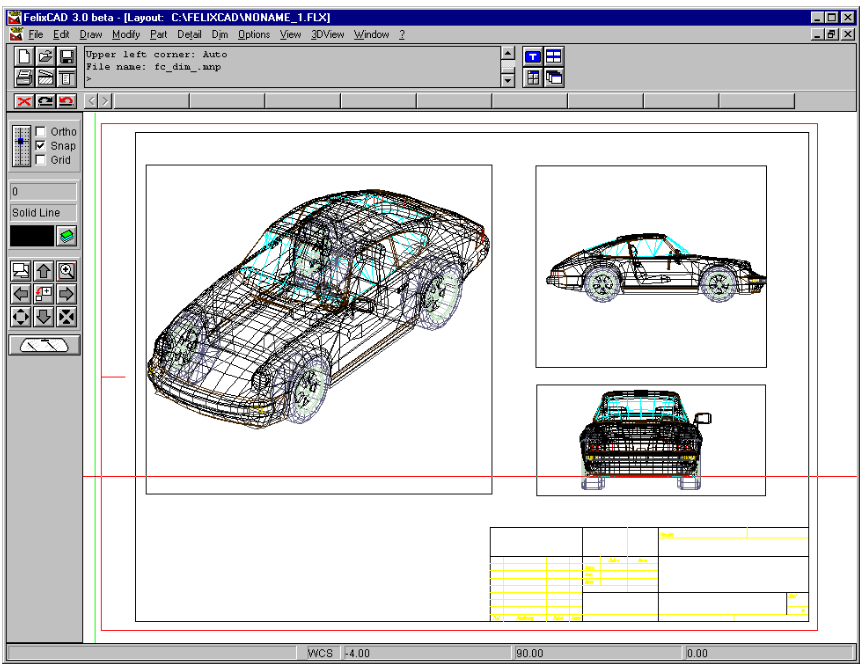
After switching to layout mode for the first time, usually you will create one or more viewports to show one or more views of your model.

The option *New* of the command VIEWPORT allows to create these viewports. *New* is the default option of the command VIEWPORT. After specifying two corners of the viewport, the entire model is displayed in its current extents and in plan view in the newly created viewport.

```
> VIEWPORT
First corner: 10,10
Opposite corner: @45,25
```

The next step will be to determine the correct Zoom ratio to allow the contents of this viewport to be displayed at a certain scale when output to paper. This scale will be directly related to the size of the title block and the plot scale used to plot this drawing.

Detailed hints and tips to apply appropriate techniques to set and keep a scaled relation between paper space viewports and model space are described later in this chapter when all the options of the command VIEWPORT have been discussed.



Viewport Alteration using Modify Commands

A Viewport is a drawing entity represented by an rectangle and acts as a container of a certain view of the construction. Theoretically any number of viewports may be created in model space.

Several editing commands can be performed on a viewport entities, because they are treated as drawing objects. You can move a viewport to another position. You can change the size of a viewport with the Stretch command. In this case the view inside the viewport is not scaled with the viewport. If you scale a viewport the displayed view inside of the viewport is scaled with the same factor as the viewport frame. You can remove viewports from the drawing by using the Delete command. Viewports may be copied. Note that, the commands Rotate, Array, Flip, and Mirror modify the position of a viewport (respectively a copy of a viewport), but the edges of the rectangle remain parallel to the X- and Y-axis of the paper space. Also, these commands maintain the displayed view in the rotated or copied viewport.

When editing viewports and when drawing in paper space you can snap to the end point, mid point, etc. of the viewports rectangle, and you can utilize the coordinate filter options. You can only manipulate viewports while in Paper space.

Turn viewports off and on

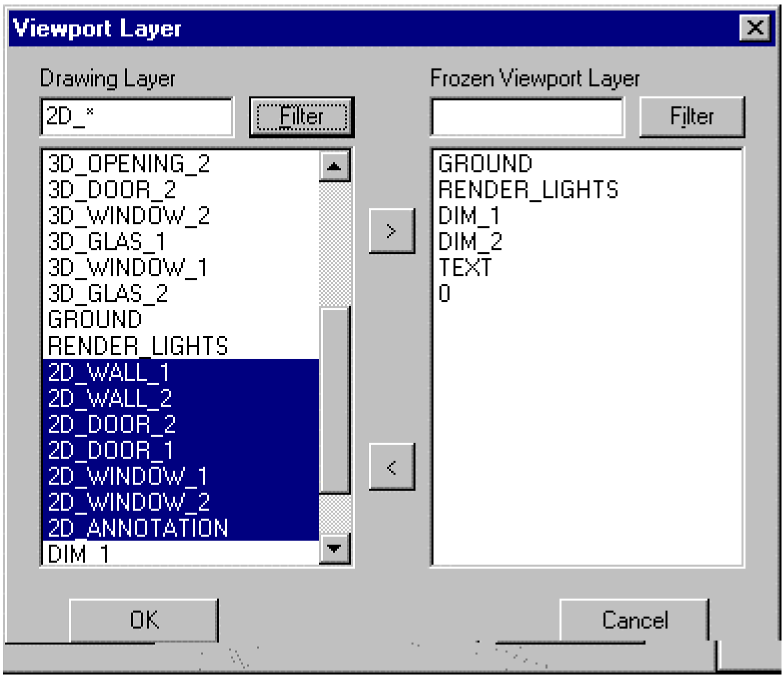
The Viewport command options *On* and *Off* allow to control the **visibility of the contents** displayed in existing viewport windows. To turn the visibility of the viewport objects off, choose the option OFF and select the viewport frame or pick to an object inside the frame. To redisplay the representation of the model within a deactivated viewport, choose the option ON and pick to an edge of the corresponding viewport rectangle.

The **visibility of the viewport frames** can be controlled through layers. Like any other drawing entity, a viewport entity is located on a layer. It is recommended to place viewport entities on one or two specific layers (for example LM_VIEWPORT_ON and LM_VIEWPORT_OFF). Now you can control the visibility of the frames of viewports by turning off or freezing a corresponding layer containing viewports.

Layer control of viewport contents

The VIEWPORT command option *Layer* allows you to manipulate the display of the contents of viewports by freezing and thawing drawing layers. To give an example: The option allows you to display 3D views in certain viewports, with only those objects visible which are located on layers named with the prefix 3D_*, whereas other viewports make use of only layers containing detailed 2D objects, which have been placed on layers using the prefix 2D_*, DETAIL_*, DIM_*, etc..

After choosing the option *Layer* the dialog box illustrated below is displayed.



The list box on the left hand side the dialog box contains all layers defined in the current drawing. The other list box is to depict those layers not to be displayed in the currently selected viewport ("Frozen Layers"). You can alter the visibility mode by marking layer names in one of the two lists and add or remove them from the list of frozen layers with clicks to the > button or < button. To freeze or thaw the status of entire layer groups, you can utilize the edit boxes by specifying strings with wild cards (for example "2D_*") and than click to the Filter button which will highlight all matching layers in the appropriate list.

View control in viewports

The VIEWPORT command option *View* toggles the layout window into a mode, where the selected viewport is displayed that it covers the entire window. This mode serves to change the view to the model inside of the specified viewport.

```
> VIEWPORT
First corner: View
Select viewport: Specify viewport
Switch to viewport view ...
Viewport control:
```

When calling the option *View* the prompt "Viewport control:" is displayed. The following options are at your disposal.

Zoom	Pan	3DView	View		

Zoom

By intention the default prompt at the command line after choosing the Zoom function says "Zoom Scale factor: ". This will remind you, that - once a scaled relation between model space and paper space has been established - you should manipulate the view within the viewport only via scaled zooming (usually by specifying whole numbers) to maintain the scale. Anyhow, in addition the option bar contains the other zoom options (Window, Extents, Previous) as usual.

Pan

The option *Pan* only moves the visible portion within a viewport in a direction specified by two points without changing the scale of the model representation in paper space.

In *Viewport control* mode any transparent command may be used. By this, you can also utilize the pan functions from the control panel (Pan Left, Pan Right, Pan Up, Pan Down); for example:

```
Viewport control: 'PANLEFT
```

3DView

Naturally, the layout mode can be utilized also to illustrate the model from different viewpoints in space. The option *3DView* of viewport control allows you to set individual 3D views or predefined views like plan view, front view, left side view.

The option *3DView* will open a dialog box similar to the one invoked by the command 3DVIEW in model space (see chapter 3).

Note:
Using the option *3DView* will zoom the content of the viewport to the drawing extents. To achieve a scaled relation between paper space and model space for isometric or other individual 3D views of the model, it is recommended that you utilize named views prepared in model space.

View

The option *View* allows you to display views which have been saved as named views in model space in specified viewports. Effectiveness of working in layout mode can be improved by establishing named views in model mode designed for reuse in the viewports of the paper space, especially if maintenance of a scaled relationship between model and print space is required. See the section *Techniques and Tips for Layout Mode* below in this chapter for further hints and recommendations.

The option *View* will open a dialog box similar to the one invoked by the command VIEW in model space (see chapter 3).

A particular help is provided with the *Show* button in the dialog box to Save/Retrieve named views, because this option allows to preview a selected named view in the drawing before confirmation.

Note:
When calling the command VIEW in layout mode outside the command VIEWPORT, you can create named views within paper space (for example to zoom fast to portions of the drawing sheet).

Utilizing transparent commands

You can utilize in *View control* mode all transparent commands. As already mentioned, you can call the functions 'PANLEFT, 'PANRIGHT, 'PANUP, 'PANDOWN from the control panel. Another example is the transparent command 'SETVIEWDIR to set an individual or standard isometric 3D view within a viewport. Or, the transparent command 'DIST, which allows you to measure the distance between two points.

Techniques and Tips for Layout Mode

Create your construction (in model space) in real world units (at scale 1:1). Utilize specific layers for dimensions, hatching, text, etc. When managing and using layers keep in mind, that you might want to illustrate different "logical views" to your model later when presenting or plotting your construction with the aid of layout mode.

At a certain state of the construction process, you will find the need to print out the drawing to present it within your company or to forward it to a manufacturer. In this situation layout mode can help in many cases, to achieve fine paper work:

1. Switch from model space to paper space using the command TLMODE. The title bar now says *Layout:* followed by the drawing name.

2. Set the limits to equal the paper size in those dimensions (units), which will be used when printing (for example A-size, B-size, C-size, ISO-A0, ISO-A1, ISO-A2, ... Letter Portrait, Letter Landscape).

3. Create a layer for the title block (unless not existent), set it current, and insert your title block in paper space at coordinate origin (0,0). Or, draw a rectangle on a temporary layer with the paper size you will use when printing.

4. Specify a layer to place the viewports on. Make this layer current before you create new viewports with the VIEWPORT command and make sure to use this specific layer only for viewports. It is recommended to activate the snap mode with applicable units (inch, millimeter, centimeter, ...) when creating viewports.

5. The next step, once a viewport has been established, is to set up scaling between model and its representation on the drawing sheet. This is the step which sometimes causes headache.
   The task, to set up the adequate scale for displaying the model, can be prepared already in model space in several ways, especially by:

   a. positioning rectangles on a temporary layer in model space, which meet to viewport windows in paper space at a certain scale;

   b. creating named views in model space with the command VIEW, which then can be retrieved within the viewports in paper space with the option *View*.

6. Once the scaled relation is set up, you should avoid uncontrolled zooms in the viewport, and confine yourself to Pan commands and to the ZOOMFAC command (usually specifying whole numbers).

7. To modify the height or length of a viewport, we recommend to use the command STRETCH. The display of the model in the specified viewport will not be scaled with this command. It maintains the scaled relation between model space and paper space.

Keep in mind the "golden rule":

Drawing entities you create in model space can only be edited in model space. Drawing objects you create or insert in paper space can only be modified in paper space.

Chapter 14

Visualization: Hidden Lines and Rendering

A 3D model is created in *FelixCAD* as a wireframe drawing.

With the 3DVIEW command you can display the 3D geometry in isometric views from any individual view direction. Also, when opening an additional window of the current drawing you can specify the 3D view direction of that new viewport.

These commands display any line of the wireframe model, and in this mode you always see those lines which are hidden in the real world. To display more realistic images of your 3D model, *FelixCAD* allows you to create hidden line representations, and to render the 3D geometry of your drawing.

This chapter describes the commands provided for visualization. These are located in the *Window* menu:

Function	Command
Hidden Line Drawing...	HIDE
Fast Hide	FHIDE
Render	RENDER
Render Parameters...	RCONFIG
Insert Light...	LIGHT
Edit Light Parameters...	LIGHTEDIT
Animated Rendering...	ARENDER

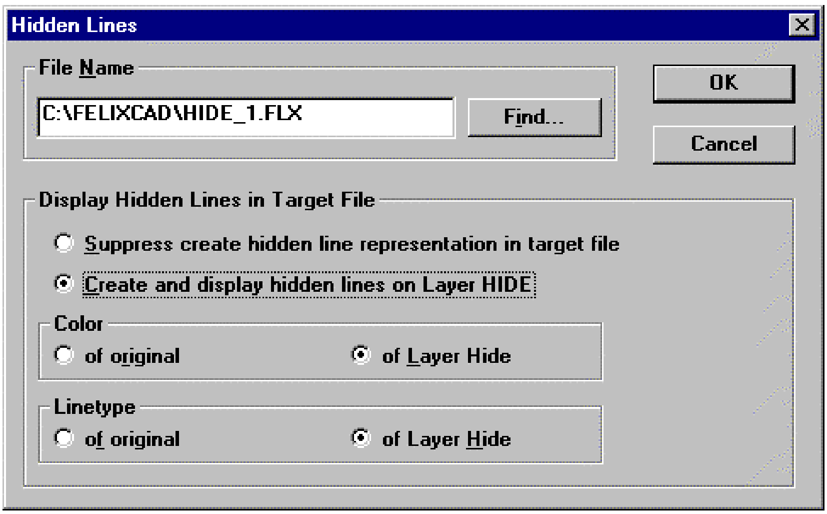
The commands FHIDE and RENDER are based on the OpenGL graphics system from Microsoft which allows high performance hidden line removal and rendering.

Hide

The HIDE command allows you to create a separate drawing which contains a representation of the current view of the drawing with hidden lines removed.

Alternatively, those lines (or edges) which are computed as hidden from the current view point may be placed on a special layer named HIDELAYER in the target drawing. When working with the drawing containing the hidden line representation, you can turn on or off this layer at will or, remove all the lines on that special layer - except those you want to maintain intentionally for demonstration purposes.

Sometimes it is useful not to suppress the hidden lines, but to display all of these lines, for instance to demonstrate certain constraints. In such a case we recommend to set a certain linetype (for example, a dotted linetype) for the layer HIDELAYER in the target drawing.



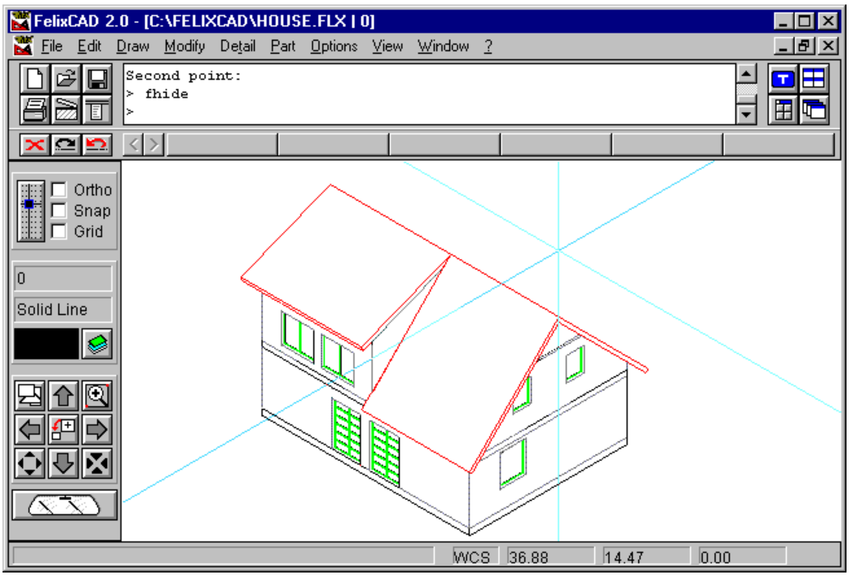
Fast Hide

In contrast to the HIDE command which creates a new drawing, the command FHIDE displays a hidden line view in the current drawing.

This representation is maintained until the drawing window is refreshed by a REDRAW, REGEN, ZOOM, or PAN command. This hide command provides you a utility to get a visual impression of the 3D model from the current viewpoint.

Note that you can continue to draw in this display mode, even utilizing the object snap functions.

The command FHIDE is based on the OpenGL system from Microsoft which allows high performance hidden line removal and rendering.



Render

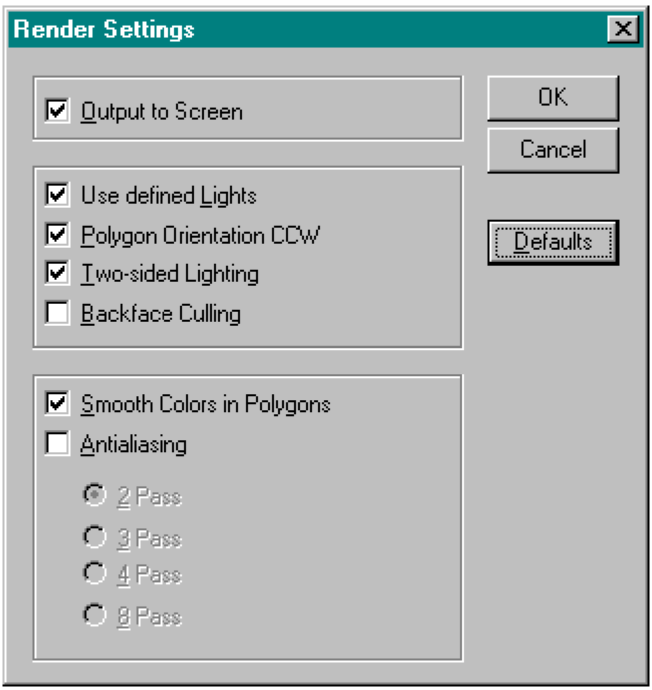
Rendering allows you to create an image of your 3D drawing that illustrates very realistically the model or plan. To increase the realism the rendering may perform smooth shading on the objects that meet at a specific angle and evaluate the lights set in the drawing.

Using rendering you visualize and present your 3D geometry like it would appear as if it has been constructed or built.

To render the current viewport of a drawing call the command RENDER. The command is immediately performed after you have specified where to display or output the rendered view:

Screen	File	Clipboard			

The rendering is based on the preferences set with the RCONFIG command and evaluates the light sources found in the model. The command LIGHT allows you to place lights in the drawing and define their type and settings. The command LIGHTEDIT is used to modify the settings of light sources.



Render Settings

The button **Defaults** resets the preferences to default values (shown in the illustration above). This is convenient, because a number of settings can influence the performance of render computations dramatically.

The option **Output to Screen** controls whether the rendered polygons are output on the screen (to the front buffer) during computations or are written into a display buffer before presenting the rendered image. If this check mark is not set, the rendered image is not displayed before the entire rendered image is computed.

The option **Use defined Lights** allows you to activate or deactivate the use of light sources. If disabled, the program generates - in computer graphic terms - a simple **shading** of the 3D model, otherwise the inserted lights are utilized during the rendering.

Three other options allow users who are familiar with computer graphics and advanced rendering techniques to save time when careful drawing methods have been used during the creation of a 3D model:

- Polygon Orientation CCW
- Two-sided Lightning
- Backface Culling

It is recommended to use the default values if you are not familiar with these terms. The option **Smooth Colors in Polygons** specifies whether upon each individual polygon surfaces smoothing colors are computed or not. Applying an adequate light environment allows smooth coloring on individual surfaces.

The option Anti-Aliasing allows you to control the process of smoothing jagged edges caused by aliasing. Aliasing is a term in computer graphics. It describes certain display effects like the staircase effect seen on diagonal lines displayed in a drawing. Anti-Aliasing can reduce those artifacts.

Turn anti-aliasing on for high-quality rendered images. To improve the rendering significantly you can choose between 2, 3, 4 or 8 passes with the anti-aliasing algorithm. Because multiple passes will increase the rendering time dramatically, anti-aliasing should be applied only for high-quality image output (usually when rendering to a file).

By default the anti-aliasing option is disabled. It produces a faster output for viewing a rendered image on the graphics display, and you do not normally notice the elimination of artifacts on the computer screen.

Lights

The lighting environment in a 3D model is set up with light sources. The command LIGHT allows you to insert lights in the drawing.

Light Types

The render program recognizes four types of light sources:

- Ambient light
- Linear light
- Point light
- Spot light

A **ambient light** is not really a light source. It is an artificial light environment setting. The intensity value of ambient light defines an overall value of lightness for the total 3D model. The value increases or decreases the lightness of a rendering.

A **linear light** radiates light along a line, similar to a fluorescent lamp. It has a position, an intensity and a color. It is used to simulate sunlight in the model.

A **point light** radiates light in all directions. This is similar to a light bulb or incandescent lamp. It has a position, an intensity and a color.

A **spot light** radiates in a specific direction. It is similar to a stage spotlight. It has a position, an intensity, a color, a direction, and a cone angle.

Inserting Light Sources

Ambient light is given any time, but linear, point and spot light sources must explicitly placed in the 3D model. The command LIGHT is provided to insert light sources into the drawing.

Note, that although you can insert as many light sources you can have **only eight light sources turned on**.

When inserting a light into the drawing a dialog box is displayed to allow you to define several settings and parameters for that light, among them the color and the intensity of the light source (see below).

Lights sources are represented in the drawing as complex objects (drawing objects of type block). A light insertion will always be placed on a layer with the name RENDER_LIGHTS.

Light Setting Modification

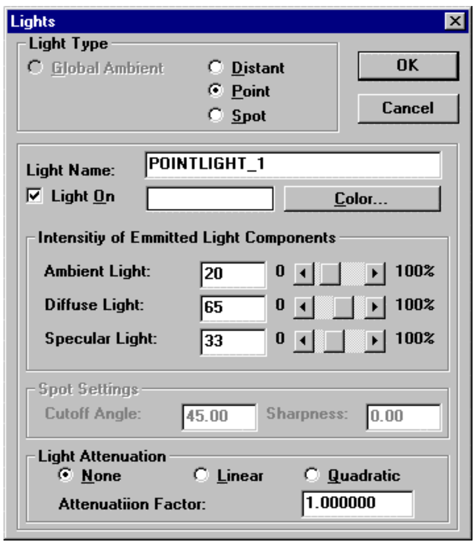
You may modify the settings of an inserted light source later with the command LIGHTEDIT. This command requests you to select a light, represented by a block insertion. Then the dialog box *Light Sources* opens in which you can alter the current preferences for the chosen light.

Light Source Relocation or Removal

To change the location of a light or the lights direction you can use the standard modify commands MOVE and ROTATE. To remove a light source from the drawing call the command DELETE.

Light Sources in Detail

The Light Preferences dialog box is utilized for any light source type. Depending on the light type, certain areas of the dialog box may be grayed out, because corresponding parameters can not be set for the specified light type. The following illustration is an example of specifying the properties and intensity values of a point light. It provides an overview on the dialog box displayed when using the LIGHT command to insert a light source or when using the LIGHTEDIT command to modify the settings of an already inserted light source.



The following discusses the light sources available in detail, emphasizing the differences and the meaning of each type.

Ambient Light

Ambient light is not really a light but defines a sort of overall value of base lightness for the total 3D model.

Ambient light surrounds the entire model to be rendered. It increases or decreases the lightness of a rendering. With ambient light you can avoid that the entire model is displayed too dark.

Ambient light has no direction, which means that all surface polygons in the drawing are illuminated by this type of light uniformly or constant (independent from the view direction). In other words, you can understand it as background lightness.

Ambient light is not represented by an light block insertion in the drawing like other light types. For that reason it is not covered by the LIGHT command.

To modify the value for the intensity of ambient light call the command LIGHTEDIT with the option *Ambient*, to increase or decrease the lightness of a rendering.

The default value for the intensity of ambient light in a new drawing is 20 percent. Note: increasing the intensity of ambient light tends to saturate your image and give it a washed-out appearance.

The command LIGHTEDIT also allows you to set the color of ambient light. The default color is white.

Ambient light may also be turned off. This corresponds to a an intensity of 0 percent. Turning ambient light off simulates absolute darkness. Only other light sources in the drawing will then make drawing objects visible. But, turning ambient light off can be utilized to generate special effects when rendering the model.

Distant Light

Distant light (also called linear or parallel light) radiates light along a line, similar to a fluorescent light bulb. In other words, parallel light emits uniform parallel light rays in one direction only.

Commonly a distant light is used to model sunlight.

It has a position, an intensity and a color. But, the intensity of the light does not diminish with distance.

Light rays extend infinitely on either side of the insertion point of this type of light source.

Point Light

A point light radiates light rays in all directions from a single point, its insertion point in the model. This is similar to an incandescent light bulb. It has a position, an intensity, and a color. The intensity of a point light diminishes with distance.

You can set the manner the light attenuates with distance (none, linear, or square) and the factor for distance falloff in the dialog box in the *Light Attenuation* section. For more information, see the corresponding paragraph below.

Spot Light

Spotlights have light rays emanating in a specific direction. This type of light source in a model is similar to a stage spotlight.

It has a position, an intensity, and a color.

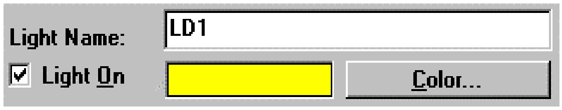
As for point light you can determine light attenuation (distance falloff) for spotlights. In comparison to the other light sources, a spot light has a direction and a cone angle.

To determine the direction of a spotlight, you are requested to specify the location point of the light source and the target point of the spotlight when inserting it into the drawing.

For more information on the light characteristics cone angle and spot light sharpness, see the corresponding paragraph *Specific Spotlight Settings* below.

Common Light Settings

Light source insertions have some common settings and some parameters that apply only to specific light types.



The common parameters described in the following paragraphs are: the light insertion name, the light On/Off status, and the color of the light source:

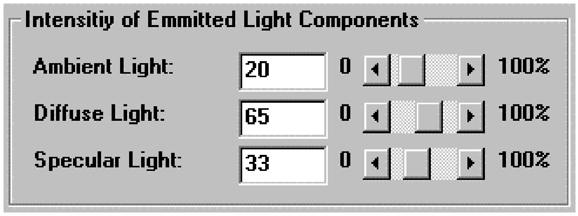
In the **Light Name** edit box you specify a name of a new light source or alter the name of an existing named light insertion. Give each light a unique name. You can refer to the light name when utilizing the LIGHTEDIT command.

The setting of the **Light On** check button determines if the light is evaluated during rendering. Turning a light off means that the render program will not use the light to illuminate the model. The light settings are retained and reused when turning the light status on.

Clicking the **Color...** button will display the standard Windows *Color* dialog box allowing you to define a color for the current light source. The default setting for a new light source is white. Note that dark colors affect the light's brightness.

Intensity Components of Emitted Light

This section of the Light Preferences dialog box allows you to set the intensity of the emitted light. Three components of emitted light are specified in this section: ambient light factor, diffuse light factor, and specular light factor.



The **Ambient Light** component is the light from that source that has been scattered so much by the environment that its direction is impossible to determine. The light seems to come from all directions.

Diffuse Light comes from one direction. It is brighter if it comes squarely down on a surface than if it barely glances off the surface. However, once it hits a surface it is scattered equally in all directions, so it appears equally bright, no matter where the eye is located. Any light coming from a particular position or direction probably has a diffuse component.

Specular Light comes from a particular direction. It tends to bounce off the surface in a preferred direction. You can think of specularity as shininess.

Light Attenuation

In this area of the dialog box you can specify distance falloff for point lights and spotlights.

First, you can determine the manner the light attenuates with distance:

None:
If this option is set, no attenuation is applied for the point light or spot light.

Linear:
If this option is set, linear falloff is applied, which is commonly used for rendering. The brightness caused by the light source decreases in inverse proportion to the distance.

Quadratic:
This option allows to use a **square** falloff, which is more realistic but takes more rendering time. The brightness caused by the light source decreases in inverse proportion to the square of the distance.

Next, if you have chosen linear or square falloff, you can specify an **Attenuation Factor** in the edit box. The default value is 1.

The attenuation factor is applied as follows: Light attenuation at a certain location in the model will be:

- for linear falloff: the inverse of the product of distance and attenuation factor;
- for square falloff: the inverse of the product of square of the distance and attenuation factor.

Specific Spotlight Settings

The following parameters can only be set for spotlights:

The **Cutoff Angle** value specifies the angle for the light cone. That is the angle between the line from the location point to the target point (determined during insertion) and the cone edge. Valid values lie in the range of 0 through 90.

The **Sharpness** value (or spot exponent) determines the attenuation of light sharpness from the spotlight's target line to the edge of the spotlight cone.

Valid values lie between 0 and 128. A valid of 0 causes a sharp accentuated light cone. With any other value, the brightness is highest at the target line of the spot light and decreases to the edge of the cone (by the exponent specified). The maximum value of 128 causes the smoothest integration of a spot light into the environment.

Chapter 15

Raster: Using and Editing Images

Load raster.dll [(xload“raster.dll“)]

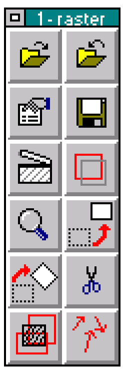
This menu option will load the portion of the program that allows you to bring raster images up on the screen to be used with your normal vector drawings. This program is not loaded automatically because not everybody needs these advanced options. If this program was loaded automatically it would use up some resources needlessly. So when you wish to insert a raster (scanned) image then run this option first to activate the rest of the menu options.

Unload raster.dll [(xunload“raster“)]

This menu option will remove the portion of the program that allows you to insert and manipulate raster images. Unloading this routine will remove all raster images from your drawing.

Raster Palette

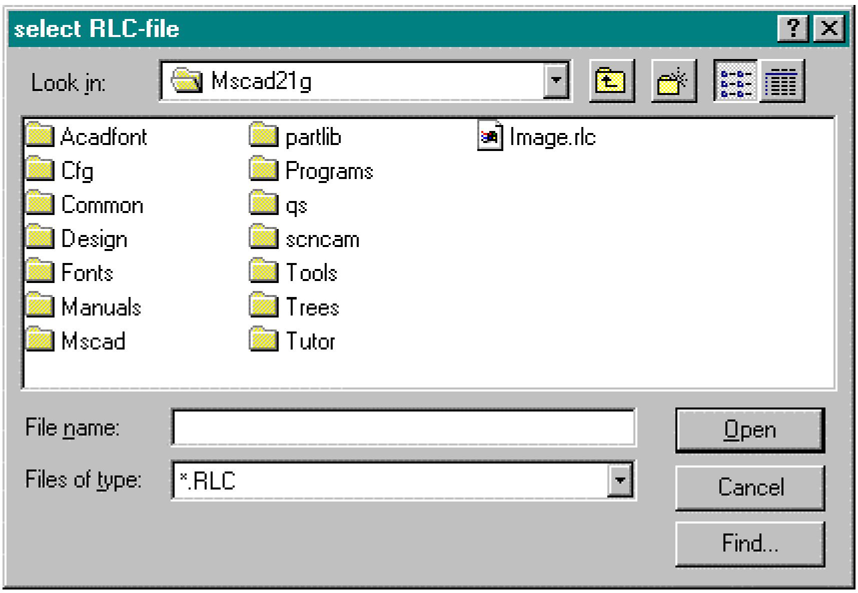
The Raster pulldown menu has it’s own palette or toolbox with most of the commands available on it. The toolbox is only active when the Raster.dll is loaded, even though the toolbox remains on screen. Here is what the toolbox looks like.



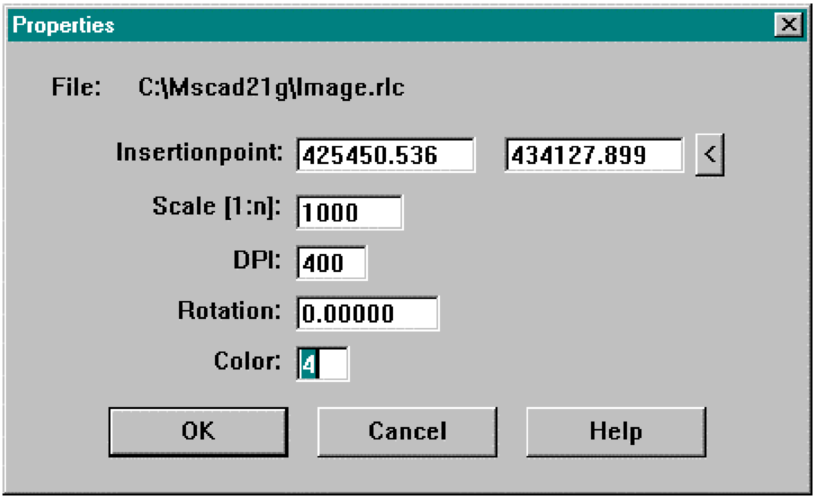
Insert RLC Image [rladen]

This menu item allows you to insert a scanned, raster image, that has been stored in the RLC format, into your current drawing thus allowing you to see the raster image and the vector information at the same time. There are many different formats that a scanned image can be stored in such as; BMP. GIF, CALS, TIFF, etc., and RLC. At this time the program will only support RLC format. There are many conversion programs on the market that may assist you in changing from another format to RLC.

When you start the routine you will be shown the following dialogue box to allow you to select the raster image you wish to insert into the drawing. You can currently only bring in 2 raster images at the same time.



Once selected you will be asked where to Insert the image, the Scale to insert it as, the Colour of the image, the DPI and Rotation factor. These questions are asked on the following dialogue box; (same as the options in edit image properties)



Once you have entered the correct information, your raster image will then appear on screen. You may need to zoom to the correct location to see it using either the regular zoom commands or the new RZOOM command.

Remove RLC Image [runload]

This Option will remove the selected image from the drawing and optionally allow you to save any changes to the properties of that image. The following prompt is displayed;

save properties for ’C:\Mscad21g\Image.rlc ?’:

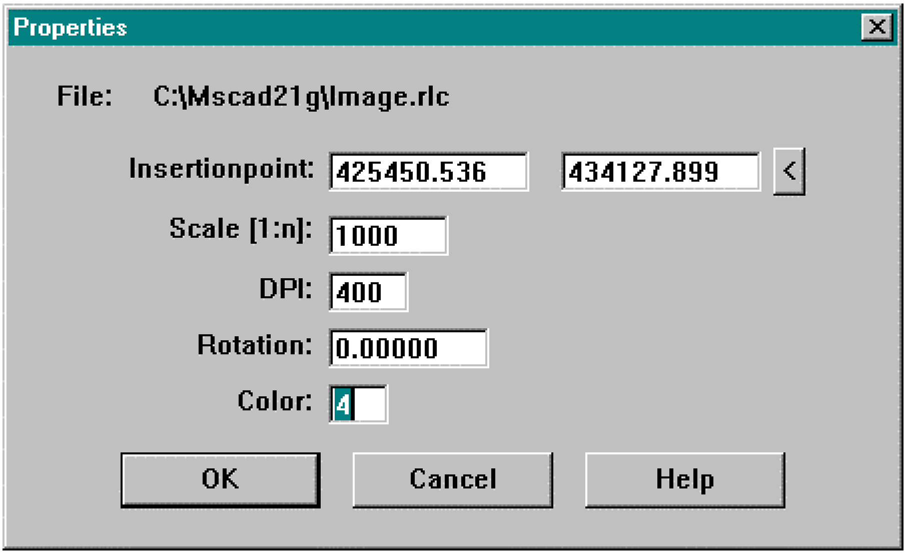
If you answer NO then the image is simply removed from the screen and the properties remain unchanged. If you answer YES the following is displayed;

properties saved for ’C:\Mscad21g\Image.rlc’.

and then the image is removed from the drawing.

Edit Image Properties (rprop)

Brings the following dialogue box up and allows you to change any of the fields. The changes are only done in the current drawing and will not affect the original image on the hard drive.



Insertion point: Location to insert the image on your drawing.

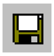
Scale [1:]: Sets the relative scale factor to match your drawing. Scales the raster image larger or smaller.

DPI: Dots Per Inch, Affects the resolution of the Image.

Rotation: Affects the orientation of the image in reference to the drawing. Positive rotation is counterclockwise, negative rotation is clockwise.

Color: Sets the color the image will take on when inserted into a drawing. The whole image is the same color. Each raster can be brought in with it's own color.

Save Image + Properties [rsich]



Allows you to save all of the changes to the image, done by the edit image properties menu item, back to your hard drive. The following message is displayed.

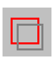
properties saved for ’C:\Mscad21g\Image.rlc’.

These changes are permanent so be careful and make sure you have a backup of the original raster image before saving the changes.

Zoom RLC Extents [rzoom]

This command allows you to zoom to the extents of the raster image(s). If you have lines drawn beyond the extents of the raster image(s), you will need to use the standard zoom and pan commands to augment this command. Lines beyond the extent of the raster image(s) will not be zoomed to with this command.

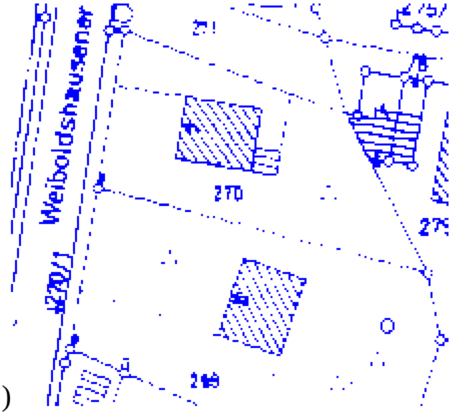
Frame On/Off [rborder]



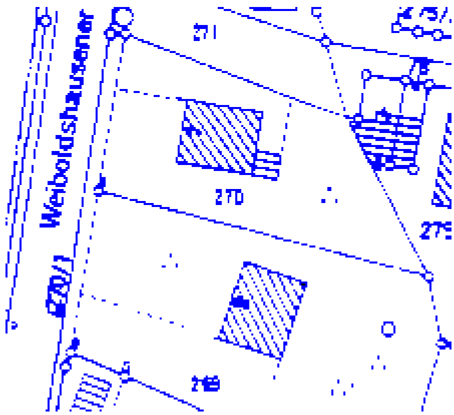
The rectangular frame border around the images can be turned off or on by using this command. All frames are affected at the same time. When on the frame shows the outer edge of the image and is in the same color as the image.

Display Skip Factor (1-100) [rskip]

A relative number of dots used to draw the image on screen.



(Skip Factor set to 1)



(Skip Factor set to 100)

The first image has the Skip Factor set to 1 (one) where the second image has the Skip Factor set to 100 (one hundred). You can see that the number of dots used to draw the images increases as the Skip Factor rises.

List Info on Raster Images [rinfo]

Below is a sample of the information that is displayed;

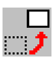
(("C:\\Mscad21g\\Image.rlc" 7521 7672 (10.000000 20.000000) 250.000000 400.000000 0.000000 5))

If there are more than one raster images on screen then all of them will be listed at the same time, in the same format as above.

An explanation of the above format;

"C:\\Mscad21g\\Image.rlc"	= Name of Raster Image
7521 7672	= Original Origin on Raster
(10.000000 20.000000)	= Insertion point for Raster Image
250.000000	= Scale Factor of Raster Image
400.000000	= Dots Per Inch
0.000000	= Rotation Factor
5	= Color of Image on Screen

Move Image [rschieb]

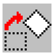


As the name suggests, this command moves an image on your screen. If you have more than one raster image in the drawing, you will be asked to select the one to be moved in the following prompt;

select raster /<point>/?/Last/#n [#0]:

You can point at the raster image, choose Last to pick the last one selected, or enter in the number of the image and the ? mark displays all the numbers to choose from. You will then be asked for a Base point and a Destination point, just like the standard move command.

Rotate Image [rdreh]



You can rotate any raster image to match your vector drawing by using this command. The prompt is as follows;

select raster /<point>/?/Last/#n [#0]:

This allows you to select the correct raster image to rotate, similar to when you move a raster image.

Rotation angle:

This is the amount of rotation, clockwise is a negative rotation and clockwise is a positive rotation.

Mirror Raster Image [rspiegeln]

This command allows you to mirror image the image of your choice. You must first select the correct image on screen;

select raster /<point>/?/Last/#n [#0]:

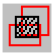
After selecting the image, you will be asked;

mirror axis:

There will be two options showing on the button bar - Horizontal and Vertical. Pick the one you wish to use. You now have the option to edit image properties

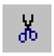
. The original image is not removed and the mirrored image will be displayed where and how the image properties are set.

Copy Portion of Image [rmont]



This command lets you copy a portion of or the whole raster image on screen by putting a window around the portion to be copied. This is similar in methodology to the Zoom Window command. You will be prompted for the First Corner and then the Second Corner. After picking the area to copy you will be asked to confirm the properties of this new image. The changes are only for this drawing unless you save the property changes.

Cut with Pline Boundary [rcut]



If you have a portion of the raster that you do not want and you wish to remove it from the job you can use this command to do so. You must first draw a closed polyline around the area to be clipped. The polyline MUST be a series of straight line segments, arcs will be treated as if the chord of the arc was the boundary, and you MUST close the polyline.

You will be prompted to pick the raster image to clip, just as in the move image and rotate image commands.

select raster /<point>/?/Last/#n [#3]:

Once it knows the correct raster image you will be asked to pick the polyline to use for the clipping. Then the program will ask you

save properties for 'C:\Mscad21g\Image.rlc ?':

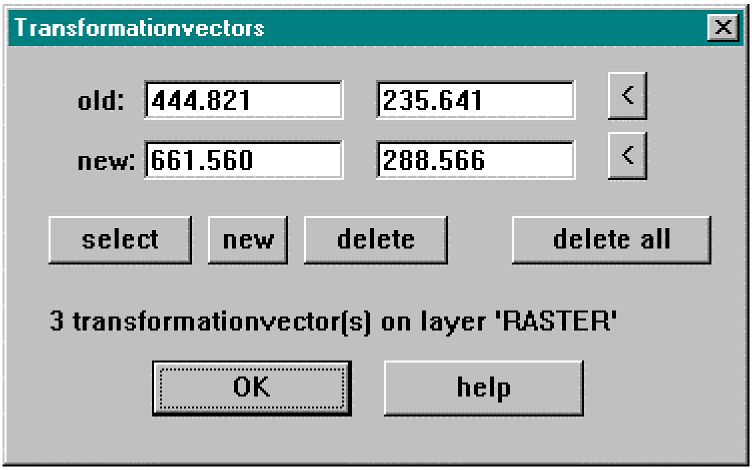
If you answer NO then the image is simply clipped and that portion falling within the polyline boundary is removed from the screen and the properties remain unchanged. If you answer YES the following is displayed;

properties saved for 'C:\Mscad21g\Image.rlc'.

and then the image is cut or clipped inside of the polyline and the properties are saved for the original raster image.

Edit/Define Transformation Vectors [rvektoren]

This dialogue box allows you to enter and edit the transformation vectors required to adjust the raster image to match the vector drawing. The vectors you define will be drawn on the layer RASTER and you can actually draw the vectors before starting this command and it will detect them for you.



Old Vector Coordinate

This is displaying the current starting location of the currently selected transformation vector. You can type in a new location if you wish or use the button to the right to allow you to visually pick a location on the screen.

New Vector Coordinate

This is displaying the current ending location of the currently selected transformation vector. You can type in a new location if you wish or use the button to the right to allow you to visually pick a location on the screen.

Select Button

This button will let you select which vector you are currently editing by allowing you to pick that vector on screen.

New Button

This button will create a new vector from the coordinates displayed in the OLD and NEW fields above. It will duplicate a vector if you pick this button while old values are in the fields above but you can still manipulate those vectors by using the two pick buttons to the right of the NEW and OLD coordinates displayed.

Delete Button

This button will delete the currently selected vector, both from the routine and the screen.

Delete All Button

This button will delete all the vectors on the layer RASTER.

Raster Layer Display

This location displays the number of vectors detected on the layer RASTER**. Do NOT use this layer for anything else or you will confuse the program.**

Pick Button for New Coordinate

This button allows you to pick a new location for the vector to end. This is the ending location of the transformation for this vector. You can pick any where on the screen as the original coordinate for the vector.

Pick Button for Old Coordinate

This button allows you to pick a new location for the vector to start. This is the starting location of the transformation for this vector. In other words the amount of transformation will be determined by the start and end of the vector. The amount of transformation is the delta X and delta Y of this vector. You can pick any where on the screen as the starting (old) coordinate for the vector.

4-Point-Transformation [c:rtrans 4]

This simple transformation allows you to stretch the raster image to match your vector line work. You are required to enter 4 vector translations in the edit/define transformation vectors menu. Then when you select this option the stretching of the raster will be done. Make sure you enter exactly 4 transformation vectors as this routine will not run with more or less than four! Once the transformation is completed you will be prompted for the properties to confirm the insertion point color, etc.

Print/Plot Sorting [rprintopt]

For printers and plotters that can output raster images (inkjet, laser, electrostatic, dot matrix, etc.), you can control if you wish to have the vectors printed on top of the image or the other way round. This way you can see the vectors on top of the image if desired. The prompt is as follows;

print raster before vector or after vector [Before]:

Valid replies are B for before or A for after.

Now you can go ahead and plot your diagram as you normally would and the raster will now print/plot as you have indicated.

Chapter 16

Customization: Palette and Resource Manager

This part of the chapter will discuss various functions which allow you to arrange the program surface.

These are: The **Palette Manager** and the **Resource Manager**

The following table contains a summary of the available commands found in the menu *File*:

Function	Command
Palette Manager	PALMAN
Palette ...	PALETTE
Menu ...	MENU
Load Tablet Menu ...	TABLET
Load Tablet Section ...	TABSECTION
Execute Macro ...	MACRO

The remainder of this chapter is devoted to a short description of each function, or a reference to the paragraph of the manual where you can find more detailed information about the particular command or function.

Palettes

Palettes (or toolboxes) are symbol groups which represent a number of commands. You can open up to ten palettes simultaneously and place them anywhere on the screen.

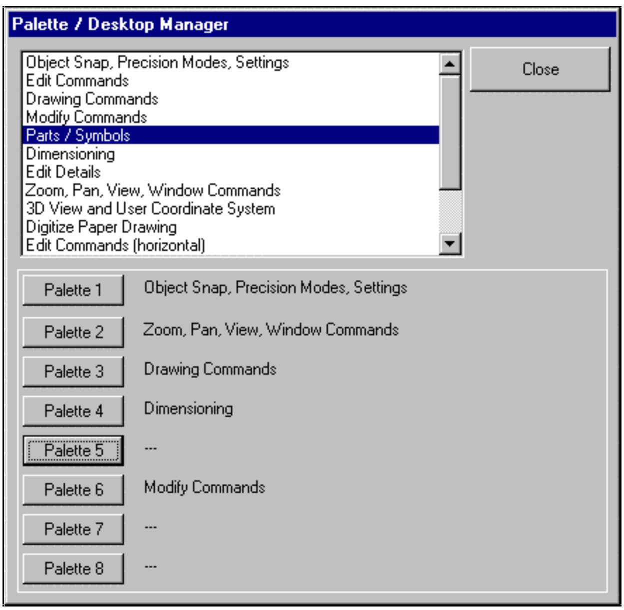
Using Palettes allows selected commands to be accessed quickly. To execute these commands, only cursor selection is required.

Normally, the commands of a given pull-down menu might be contained in a single palette. However, the program offers some palettes in which the commands from several pull-down menus are contained for practical reasons.

Using the *FelixCAD* **Dialog and Menu Editor** it is also possible to create one's own palettes, or to customize available palettes to particular tasks. Detailed instruction in this customization is contained in the *Programmer's Guide.*

Palette Manager

You may control palette opening and display and the exchange of palettes on the program surface. Select **Palette Manger...** from the menu *File*. The following dialog box will open:



In the selection area of the Box, all of the palettes which are currently at your disposal are shown. All palette files with the extension **.mnp** are included in the directory defined during the configuration of the program as a Menu Directory. Instructions for this configuration are found in the *Programmer's Guide*.

Select the desired palette. Underneath the selection area are ten switches with the designations "Palette 1", Palette 2", and so forth, to "Palette 8". By selecting one of the ten buttons you can assign an identification number (for example, Palette 1) to the palette. If there is another palette with the same ID-number already opened, this palette will be closed and replaced by the new palette having that ID number.

When the palette is opened the selecting window will close. If no palette is to be opened or replaced you may close the window by selecting **Cancel.**

To include customized palettes in the Palette Manager, proceed as follows:

Edit the file ***Menu name.pal***, which is situated in the application directory. For instance, if your menu is called ***fcad.mnu***, you must edit the file ***fcad.pal***. The structure of the file is:

```
Palette description 1
palette1.mnp
Palette description 2
palette2.mnp
Palette description 3
palette3.mnp
```

and so forth.

The palette description will be found in the list shown in the Palette Manager Dialog Box, and the line following will always name the palette file. You can complete or delete the entries as you wish.

Commands for Palette Control

PALETTE

The command PALETTE serves to open and place a palette on the program desktop.

The difference between PALETTE and the Palette Manager is in the procedure. The procedure for using PALETTE is:

- PALETTE allows you to select palettes from any directory. It is not necessary that the palette files are situated in a particular directory.
- PALETTE can only be executed if *less then ten palettes* are opened. When the maximum number of ten palettes is opened the message ***A maximum of ten palettes is available!*** is displayed and the operation of the function is interrupted.

Type in command PALETTE or by select the palette symbol in the toolbar.

From the dialog box that is opened you can select the desired palette. On the right side of the selection area you will find the standard selection window for the change of drives and/or the directories. Confirm the selection by OK.

A palette can also be opened by customizing. In this case you have to select the file format **.mnp** in the later available file selecting window. The selection of the palette is then made according to the procedure described above.

Every palette opens in its own window. In the title bar of the palette window you will find the name of the palette as well as an ID-number. Palettes can be moved, rotated, and placed anywhere on the screen.

PALCLOSE

To remove a palette from the screen use the command PALCLOSE. Specify the Palette ID number (P1, P2, P3 etc.) displayed in the title bar of the palette. The corresponding toolbox will be closed.

Menu...

The program is designed so that it is possible to work with different pull-down menus. The pull-down menus are located in menu files with the extension **.mnu**.

It is also possible to create your own pull-down menus using the program module **Dialog and Menu Editor**, and then to store them in available menu files, or make changes in them using the proper change procedures. You will find detailed instructions on this procedure in the *Programmer's Guide*.

To load a pull-down menu follow this procedure:

- Select, from the submenu *Menu/Customization* under the *File* menu, **Menu...** or call the function by selecting the menu symbol in the functions bar or by input of the command PULLDOWN: then
- Select from the dialog box the desired menu file.

Confirm the selection by OK or leave the selecting window by ESC.

Dialog and Menu Editor ...

With the Dialog and Menu Editor *FelixCAD* provides a powerful tool for customizing the program. This item in the menu (executing the command DLGEDIT) allows to call the dialog and menu editor. Within a file dialog box specify the type of the resource file to be edited:

mnp	Palette file
mnu	Menu file
dlg	Dialog file

Detailed information on these items of customization and about the dialog and menu editor are provided in the *Programmer's Guide*.

Macro ...

Using macros automates the execution of a particular sequence of operations and minimizes repeated calling of the same command sequences and the same value entries. You may save the command sequence and the entry of values in a file which can be executed as many times as you require.

The procedure for this is as follows:

- To execute a macro select the Run Macro command from the submenu *File / Resource Manager*;
- Select the desired macro file (extension .mcr) and confirm the selection by OK.

There are further instructions for creating macros in the *Programmer's Guide*.

Load Lisp File...

The command LOADLISP allows you to load an FLISP file via a file dialog box. Detailed information on the built-in Lisp Interpreter is found in the *Programmer's Guide*.

Editor...

The command EDITOR calls the Windows Notepad editor allowing you to edit Lisp files or other resource files (like linetype or hatch pattern files). A file dialog box to specify the file type and file name is displayed when invoking the command.

Display Text-File...

The command LISTFILE displays a specified file in a list box. This allows to read or investigate text or resource files without editing them.

Tablets and Digitizers

The program supports the use of a digitizer tablet as an input medium and for cursor control. Coordinates for drawing points can be entered as well as commands and functions by the proper symbols.

To control the tablet it is necessary to load a Tablet Menu file. This load is executed automatically during the program start if the tablet configuration has been selected as a portion of the program configuration.

The function **Menu...** enables you to change the tablet menu file in the same fashion as the previously described exchange of the pull-down menu file.

To load a tablet menu follow this procedure:

- Select **Menu...** from the submenu *File / Resource Manager*. You can also select the **Menu** symbol in the function bar or input the command TABLET; then
- Select the desired tablet menu file from the dialog box by changing the file type to .mnt
- Confirm the selection by OK.

A tablet can also be loaded by selecting the symbol for program customization. In this case you must select the file format **.mnt**. The remainder of the selection is made according to the previously described procedure.

Replace Tablet Menu Section ...

The option **Replace Tablet Menu Section** enables you to load an individual section of the tablet menu, changing the commands and functions of the complete menu.

To do so, enter the command TABSECTION. There is also a small palette made for the tablet configuration. Using the palette manager, load the Tablet Menu palette.

Determine which tablet menu section should be loaded. Identify the desired section by its section number. Enter the name of the tablet menu file that contains the defined area.

```
Replace Tablet Menu Section
ID (S1...S8) or section name: S1
File name of the new tablet menu-description: tabmenu1
```

The identified area with the commands and functions from the entered file will be restored.

Configuring a Tablet

FelixCAD uses the WINTAB system within Windows. You must contact your tablet manufacturer for the necessary Windows device drivers. When the tablet is working as a standard Windows pointing device, then it can be configured correctly for FelixCAD.

FelixCAD supports the tablet in two different modes: Digitizing and Command Entry. When set to Digitizing mode, you can copy existing drawings into FelixCAD with as much accuracy as your tablet can provide. When used in Command Entry mode, the tablet is used to position the cursor on the drawing window, select commands from menus and tool palettes, and it can optionally be used to select commands from menus on the tablet itself.

The tablet uses a menu file with the extension .mnt. A sample has been included with FelixCAD called TEMPLATE.MNT. This tablet menu is currently empty. The tablet can contain up to 8 menu areas.

Tablet configuration is done using the TABCONFIG command. This command loads a dialog that is used to define the screen area location on the tablet, and the menu locations on the tablet. The number of rows and columns for each screen area is defined in the tablet menu file.

Setting up the tablet areas can be done interactively, but picking the bottom left and top right corners of each area, or manually by directly entering the digitizer coordinates in the dialog. Most users will prefer to do this interactively.

Digitizing Existing Drawings

To digitize existing drawings you need to have two defined coordinate locations on your drawing. Place your drawing on the tablet and tape it down so it cannot move. Enter the command DIGICONFIG. The program will ask for the value of two coordinates and then ask you to pick the corresponding location on the plan. You can turn on and off the digitizing function with the commands DIGION and DIGIOFF. The program has a digitizer palette that can be activated to make the process easier. It is called the “Digitize Paper Drawings” palette in the palette manager. You can load it quickly by entering PALMAN at the command line.

Editing Tablet Menus

The tablet menu commands are stored in an ASCII file that can be edited using any standard text editor, like the Windows Notepad. If you edit the file with your own word processor, make certain that the file is save as “pure” text only. Word processors normally store files in a proprietary binary format.

To modify a tablet menu file, you need to edit 2 main areas of the file: the SETTINGS which defines the row and column layout of the menu areas, and the FIELD lines, which define the exact commands that will be executed when you pick with your tablet cursor pick button.

The SETTINGS Section

The header looks like this:

```
SECTION
      SETTINGS
      BEGIN
         SECTIONIDENT = "Section_1"
         Columns      = 10
         Lines        = 10
      END
```

The SECTIONIDENT defines the name of this menu section. The Lines and Columns define how many rows and columns will be immediately below this section. You can have up to 8 sections in the menu file.

The FIELDS Section

Here is an example of a field line:

```
TABLETITEM = "A_01",{Help Status Bar | Help-Item},[FCMD]^^COMMAND
```

The “A_01” defines the row and column item.

The “Help Status Bar” defines the information that will be displayed when the cursor selects the item.

The “Help-Item” defines the help item in the help file.

The “[FCMD]” identifies that a command is coming and should not be modified.

The ^^COMMAND defines the command that will be executed. Normally you prefix all commands with the ^^ symbol which represents pressing the ESC key twice to cancel any current commands.

Here is an example of what you might enter if you want to execute the Line command:

```
TABLETITEM = "A_01",{Draw Line(s) | LINE},[FCMD]^^LINE
```

Note that this structure is *very* similar to the menu files (.mnu) structure.
If you need more examples of how commands are executed, you can edit the fcad.mnu file. Be VERY careful not to modify the fcad.mnu. You should copy this file to another location and learn from it there.

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