



# FelixCAD™ 5 LT

## User Guide

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**Please note** that this free version of FelixCAD 5 has a reduced command set. FelixCAD 5 LT does NOT contain DWG export functionality, 3D functions (except Poly3D command) and programming interfaces for C/C++ and LISP which are available in the full version of FelixCAD 5. All functions unavailable in FelixCAD 5 LT are marked with a red asterix (\*).

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# Introduction

**FelixCAD™** is an advanced CAD system designed specifically for 32-bit Microsoft Windows™ 95, 98, and NT™ computers. Designed for fast and effective solutions of design projects, FelixCAD offers all the power, precision, and ease-of-use that today's CAD user demands.

For commercial, corporate, and government application developers, FelixCAD is available as the low cost **Graphic Developer's Engine™** (GDE™). For information on how you can utilize GDE in business or graphics applications of your own design, visit [www.fcad.com](http://www.fcad.com) (North America) or [www.fcad.de](http://www.fcad.de) (outside North America).

## Range of Capabilities

FelixCAD includes all the commands 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** make short work of drawing tasks. You create drawings with 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** (blocks), and helps you build libraries of parts for use in repetitive drawing tasks.

Drawing entities may be altered 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 designs.

**Attributes** of any part may be defined or edited, insuring that all pieces making up a drawing contain intelligence for later use.

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

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

**Layers** help you organize your work. This improves drawing management, and makes it easy to create effective, clear, and well-detailed drawings. FelixCAD conforms to the layer conventions used by AutoCAD<sup>®</sup>, from Autodesk Inc.

Instead of fumbling with a complex system of dividing the drawing into different viewports to see different views and perspectives, FelixCAD allows you to open up to four drawings, each with up to four viewports. All views are displayed simultaneously on the screen while you work on the “active” view. You may have as many as sixteen viewports open on the screen during the drawing process.

FelixCAD incorporates simple to use static and animated rendering of 3D objects, along with light source definition and placement.

At every step, on-line **help** supports you. In addition, a useful and detailed on-line **tutor** graphically illustrates basic geometry behind various commands. The tutorials allow you to call up an illustrated description of the command, see how it works, and work through an exercise. The tutorials includes a discussion of setting up the drawing, the user coordinate system, the layer concept, the object snap, and a few basic drawing exercises.

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 software, and analysis programs.

FelixCAD allows the **opening** and **saving** of drawings in industry standard DXF<sup>®</sup> format or DWG<sup>®</sup> format, transferring graphic and non-graphic information without destruction or loss of data. This ensures transparent file exchange with a variety of CAD programs and applications, including AutoCAD.

## Flexible and Adaptable

The configuration and customization commands allow many other possibilities for the program. You manage the use of colors, scripts, input, and output media, and hotkey assignment of function keys – just to name a few examples.

FelixCAD is very flexible with regard to configuration of its user interface. You can choose between a standard pull-down menu, an AutoCAD “work a like” menu, or create one of your own.

## Menu, Palette, and Dialog Box Customization

FelixCAD contains an integrated **Dialog and Menu Editor (DME)**. This editor adapts menus to specific tasks, and creates brand new menu structures. Most importantly, these changes do not require any programming knowledge.

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

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

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

## FLISP Programming

The integrated FelixCAD LISP interpreter (FLISP) facilitates the adaptation and extension of FelixCAD. It also opens up the possibility of working with other programs, utilities, routines, and libraries.

The function library of the FLISP interpreter includes functions to display dialog boxes and evaluate user input. You can create your own dialog boxes with Dialog and Menu Editor.

FelixCAD includes utilities to assist in converting existing AutoLisp<sup>®</sup> programs written for use with AutoCAD. Utilities exist to convert dialog boxes, menus, and programming code.

## Developer's Toolkit Manual \*

FelixCAD includes an API (application programming interface), which allows professional developers and advanced users to access nearly all FelixCAD functions through the use of dynamic link libraries (DLL).

The FelixCAD Developers Toolkit (FDT) manual is included in electronic format for the convenience of advanced developers wishing to extend the program's capabilities using C, C++, or Delphi programming languages. The API manual can be opened and/or printed using Adobe's Acrobat reader (included with CD versions of FelixCAD, downloadable from [www.fcad.com](http://www.fcad.com) or [www.adobe.com](http://www.adobe.com) for on-line Internet purchases). Search the \manuals folder for any file starting with FDT, and having the extension .pdf.

## DOCUMENTATION

FelixCAD includes this comprehensive *User Guide*. In addition, on-line documentation is included for all FelixCAD customization and programming, and for existing AutoCAD users. Of note are the manuals detailing program customization, FLISP programming, and the FelixCAD Developer's Toolkit (FDT), an advanced C/C++ language programming interface available with FelixCAD.

All on-line manuals for FelixCAD are saved in Adobe Acrobat Portable Document Format (files end with the extension .pdf). To view or print the on-line manuals, you must first install the Adobe Acrobat Reader (version 4 or higher), which is contained on the installation CD, or available for download off the Internet at [www.fcad.com](http://www.fcad.com) or [www.adobe.com](http://www.adobe.com).

## User Manual

This *User Manual* contains instruction on the layout and general functions contained in FelixCAD. It is included in Acrobat .PDF form and can be viewed and printed (have 300 pages ready) using the Adobe Acrobat Reader (which must be installed separately from FelixCAD).

**Note:** If you purchased FelixCAD on-line, you should place all program documentation (.pdf files) into the \manuals sub-folder of the FelixCAD installation folder.

Chapter 1 covers basics on working with the program.

Chapter 2 offers a detailed description of drawing aids. There 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.

Chapter 3 explains the techniques used to view drawings, with a detailed explanation on the use of viewports, how to achieve good results from FelixCAD's ability 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, rendering, and outputting the drawings on a plotter or a printer are explained.

### **FELIXCAD FOR AUTOCAD USERS GUIDE**

To assist AutoCAD familiar operators and programmers with their use of FelixCAD, a separate *.pdf* document is included. The FelixCAD for AutoCAD Users and Programmers guide explains how the FelixCAD interface and command operations can be easily adapted to work more like AutoCAD.

The guide also contains details on the main differences between FelixCAD and AutoCAD with regards to use, customization, and programming techniques. You should place all documentation into the **\manuals** folder.

To view/print (have 70 pages ready) the FelixCAD for AutoCAD Users guide, use the Adobe Acrobat reader to open the appropriate *.pdf*, contained in the **\manuals** folder.

### **CUSTOMIZATION AND LISP PROGRAMMER'S GUIDE**

The *Customization and Programmer's Guide* manual appeals to users interested in further adjusting, individualizing, customizing and programming FelixCAD and its functions to specific application needs. To view the programmer's guide, use the Adobe Acrobat reader to open the appropriate *.pdf* file, contained in the **\manuals** folder.

Initial chapters contain a number of suggestions for possible arrangement of the user interface, working with alias commands, macros, and function key assignments, as well as a guide to the directory structure of the program.

Chapters are included to describe the meaning and use of template drawings, as well as the definition of linetypes and hatch patterns.

Middle chapters 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.

Detailed information is provided about the integrated FLISP interpreter allowing you to extend the program or comply it with special working requirements.

And finally, a detailed overview of the global and local variables in the system is outlined.

### ***ADVANCED PROGRAMMING USING C, C++, OR DELPHI***

The FelixCAD Developer's Toolkit Application Programming Interface (FDT-API) Manual contains instructions of how to extend FelixCAD through use of its advanced programming interface. The FDT API manual is supplied in *.pdf* format.

If you plan to create integrated Lisp, C, C++, or Delphi solutions using FelixCAD, contact us at [www.fcad.com](http://www.fcad.com) (North America) or [www.fcad.de](http://www.fcad.de) (outside North America) to obtain details on our Graphic Developer's Engine. GDE licensing allows low cost distribution of vertical market applications, with flexible low cost licensing designed to suit the needs of any size developer.

### **Contact Us**

We look forward to your successes with FelixCAD. Please don't hesitate to contact us for assistance at [info@fcad.com](mailto:info@fcad.com) (North America), or [info@fcad.de](mailto:info@fcad.de) (outside North America). Tell us your thoughts on the product and your experiences with its use, whether you are a user or a developer. Your input is welcome, and we look forward to hearing from you!

## **PROGRAM INSTALLATION AND SETUP**

FelixCAD is available for purchase on CD, or via download off the Internet.

FelixCAD includes an automated installation routine, which is run by using the appropriate Windows “run” function (under the Windows “Start” menu”) to execute the program Setup.Exe.

For information on where to purchase FelixCAD, please visit [www.fcad.com](http://www.fcad.com) or [www.fcad.de](http://www.fcad.de).

### **Installation on Windows 95/98, NT, 2000, XP and Me.**

1. If you purchased a CD version of FelixCAD, go to step 2. If you downloaded FelixCAD off the Internet, save the file(s) from your download into a temporary directory on your computer and proceed to step 3.
2. Insert the FelixCAD CD-ROM into the CD drive of your computer.
3. From the **Start** button, browse the CD-ROM (or download folder) to select the **Setup** program file. CD versions are set to “autorun”, so the installation may start up automatically upon insertion into your CD drive.
4. Follow the on-screen prompts to complete the installation.

FelixCAD offers three types of installation:

- **INSTALL FULLY LICENSED VERSION**  
Select this option, if you have purchased a fully licensed software version (including a valid CD-Key Number).
- **INSTALL EVALUATION VERSION**  
Choose this option, if you wish to evaluate the program for a 30-day period with full functionality. The program will cease operation after 30 days.
- **CONVERT EVALUATION TO FULL VERSION**  
If you purchased a CD Key Number, this option allows you to convert an already installed evaluation copy to a regular fully licensed version of the software.

### **Uninstall**

FelixCAD can be uninstalled from your computer. When using the Windows 95/98/NT4 desktop, the program can be removed using the **Add/Remove** function in the **Control Panel**. The **Uninstall** program removes the program groups and *.ini* files from the Windows directory. An Uninstall icon is also available under the FelixCAD program group.

*Be careful 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-ROM, run the **AcroInst.Exe** program situated in the \acrobat directory of the CD. Follow the on-screen prompts to complete the installation.

If you purchased or downloaded the Internet version of FelixCAD, visit [www.fcad.com](http://www.fcad.com) or [www.adobe.com](http://www.adobe.com) to download a free copy of the reader. You will need the reader to view or print the supplied documentation.

## **Technical Support**

Please contact your selling dealer, or send an email to [support@fcad.com](mailto:support@fcad.com) (North America) or [support@fcad.de](mailto:support@fcad.de) (outside North America).

From our Web site(s), choose the Support page for hints and tips, and bonus files. You can also access the FelixCAD home page for technical requests or reports.

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## WHAT'S NEW IN FELIXCAD 5

### New Entities:

#### Lightweight Polyline

Now FelixCAD supports the memory-optimized LWPOLYLINE-Entity. The creation of lightweight or "previous style" polylines can be controlled with the global system variable PLINETYPE.

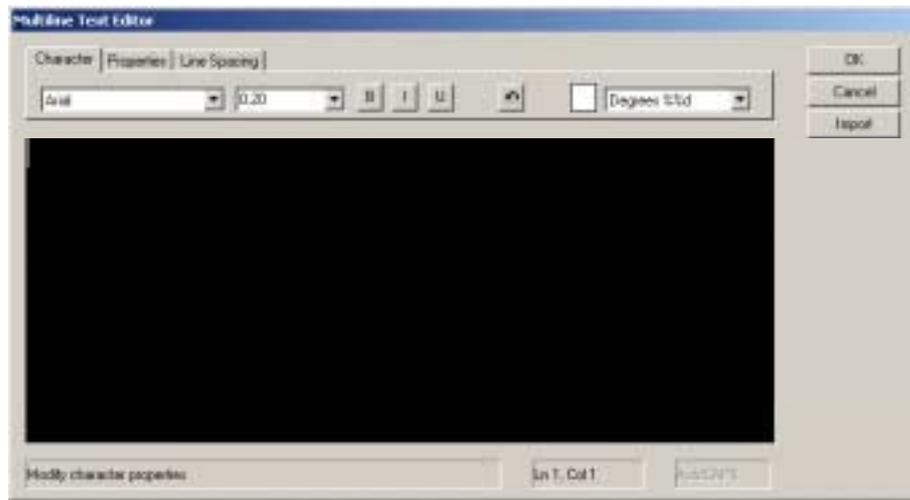
This variable specifies whether FelixCAD uses optimized 2D polylines. PLINETYPE controls both the creation of new polylines with the PLINE command and the conversion of existing polylines in drawings from previous releases.

- |   |   |
|---|---|
| 0 | Polylines in older drawings are not converted when opened; PLINE creates old-format polylines |
| 1 | Polylines in older drawings are not converted when opened; PLINE creates optimized polylines  |
| 2 | Polylines in older drawings are converted when opened; PLINE creates optimized polylines      |

### New Commands:

#### Command MTEXT

FelixCAD 5 now fully supports MTEXT. After issuing the command and specifying the location of the new text, you are presented with a dialog in which you enter your text. Three tabs allow you to exercise control over all text parameters and an Import function enables you to import an external text file into the window.



MTEXT Dialog showing Character options



MTEXT Dialog showing Property options



MTEXT Dialog showing Line Spacing options

### Pre-Selection of Objects

The global PICKFIRST system variable controls whether you select objects before (noun-verb selection) or after you issue a command.

- 0 Turns off PICKFIRST; you select objects after you issue a command
- 1 Turns on PICKFIRST; you select objects before you issue a command

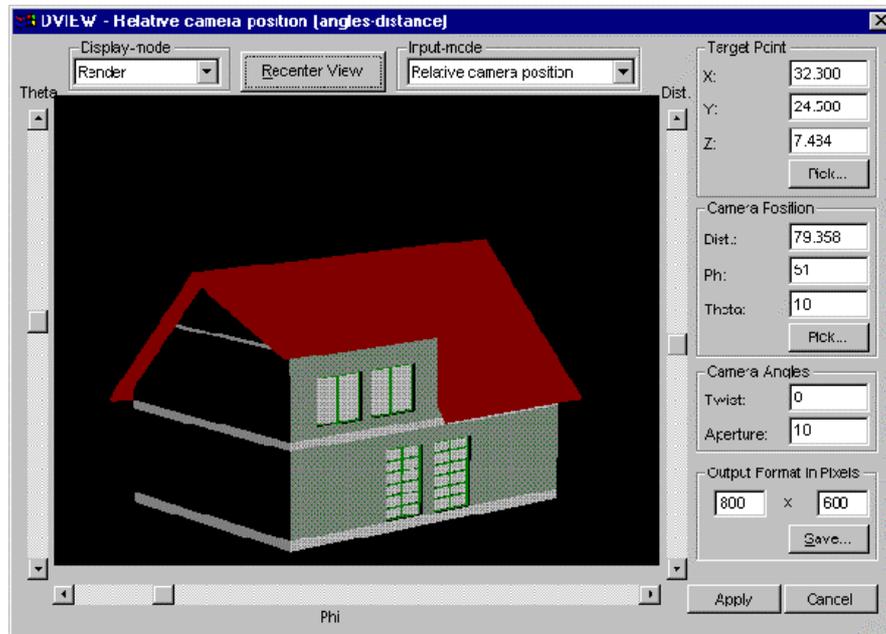
## Object Snap

Snap points of hatch pattern lines will be ignored during the point selection.

## Command FLXRECOVER

This new feature enables the user to recover damaged flx-drawing files of the FelixCAD 5 or 4.02 file format.

## Command DVIEW \*



The DVIEW command creates a perspective view in different display modes. Supported display modes are rendered, hidden line and wire frame.

Different input modes make it easy to adjust the viewpoint of the drawing. The target point and the camera point can be selected in the drawing. The result can be saved in a bitmap file with a predefined resolution.

The center view button brings the drawing into a base position at the center of the DVIEW output window. If the dialog is closed by the apply button, the current view direction of DVIEW will be set to the drawing.

## Command SAVEAS401 and SAVEAS402 \*

The command SAVEAS401 saves a drawing in the previous FelixCAD 4.01 file format. SAVEAS402 saves in the new recoverable FelixCAD 4.02 file format.

## Command DRAWORDER

DRAWORDER changes the drawing and plotting order of any object in the FelixCAD drawing database. In addition to moving objects to the "front" or "back" of the sort order, you can order objects relative to another object (that is, above or below a selected object). DRAWORDER provides the following options:

Above	Under	Front	Back
-------	-------	-------	------

- **Above** - Moves the selected object above a specified reference object.
- **Under** - Moves the selected object under a specified reference object.
- **Front** - Moves the selected object to the top of the drawing order. The selected object becomes the last drawn entity in the drawing database.
- **Back** - Moves the selected object to the bottom of the drawing order. The selected object becomes the first entity in the drawing database.

## Differences between FelixCAD and AutoCAD

Different object sort methods are not supported in FelixCAD. The object order is changed in the database directly, instead, using DRAWORDER. A SORTENTS system variable for controlling the object sort mode is therefore not required.

## Recording of the User Input

For support purposes we have integrated in FelixCAD, a new user-input-recording function. FelixCAD writes all text window contents into a specified file. The command can be activated with the command LOGFILEON. The LOGFILEOFF command stops the prompt echoing to a file. An output file name and path can be set via the global system variable LOGFILENAME. The global system variable LOGFILEMODE controls whether or not a logfile can

be written. (Please see also the system variable chapter in the developer section)

## Command and Function Enhancements:

### Purge Command

Enhanced performance of the purge command. An internal reference counter makes the purging of unreferenced FelixCAD table entries faster.

### Transparent Alias Commands

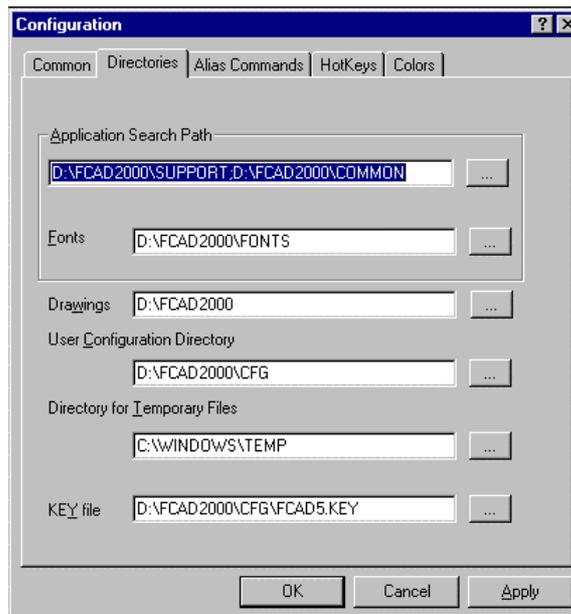
Alias commands can be used while a command is active. See the example of the transparent alias command 'Z below:

Definition in the FCAD5.key file:

Z=ZOOM

```
> LINE
From point:
To point:
To point: 'z
Zoom scale factor: Window
First corner:
Second corner:
To point:
```

## CONFIG Command



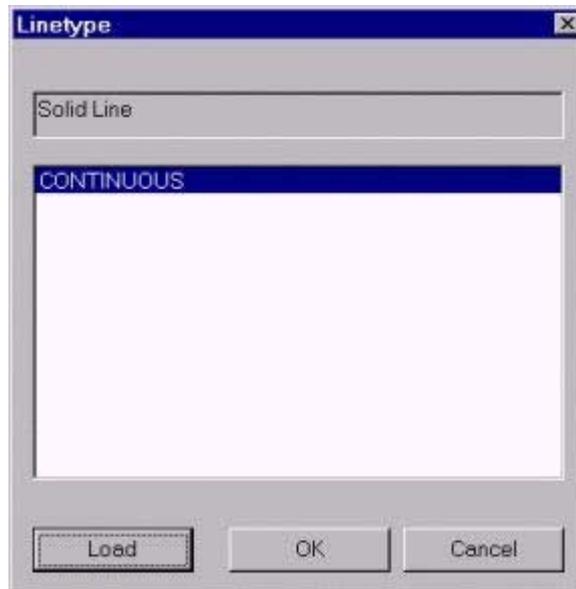
In the register *Directories* of the **CONFIG** command, browse-buttons now extend the ability to adjust FelixCAD path settings. The font path now supports multiple path selection, which can be useful if you have multiple text font directories. If two font directories contain the same font definition, the first-found font will be used.

## Autosave

Autosave in FelixCAD essentially now operates in the same manner as in AutoCAD

**Command Extensions:****Layer Command:**

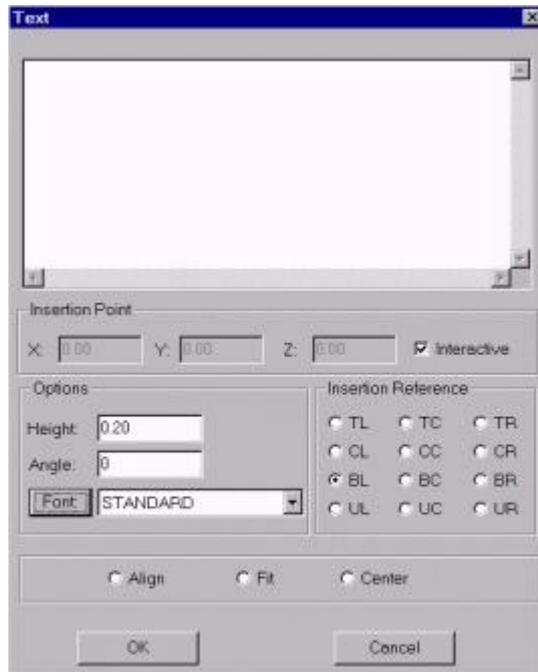
New line type definitions can be loaded without leaving the layer command. In the sub-dialog box you can select the **Load** button to accomplish this:



In the layer manager the default button has been changed to the **New** button. If you type in a layer name and press the return key, a new layer will be created while you remain in the layer manager.

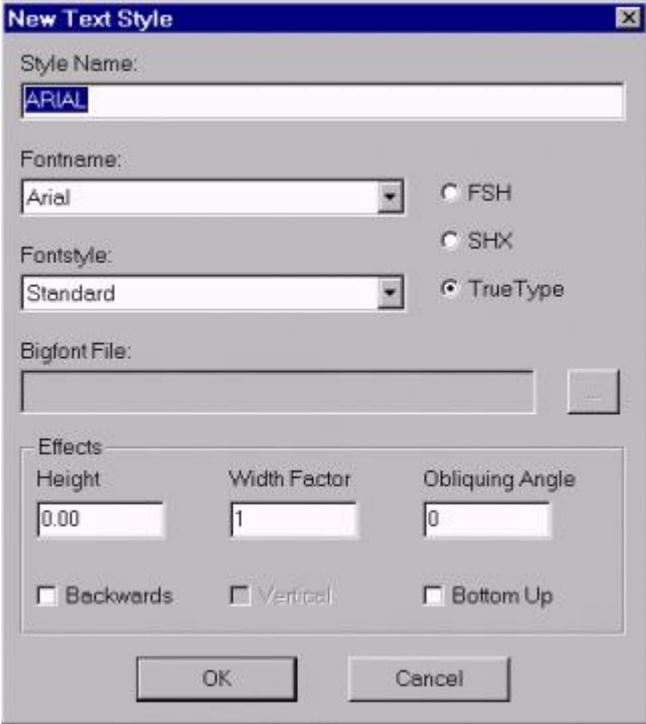
**Text and ATTDEF Command:**

New text styles can be directly created while executing the text and ATTDEF command. The FONT command can also be started, by clicking on the font button beneath the current font list box.



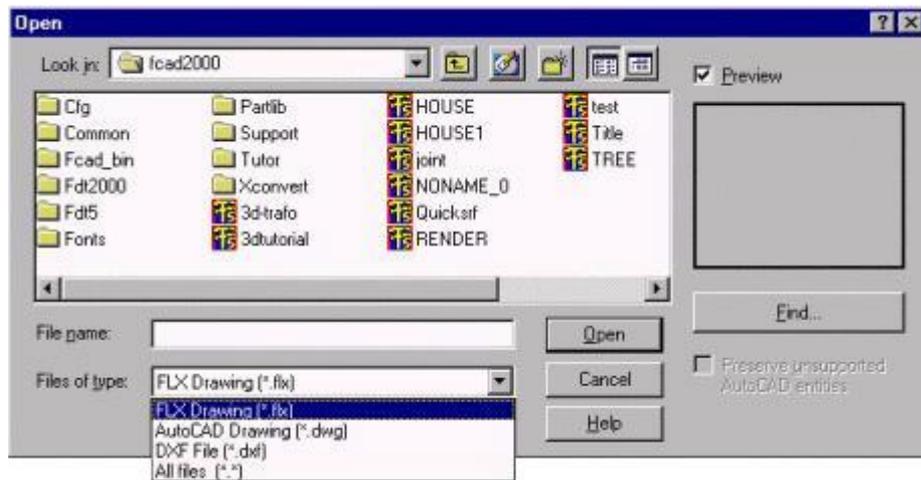
**Command Font:**

During font creation, a default text style name is displayed that is dependent upon the selected font name. If you make changes to the default name in the text style edit box, a new default name will no longer be suggested and your changes will remain in effect.



## Command Open

The file type box has been extended by the option "All files (\*.\*)".



## Open Command And The Import of MTEXT-Entities

On opening an AutoCAD format DWG or DXF file, "MTEXT" entities are converted directly to text entities.

## Command Save

The save command creates a preview bitmap for AutoCAD R14 and 2000 DWG files.

## Command Hatch

A new solid hatch pattern is integrated into FelixCAD 5.

## Commands 3DVIEW and SETVIEWDIR (VIEW TWIST)

Now the commands 3DVIEW and SETVIEWDIR support the rotation of the view in the XY-Plane. This is especially useful for quickly re-orienting the drawing within the drawing surround prior to plotting.

## Roundtrip DWG/DXF 2000

If you open an AutoCAD drawing, you now you have the possibility to "Roundtrip" all unsupported AutoCAD 2000 entities. That means, that while being edited in FelixCAD, unsupported AutoCAD 2000 entities remain unchanged and invisible the background. If the drawing is saved in AutoCAD 2000 drawing format again, all entities including these unsupported AutoCAD 2000 entities will be written back to the file.

### "Roundtripped" AutoCAD Entities:

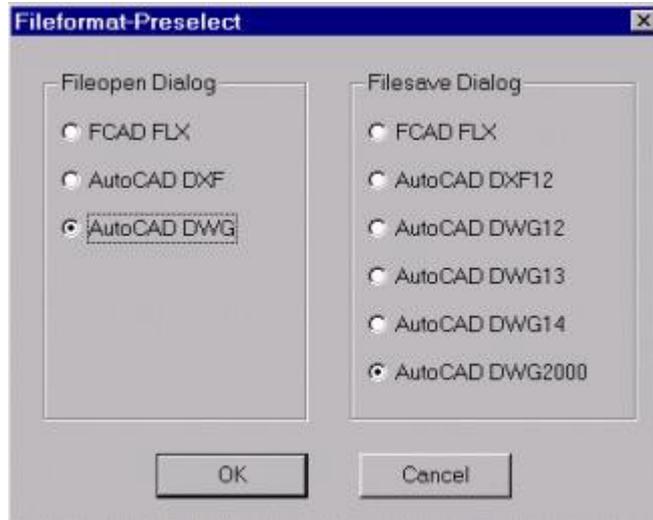
ARCALIGNEDTEXT	
BODY	
IMAGE	
LEADER	
MLINE	
OLE2FRAME	
PROXY	
RAY	
REGION	
RTEXT	
SOLID3D	
TOLERANCE	
WIPEOUT	
XLINE	
XRECORD-OBJECTS	

### "Roundtripped" AutoCAD Table Information:

DICTIONARYWDFLTOBJTYPE	
DICTVAR	
IMAGEDEF	
MLINESTYLE	
OBJECTPOINTER	
PLACEHOLDEROBJECT	
VBAPROJECTOBJECT	
WIPEOUTVAROBJECT	

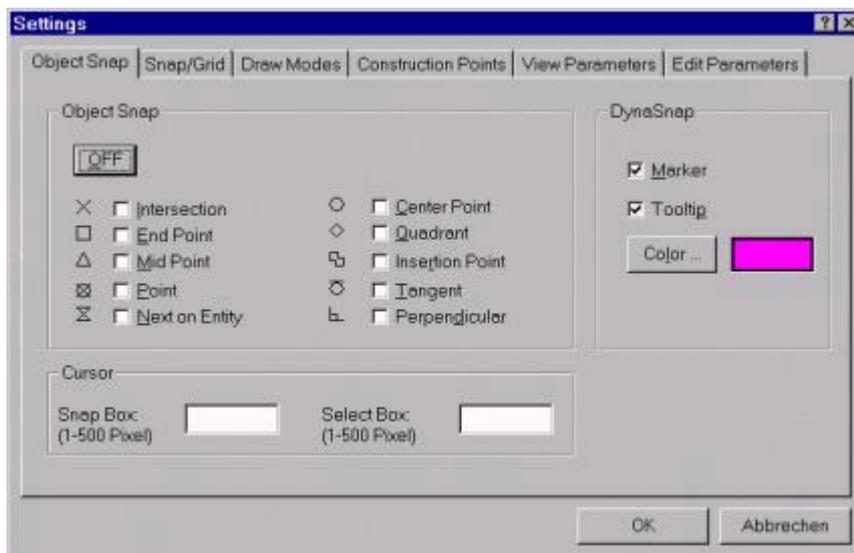
## Default File Format

The default file format for the open and save dialog can be predefined in the CONFIG command register *Common*. If you select the button *Fileformat*, you will get the dialog displayed below. In this dialog, you can select your default file format to work within FelixCAD.



## DynaSnap

DynaSnap stands for dynamic snap functionality. Now, FelixCAD has the possibility to identify the currently active snap point dynamically, by using distinctive symbols (markers). You can choose the desired snap point or switch between different modes by the tab-key. DynaSnap mode can be turned on in the register **"Object Snap"** of the SETTINGS dialog or by switching the marker and/or the tool-tip button on. Different colors of the marker can be also selected. The displayed markers or symbols are the same as shown in the following dialog to the left of each respective object snap mode.



**Object Snap Modes:**

Turns on running object snap:

Marker	Mode	Supported Entities
	Intersection	Snaps to the intersection of an arc, line or, polyline segment.
	End Point	Snaps to the closest endpoint of an arc, line or polyline segment, solid, or 3dface.
	Mid Point	Snaps to the midpoint of an arc, line or polyline segment, solid, or 3dface.
	Point	Snaps to a point object.
	Next on Entity	Snaps to the nearest point on an arc, circle, line, point, or polyline segment.
	Center Point	Snaps to the center of an arc, circle, or polyline arc segment.
	Quadrant	Snaps to the quadrant point of an arc, circle, or polyline arc segment.
	Insertion Point	Snaps to the insertion point of an attribute definitions, part/block, or text.
	Tangent	Snaps to the tangent of an arc, circle, or polyline arc segment.
	Perpendicular	Snaps to a point perpendicular to an arc, circle, polyline segment, line, solid, or 3dface.

## **New Temporary Object Snap Mode NONE**

In FelixCAD 5, all current selected object snap modes can be switched off for a single selection by selecting the **NONE** option

## **Menu Echo Toggle ^P**

With the option ^P the user can toggle the menu echo on and off in one pull down menu command line.

For example:

```
MENUITEM, (#|ACTDB|<0), "&Line", {Draw line(s)|LINE},  
[FCMD]^P^^LINE
```

## WHAT'S NEW IN FELIXCAD 4.01

A number of substantial program enhancements have been made to FelixCAD since version 4.0 was released in Spring of 1999. Please search the FelixCAD installation folder for "What's new" files containing information on updates and changes.

For the latest information on FelixCAD product improvements and changes, please visit our web sites at [www.fcad.com](http://www.fcad.com) and [www.fcad.de](http://www.fcad.de). FelixCAD 4.01 highlighted additions and enhancements include:

- Support for True Type fonts, and AutoCAD .SHX fonts (see **FONTS**) with font mapping.
- Improved DWG®/DXF® file exchange capabilities.
- Raster image import and handling has been improved and extended to support color BMP and TIF images, in addition to the existing monochrome RLC file formats (see **RASTER**).
- Improvements have been made in display and object selection speed. The **UNDO** command has been sped up and improved.
- Dramatic enhancements have been made to the printing/plotting function, with file sizes and printing times reduced by up to 90%.
- A new batch processing (see **BATPROC command**) function has been added, allowing unattended printing or conversion of multiple files in a single operation.
- A new Mouse Context toggle has been added (**Tools>Mouse Context On/Off**). When switched on, simply left mouse click on any drawing entity for a complete listing of available options.
- New Dynamic zooming and panning functions (see **DZOOM and DPAN**).
- Improvements to the FCAD Drawing Explorer™ (see **XPLORER command**).
- A new graphic Image library has been added (Image Library icon in the FelixCAD program group) for simple management of bitmaps and slides. The ability to create slide .SLD files has been added.

- Dimensioning has been improved and a new DIMLIN command is available.
- New **MEASURE** and **DIVIDE** commands have been added allowing 2D objects to be divided into a set number of segments, or new points created at user defined distances.
- New 3D commands have been added (**Draw>3D**), allowing for quick and easy creation of 3D boxes, wedges, cylinders/cones, spheres, ellipsoids, and toruses. See the corresponding command.
- Memory management for polylines has been completely overhauled, making polyline processing more efficient.
- The FelixCAD Lisp, C/C++, and Delphi programming API has been extended with new functions and calls. See the appropriate readme file and/or programming manual for details.
- A new AutoPort™ tools library (Click on **Tools icon in the FelixCAD program group**) has been added to automate porting of existing AutoLisp® and ADS® programs written for use with AutoCAD. The library also includes tools to convert menus and dialogs.



# Getting Started: The Basics of Drawing with FelixCAD

This chapter provides you with an introduction to FelixCAD. You learn about some of the most frequently used commands and tools.

This chapter begins with a description of how to start the program. This is followed by how to create a new drawing and open existing drawing files. You also learn how to save the result of your work and how to exit the program correctly.

Later sections of this chapter describe the user interface, its elements, and components. You'll find instructions to control the interface and the customized arrangement of its elements.

To control FelixCAD, you may use the mouse or other pointing device (such as a digitizing tablet) to select commands the menu bar, tool areas, or palettes. Commands can also be executed using the keyboard. For defining points, distances, and angles, the keyboard provides you with the advantage of greater precision. Ideally, using a combination of the mouse and keyboard optimizes your productivity.

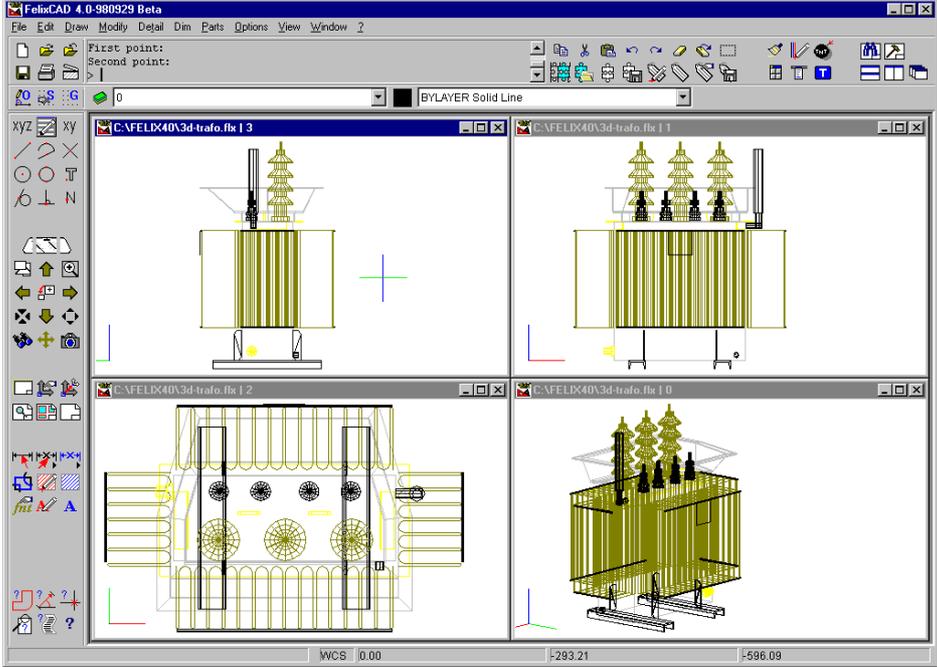
We recommend that the novice CAD user read the parts of this chapter containing explanations of command selection, coordinate input, and command options -- whether by keyboard, mouse, or digitizer. FelixCAD provides many alternatives for data and command input and selection. The variant that you choose depends on your personal style.

## General Remarks

FelixCAD runs on a computer with Microsoft Windows 95, 98, and NT. All aspects of the program function according to Windows standards. This is true for pull-down menus, toolboxes, dialog boxes, edit boxes, scroll bars, etc. There are buttons for opening new and existing drawing files, to save and print drawings, as well as for changing window arrangements. It is recommended that you load FelixCAD on a PC with the most recent operating system and 'patches' available. It is also good practice to have the latest device driver files, such as for Video Cards and Printers, on your system.

To work with FelixCAD, we assume that you already know how to use the Windows operating system. For this reason, we do not explain how to use the user interface elements. If you are not sure, please consult your Windows *User's Guide* manual, the on-line help system, and other source of information on the subject of Windows.

In this manual, the term *drawing* is used as a synonym for *file*. When we talk about “opening or closing a drawing,” it’s the same thing as opening or closing the file.



Typical FelixCAD Screen (shown with 4 views open)

## STARTING FELIXCAD

During the installation of FelixCAD, the Windows Setup program places a FelixCAD icon on the desktop. (A detailed description of the installation is contained in the “Introduction” of this manual.)



Start FelixCAD by double-clicking the icon. As an alternative, click the **Start** button on the taskbar, then select **Programs > FelixCAD > FelixCAD**.

To Change the FelixCAD icon properties, right mouse click on the icon and scroll down to 'properties'. Under the 'Shortcut' tab of the icon properties dialog box, the following options can be changed:

- (1) 'Start in' - where FelixCAD first looks for and saves files,
- (2) 'Shortcut key' - allows assigning a Shortcut Key or Keys to launch the program
- (3) 'Run' - setting the Window size for the program to Run: minimized, normal or maximized.



Look in your Windows documentation for details on changing an icon's properties.

**OPENING AND SAVING DRAWINGS**

In this section, you learn about the commands for opening a drawing, for saving your work, and for exiting FelixCAD.



**The File Pull-down menu**

File Operations

Printing and Plotting setup and execution

Batch Processing. Allows printing/plotting or conversion of multiple files at one time

Paper Layout Mode (Paper space/Model Space)

System configuration, desktop management, and resource manager



Last opened files

By the end of this section, you should understand these commands:

Command Name	Meaning
New	Starts a new drawing.
Open	Opens an existing
Close	Closes the current
Save	Saves the current drawing.
Save As * (only FLX)	Saves the current drawing

Save All	Saves all open drawings.
File Manager	Visual File Management
Delete Locked Files	Erases locked files.
Exit	Exits the FelixCAD

These commands are located in the pull-down menu labeled **File**.

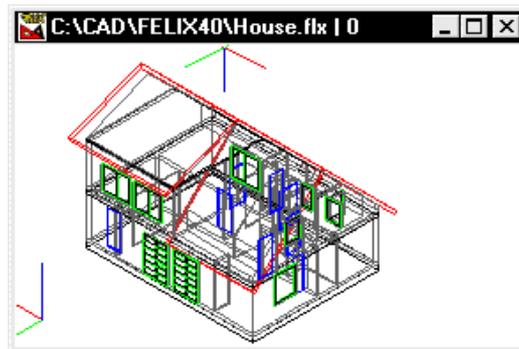


There are also icons in the toolbar for the most used commands. The commands for printing and plotting your drawing, which are also found in the **File** menu, are explained in the Printing and Plotting chapter of this manual.

FelixCAD 4.01 introduces batch processing of multiple files. Files can be printed/plotted, or converted from one format into another using batch processing which is described in a separate chapter of this manual (see BATPROC command).

By default, FelixCAD saves drawing files with the file extension *.flx*. Each file contains the drawing and, if necessary, up to four independent views of the drawing. Files can optionally be opened or saved in *.DWG* or *.DXF* format.

Each drawing, as well as each view of a drawing, is displayed in a separate window. FelixCAD can open up to four drawings, each with four independent views – for a maximum of 16 windows open at a time. All windows carry the filename on the title bar. In addition, a number between 0 and 3 indicates the view. See figure below.

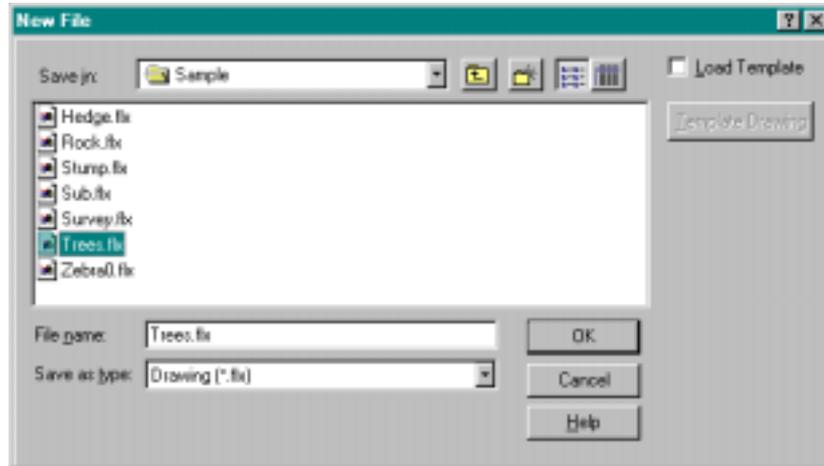


To avoid accidental loss of data, we strongly recommend that you periodically save your drawings as you work on them. For this purpose, use the **AutoSave** function. Here's how: Select **File > Configure**, then click the **Common** tab. In

the **Safety** section, click the **Automatic Save** check box, then choose the interval in minutes. Do not use a value shorter than 10 minutes, unless you have a very fast computer or a very small drawing.

## New Drawing

Before you can start drawing for the first time, a new file must be opened. The **New** command (select **File** from the menu bar, then select the **New** item) displays the following dialog box:



The **New File** dialog box

By default, FelixCAD names the new drawing **Noname\_0.Flx**. As an alternative, you can type any filename up to 255 characters long in the **File name** field of the **New File** dialog box. In addition, you can choose the drive and folder (via the **Save in** list box) to store the file in.

If a drawing file of the same name already exists, the program warns you and requires a confirmation to replace the existing file.

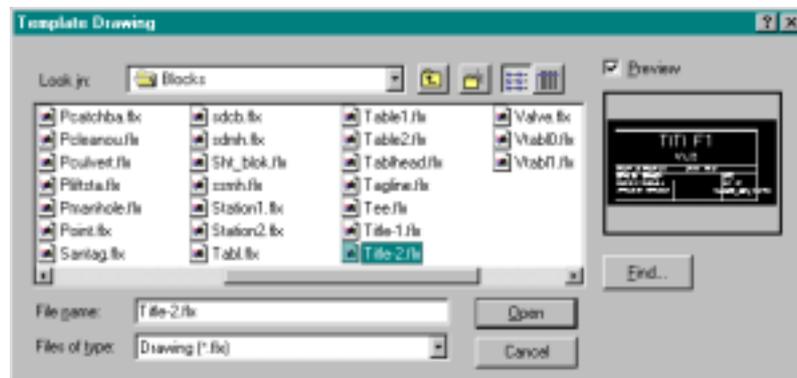
Click **OK**. Notice that FelixCAD creates a new, blank drawing window.

## Load Template

A *template drawing* (called a prototype drawing in AutoCAD) may be used when starting a new drawing. A template drawing is a model, which can be used as basis for a new drawing. Templates are typically used to the establishment of a standardized drawing environment, such as:

- Drawing border and title block.
- Dimension settings.
- Pre-defined layers.
- Preset views.
- Standardized linetypes.
- Settings of system variables.

To start a new drawing based on a template file, click the **Load Template** check box, then click **Template Drawing** button. FelixCAD displays the **Template Drawing** dialog box. Select the drawing to be used as a template drawing and click **OK**.



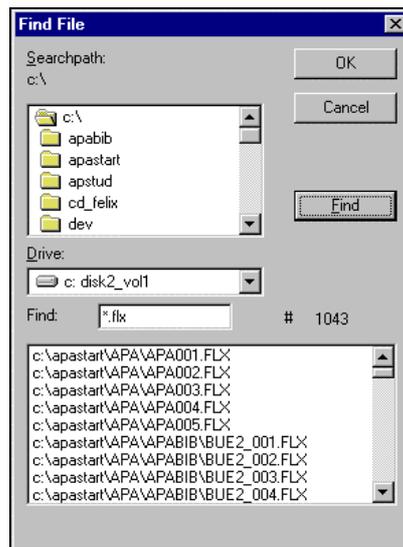
The **Template Drawing** dialog box

## Find

If you do not know the exact storage location of a drawing file, click the **Find** button. In the **Find File** dialog box, select the drive to search. Enter as much of the file name you know. You may use the “wildcard” characters \* (for any group of characters) and ? (for any single character).

For example, entering **Floor\*** finds **Floor**, **Flooring**, and **Floors**. On the other hand, **Floor?** finds only **Floor** and **Floors**.

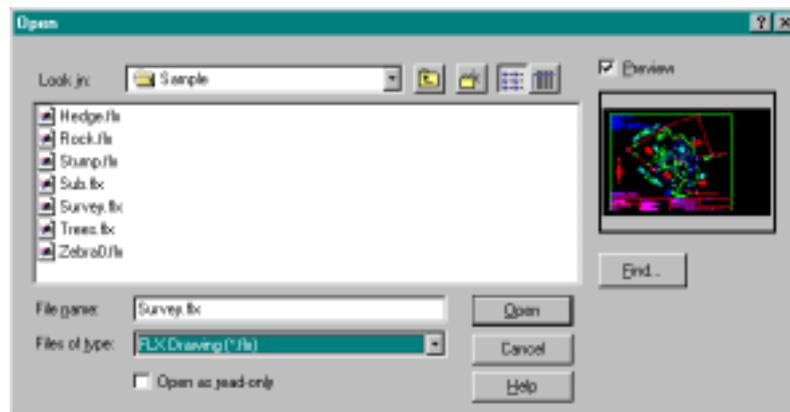
Click **Find** and FelixCAD searches for the corresponding file names in the indicated drives.



The Find File dialog box

## Open an Existing Drawing

The **Open** command (**File > Open**) is used to open an existing drawing file. After entering the command, you see the **Open File** dialog box.



The **Open** dialog box



You may select a drive, folder (or sub-directory), file name, and file type. By clicking on "Files of

Type", FelixCAD can open *.flx* files, *.dwg* files saved by AutoCAD version 2.5 through Release 14, or *.dxf* files created by other CAD programs.

Click **Open** to load the drawing.



The open drawing progress bar

While opening a drawing, a blue progress bar appears at the bottom of the program window (the status line). It shows the progress of loading the file. The bar moves from left to right and disappears when the file is completely loaded.

### Open as Read-only

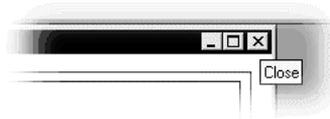
You can specify that the file be opened as read-only. Click the **Open as read-only** check box in the **Open File** dialog box. The read-only mode opens the file but does not allow you to save it, except by a different filename.

This is a protective mechanism against inadvertently modifying or changing the file. After opening in read-only mode, you can modify the drawing in any way, but the modifications are not saved.

When you try to save a read-only file, FelixCAD displays a warning and asks if the file should be saved under different name. If you want the changes saved, then select the **Yes** button, and type a filename in the **Save As** dialog box.

### Closing a Drawing

When you have finished working on a drawing, you *close* the drawing. Closing the drawing saves it and removes the drawing from the program. FelixCAD itself remains open until you exit the program (more later). To close the drawing, type the **Close** command, or select **File > Close** from the menu bar, or press the shortcut keys **Ctrl+F4**. As an alternative, you can close the drawing (as in all Windows programs) by clicking the small **x** at the far right end of the title bar.



The **Close** window button

If a drawing has been changed before closing, a prompt asks you if you want the file saved -- this prevents the unintended loss of work. Drawings may be saved in *.flx*, *.dwg* (R12, R13, or R14), and *.dxf* formats.

## Save the Drawing

The **Save** command (**File > Save**) saves the drawing in the active window to be saved. (The *active* window is the one with the dark title bar; *inactive* windows have a light or gray title bar color.)

The file is saved by its name in FelixCAD's own *.flx* format. The file is saved in the same drive and folder as at the time of opening the file.

Actually, before saving the drawing, FelixCAD renames the file with the extension of *.bak* as a backup file.

## Save All Drawings

The **SaveAll** command (**File > Save All**) saves all open drawings. This is quicker than using the Save command four times, once for each open drawing.

## Save As \* (only FLX)

The **SaveAs** command (**File > Save As**) also saves the current drawing, like the **Save** command, but it allows you to change the name of the drawing.

The **Save As** dialog box allows you make these changes:

- Save by a different file name.
- Save in another file format.
- Save on another drive.
- Save in another folder (sub-directory).

Drawings may be saved in *.flx*, *.dwg* (R12, R13, or R14), and *.dxf* formats.

*Note: to convert multiple drawings from one format to another (e.g. convert five DXF files to BMP, or eighty DWG "blocks" to FLX), refer to the Batch Processing (BATPROC) section of this manual.*

## File Manager

The Fileman command (**File>File Manager**) visually displays all *.FLX* drawings in the current directory.

## Deleting Files

The **DelFlk** command (**File > Delete File**) deletes files, specifically *lock* files. Lock files are security files 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*. The lock files prevent two or more users working on the same drawing at the same time in a networked environment.

Normally, lock files are deleted automatically, if the drawing files are closed in the normal manner. If the program is unexpectedly interrupted and the files have not been closed before the interruption, the *.flk* do not get erased. When you try to reopen the drawing after the interruption you get the message, “File locked.”

To gain access to the drawing, the corresponding lock file must be deleted. Delete the offending lock file and the drawing file can again be opened.

## Exiting the Program

It is important that you end every working session correctly. This is done by using the **Exit** command (**File > Exit**) or pressing the ALT+F4 shortcut keys.

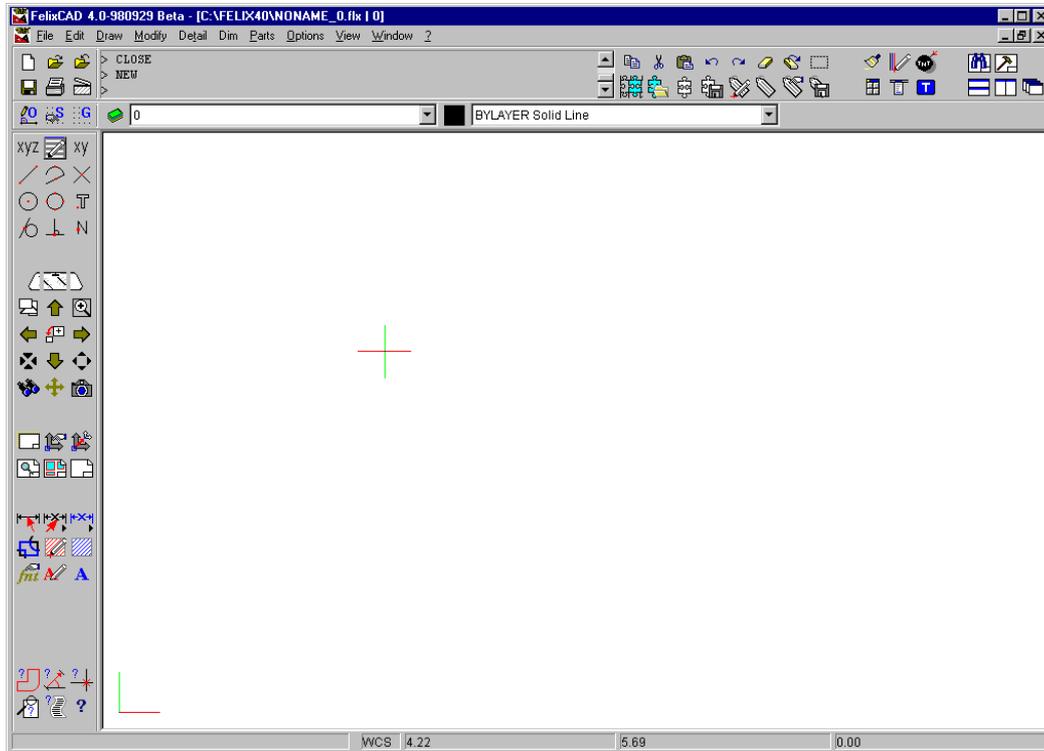
If there are drawings still open that have not been saved, the program prompts you to save the files first. If you confirm, the **Save File As** dialog appears.

After saving the drawings, the working session with FelixCAD is finished.

## THE PROGRAM'S USER INTERFACE

Upon starting FelixCAD, you see the *desktop*. It contains a series of standard elements, including the menu bar, the function bar, the vertical symbol bar, and the status bar. In addition, the desktop contains several palettes (toolbars) and a cursor menu, when applicable.

The following picture shows the standard look of the desktop.



The standard FelixCAD desktop

Usually, the desktop layout appears as shown in the figure above. However, you can customize the menus, toolbars, and other components of the program's desktop to suit your needs. In particular, you may wish to experiment turning various desktop components ON or OFF to achieve the best balance of quick function access and maximum drawing area.

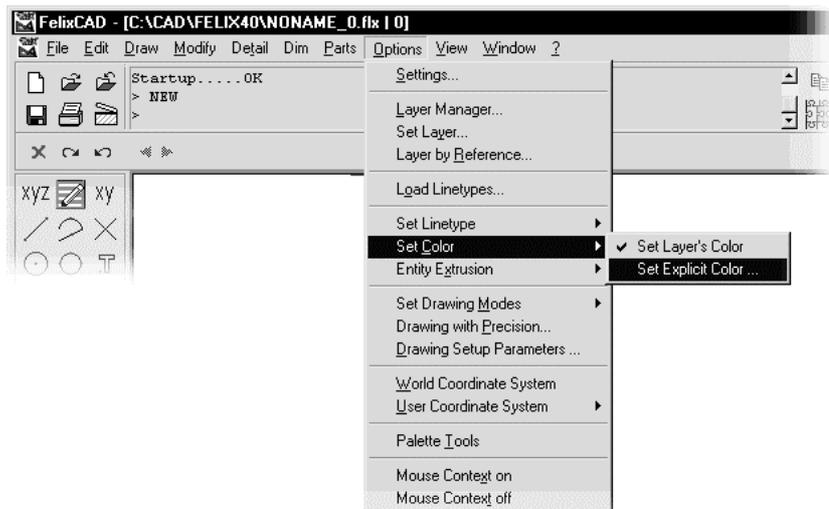
**Note for AutoCAD Users:** Refer to the "FelixCAD for AutoCAD" companion guide for instructions on how to convert the default FelixCAD desktop to an AutoCAD emulated desktop.

Alteration of the arrangement of the desktop layout can be achieved easily. In addition, you can replace the buttons of the toolbars of the program's desktop on the fly. Below in this chapter you find the corresponding information on both topics.

Advanced methods of desktop layout modifications are covered in the *Programming and Customization Guide*.

## Menu Bar and Pull-Down Menus

The pull-down menus are arranged on the menu bar. Each pull-down menu contains groups of thematically-related commands: **File**, **Edit**, **Draw**, **Modify**, **Detail**, **Dim** (dimensions), **Parts** (blocks), **Options**, **View**, **Window**, and **?** (help).



Menu items can have these states:

- Grayed-out commands are not available and cannot be selected. The command **Save**, for example, can not be executed if no drawing has been created or opened.
- Three dots (...) behind the command, such as **Settings...** in the figure above, means that a dialog box will be displayed.
- An arrow (➤) behind a menu item, such as **Set Color ➤** in the figure above, indicates that a submenu will be opened.

**Note:** the contents of the menu bar can vary, depending on the menu loaded.

Different menu files may be loaded, which allows you to adapt the menu bar to your specific needs.

Menu files are identified by the extension *.mnu*. The program contains a standard menu file located in the sub-directory *\applic*.

To load another menu file, such as **Fcad14e.Mnu** (which provides a menu that looks similar to AutoCAD Release 14's menu), select **File > Resource Manager > Pulldown Menu** (or type the **Menu** command). Refer to the FelixCAD for AutoCAD guide for further details.



The standard FelixCAD pull-down menu

For further information on menu and palette management, refer to the “Customization: Palette and Resource Manager” chapter of this manual

FelixCAD offers you the opportunity to graphically create or edit custom menus, dialogs, and palettes. These files can be created and edited with the help of the Dialog and Menu Editor (see complete explanation in the *Programming and Customization Guide*).

## The Toolbar

When you install FelixCAD, below the menu bar are two rows of buttons, on the left and on the right (with the command line area between them). This component of the desktop layout is called the *toolbar*.



The File icons and command line

The group of six buttons, located on the left side of the toolbar, executes commands from the **File** menu. Each button consists of a symbol (or icon), that represents a command. Simply click on a button to execute the command. This is faster than selecting the **File** menu, and then the menu item.

The symbols are arranged by subject, in groups. You find detailed explanations of each command, its icon, and their use in this and the following chapters of the manual.

The buttons shown in the figure below are called the System buttons. They perform system commands. From left to right, these are: **New**, **Open**, **Close**, **Save**, **Print**, and **Settings**. Just as all FelixCAD menus, the Toolbar conforms to Windows guidelines.



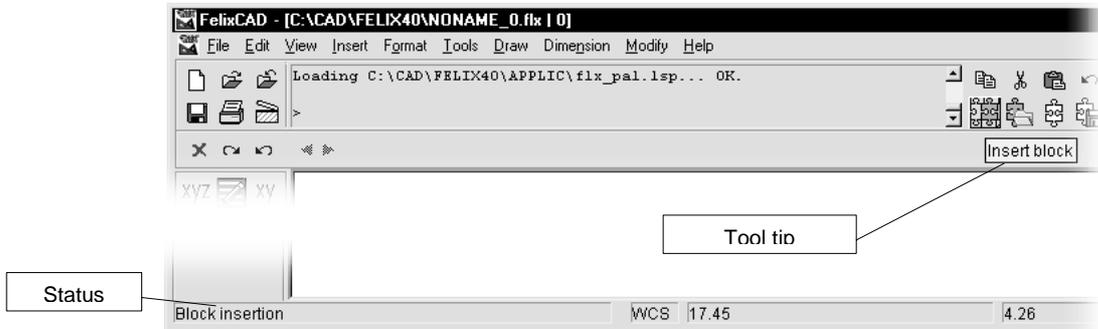
System icons for file operations

The other group of symbols, also arranged in two lines, contains additional commands that are commonly used in FelixCAD.



Additional buttons for commonly used operations.

To find the meaning of a symbol, pause the cursor over a button. FelixCAD displays a brief, helpful explanation of the symbol in two places: (1) in a yellow ToolTip that floats over the button; and (2) on the status line at the bottom of the screen.



The status line (bottom left) and ToolTip (upper right) display the meaning of symbols.

For example, the “puzzle piece” symbol (first icon in the second row) is the **Insert Block** command.

### Command Line and Text Window

The command line and the text window are the primary communications link between you and the program. In its default condition, FelixCAD places the command line directly below the menu bar. (In the figure above, the command line is the rectangular box containing the words “Loading C:\CAD...”.)

The command line is the place where:

- You type commands and option names, if you prefer typing to mouse clicking.
- You specify coordinates and values, select objects, and confirm inputs.
- FelixCAD displays the selection or input steps for the user (for example, *4 selected*).
- FelixCAD displays messages, indications, and warnings (for example, *Command not found*).

All commands you select, whether by menu, palette, or at the command line, are displayed at the command line. This lets you verify commands and options before they are confirmed. At the same time, the command line (and the text window) provides a history of your design session. You can read back and reconstruct program steps of the working session.

The command line is not used for everything. When a command has many choices, FelixCAD displays a dialog box instead. Dialog boxes are used for:

- Displaying and setting system variables.
- Listing geometric information of selected entities.
- Displaying information regarding layers, loaded linetypes, text styles, defined views, and user coordinate systems.

The command line is not fixed in its location. Use the **File > Desktop Manager > Command Line** command to move the command line to the bottom of the screen or to turn it off (which provides more screen real estate).

The text screen is a larger version of the command line. To see the text screen, click the  button in the toolbar or type the **TScreen** command. The text window can be arranged anywhere on your computer screen; the size of the screen can be adapted to your needs. As an alternative, the **TScreenFull** command shows the text window covering the entire display.

In both the command line and the text window, with the help of the scroll bar, or with the **PgUp** or **PgDn** keys, the text can be scrolled up and down.

**Note:** In the following chapters, the terms “command line,” “command line area,” and “text window” are synonyms. If there is any difference in the function of these terms, the correct term is used and the difference is spelled out.

## The Options Bar & Menu

Most commands in FelixCAD have two or more options. These options determine how the command progresses. For example, after you start the **Line** command (for drawing line segments), the following option menu appears below the command line:



The **options bar** displays the command's current options and sits in a fixed location

The left end of the option bar contains three permanent symbols. From left to right, they are:

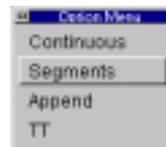
- **Cancel** command (same as pressing the **Esc** key).
- **Redo** last undone command (same selecting **Edit | Redo**).
- **Undo** the last command (same as selecting **Edit | Undo**).

The two arrowheads scroll the list of options, if the screen is not wide enough to list all options.

To use a command option, you can either (1) select the option name from the options menu; or (2) type the capitalized letters at the command line. In the example above, to draw a continuous set of line segments, you can click the Continuous work on the option bar, or you can type the letter C at the command line:

```
> line
From point: C
From point:
```

As an alternative to the options bar, you can have the options appear in a floating menu. Select **File > Desktop Manager > Options > Menu** from the menu bar:



The **options menu** displays the command's current options and floats on the screen

**Note:** The wording of the option menu and bar change with each step of a command.

## The Property Bar

The property bar gives you a quick and comfortable way of setting some main properties for further construction and drawing. To turn on the property bar, select **File > Desktop Manager > Property** from the menu bar.



The property bar gives you fast access to the most important properties

The first three buttons control:

- **O**: Orthogonal mode.
- **S**: Snap mode.
- **G**: Grid display.

Click the appropriate button to *toggle* the mode. Toggle means to turn off and on, like a light switch. Explanation of these drawing tools is found in Chapter 2 “Drawing with Precision.”

The remainder of the property bar set entity properties of layer, color, and linetype:

- **Layer button**: opens the **Layer Manager** dialog box, which is where you create new layers, specify their properties, rename them, and so on.
- **Layer drop-down list**: contains all layer names of the current drawing, which is where you choose the layer you want to set as current.
- **Color button**: displays the current color. Click the button open the **Color** dialog box for selecting another color, which is applied when creating new entities.
- **Linetype drop-down list**: contains the linetypes defined in the current drawing, which is where you select another linetype.

**Note:** We recommend that you set both the current color and the current linetype to the logical BYLAYER. This means that the entities take on the color and linetype of the current layer. It also means that entities moved to another layer will automatically change to the color and linetype of that other layer. In the long run, this makes drafting with FelixCAD much easier.

You find detailed references about layer concepts, linetype, and color properties in Layer and Object Properties.”

## The Control Panel

The control panel is located on the left side of the screen. This panel contains a permanent series of command or function buttons. You can turn the control panel on and off by selecting **File > Desktop Manager > Left Toolbar > On** or **Off**.



The symbols are arranged in the control panel in five groups:

- Precision Drawing Assistants:**  
 Explanation of these commands is found in “Drawing with Precision.”
- Controlling Drawing Views:**  
 Explanation of these commands is found in “Viewing Drawings.”
- User Coordinate Systems and Layout Mode:**  
 Explanation of user coordinate systems are in “Viewing Drawings”; the layout mode is explained in “Printing and Plotting & Using the Layout Mode.”
- Dimensioning, Hatching, and Text:**  
 Dimensioning is discussed in “Dimensioning”; hatching is explained in “Hatching”; and text editing is covered in “Text Objects.”
- Inquiry Commands:**  
 Explanation of these commands is found in “Inquiry and Information Functions.”

The control panel

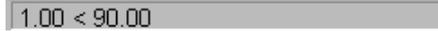
## The Status Line

The status line is located at the bottom of the program's desktop. It contains a series of information fields that report on the status of FelixCAD. You can turn off and on the status bar with the **File > Desktop Manager > Status Bar**.



The status line displaying help text (left) and X, Y, Z coordinates at right

The first field, at left, displays the current drawing coordinates in relative mode. For example, 1.00 < 90.00 means the entity being drawn is currently 1.0 units long, at 90.00 degrees from the last point.



The status line displays relative coordinate during drawing

When the cursor is over the button of a symbol bar, menu item, palette (toolbar), a concise description of the command's functions is displayed here. In the center, the current coordinate system mode is displayed:

- **WCS:** world coordinate system (the default).
- **UCS:** user-defined coordinate system.

The last three information fields specify the X, Y, and Z coordinates of the current cursor position.

## The Drawing Area

The *drawing area* is the part of the desktop where you draw and edit drawing entities.

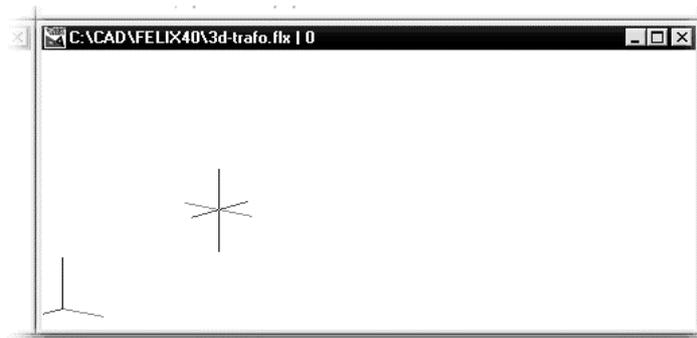
You can open up to four drawings, each with up to four viewports, simultaneously in the drawing area. As mentioned earlier in this chapter, each drawing and each view is represented by their own window. The windows can be tiled, cascaded, or enlarged to fill the drawing area.

When more than one window is open, only one is the *active* window. The active window is recognized by its dark title bar. You can only work on the drawing in the active window. To make a window active, simply click on it.

Within the drawing area, the cursor appears as a tri-color crosshair:

- **Red:** x-axis.
- **Green:** y-axis.
- **Blue:** z-axis.

The tri-color cursor makes it easier to know where you are when working with three-dimension drawings. (Outside of the drawing area, the cursor assumes the typical Windows pointing shapes.) You can change the colors of the cursor with dialog box displayed by the **File > Configure > Colors** command.



The tri-color cursor (center) and coordinate symbol (left corner) in a 3D view

FelixCAD places a coordinate symbol in the lower left corner of each drawing window. This symbol indicates the position of the axes of the coordinate system. It uses the same color coding system as the tri-color cursor. You can turn off the coordinate symbol with the **TIcon** command.

## **ENTERING COMMANDS**

To draw and edit with FelixCAD, you enter commands and their options, define points, distances, angles, and select entities. FelixCAD offers several options for doing this: all command input can be performed by the keyboard, or by the mouse controlling the cursor. Or, you can combine the two.

For example, when drawing a line you can define the first point by typing the X, Y coordinates of the start point on the keyboard, and then define the second point by pointing the second point with the cursor control. The selection of input device is your decision.

## Mouse Buttons

The buttons on your mouse perform specific functions. We recommend that you memorize the meaning of these buttons:

### Left Mouse Button

The left mouse button indicates points, select entities, and selects commands from pull-down menus and palettes (toolbars).

### Right mouse button

The right mouse button is the same as the **Enter** key. It is used to:

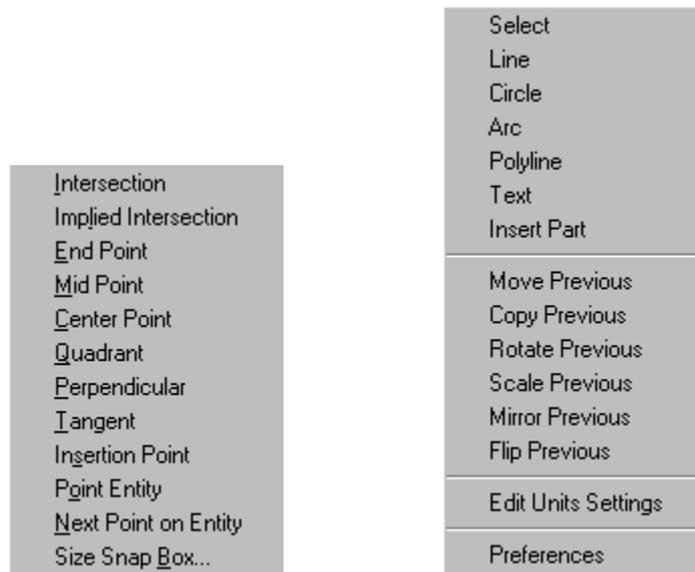
- Confirm input and output operations.
- Accept default values.
- Repeat a command.

### Middle Mouse Button

The middle mouse button displays a floating menu. If your mouse has just two buttons, hold down the **Ctrl** key and press the left mouse button.

During a command (when is command is active), pressing the middle mouse button displays an object snap menu with available options.

When no command is active, the middle mouse button displays a menu of the most commonly used commands. (The commands are explained later in this manual, with more detail, in the appropriate sections.)



The floating menus displayed by the middle mouse button

## Command Options

Almost all commands include options. For example, the **Circle** command has several options so that you can specify the circle several different ways, such as by centerpoint or circumference, radius or diameter, or tangent points.

## Keyboard

Unlike most Windows applications, FelixCAD allows you to type commands via keyboard input. The advantages are that you work faster and can enter precise parameters. In fact, combinations of both keyboard and mouse input are still more efficient. An example would be starting a Line command by simply hitting the "L" key and a right mouse click to start the command, etc.

**Note for AutoCAD Users:** Refer to the "FelixCAD for AutoCAD" companion guide for instructions on how to convert the default FelixCAD command "alias" to an AutoCAD emulated command alias.

Most of the commands that create, modify, and detail drawings are keyboard-based, as are FLISP expressions. Some specific Window operations are accessed via *accelerator keys*, such as Ctrl+F4 to close a window.

To type a command at keyboard, enter the command name behind the *command prompt* in the Command Line area. The command prompt looks like this: >

You are then asked to enter additional input, which you can do with the keyboard or the mouse:

```
> circle
Center point: [pick]
Radius: 2.35
>
```

## Pull-Down Menus

Commands (and their options) can be selected using the pull-down menus. You can click on the menu items or use the keyboard.

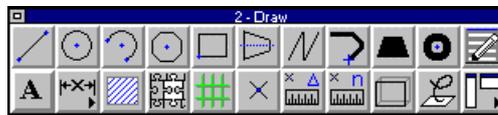
To access the **Circle** command from the menus, click on the **Draw** menu, then click the **Circle** item on the pull-down menu.

To use the keyboard, hold down the **Alt** key then press the underlined letter of the menu item. For example, for the **Circle** command, hold down **Alt** and press **D**. Let go of the **Alt** key and press the **C** key.

## Symbol Buttons

Frequently used commands are grouped together in palettes. The palettes contain rows of button containing symbols. 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.



The **Draw** symbol palette

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

## Transparent Commands

Most commands are executed at the **>** command prompt. In fact, you usually cancel one command (by pressing the **Esc** key). Sometimes, however, you need to use one command during another command. For example, you may need to pan the drawing over during the **Line** command.

There is a special group of commands that you can use in the middle of another command. These commands are called *transparent* commands. You use a transparent command by prefixing the name with an apostrophe, such as **'Zoom**.

Examples of transparent commands include:

- Changing the view of a drawing – **'Zoom**, **'Pan**.
- Arranging the drawing windows – **'WCascade**, **'WTile**.
- Setting the precision – **'Grid**, **'Snap**, **'Ortho**, an object snap functions.
- Switching between command line area and text window – **'TScreen**.
- Loading and closing palettes – **'PalMan**.
- Calling the **'Help** command.

After the execution of the transparent command, the program continues with the original command.

**Note:** Transparent commands *cannot* be executed if the program is requesting input of text or if a command's dialog box is active.

## Repeating a Command

Often, you find you need to repeat a command. For example, you might be using the **Line** command over and over again. To repeat the same command by pull-down menu or keyboard entry gets annoying and inefficient.

You repeat the last command by one of three methods:

1. pressing the **Enter** key or
2. clicking the right mouse button or
3. Pressing the **Spacebar** key.

For example, the first **Enter** ends the **Line** command; the second **Enter** repeats the command:

```
> line
From point: P1
To point: P2
To point: P3
To point: [Enter]

> [Enter]
LINE
From point: P4
To point: P5
```

To point: **P6**  
 To point: **Close**

**Note:** Some commands, like **Point** and **Trim**, automatically repeat themselves.

## Canceling a Command

Sometimes you want to cancel a command before it has been completed. This can happen if you start a command was activated by accident, or if the command automatically repeats itself.

To cancel the execution a running command:

- Click the symbol .
- Press the **Esc** key.

This interrupts the execution of the current command. At the command line, you see the message **\*\*\* Cancel \*\*\***.

## Reversing a Command (Undo)

If you recognize that you made a mistake, you can reverse the effect of the command. FelixCAD archives all steps you've taken during the current drawing session. This archive is called the *undo list*.

The **Undo** command reverses the each step sequentially. You can keep using the **Undo** command until you get back to the beginning of the drawing session. At that point, FelixCAD declares, "Undo list is empty!".

- Click the Undo button .
- Select **Undo** from the **Edit** menu.
- Type **U** at the keyboard and press **Enter**.

Some commands are not stored in the undo list. These include commands that control the viewport (**Zoom**, **Pan**, and **Window**), set parameters, and other transparent functions. These steps **cannot** be reversed by the **Undo** command.

The undo list is kept only with the current session of the program. It is not saved with the drawing.

## Restoring the Reversal (Redo)

If you undo too far, you can use the **Redo** command. It cancels the last **Undo** operation. The drawing is restored to the last state prior to the **Undo** command – and this is only possible if there has been no other command executed between the **Redo** and **Undo** commands

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

## Deleting & Undeleting Entities

While creating a drawing, you may need to erase certain parts of the it. The **Delete** command erases entities from the drawing, as follows:

- Click the symbol .
- Select the **Delete** command from the **Edit** menu.
- Type the **Delete** command in the command line area.

You can use the **Delete** command these entities:

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

To delete an entity that is in a part (block), you must first explode the part with the **Explode** command. Delete the entity, then regroup the remaining entities into a part again with the **PartDef** command.

Entities that have been deleted by mistake can be restored by the **Undelete** command:

- Click the symbol .
- Select the **Undelete** command from the **Edit** menu.
- Type the **Undelete** command at the command line.

This causes all previously deleted entities to be restored.

## Renaming Names

The **Rename** command allows you to alter the names associated with blocks, layers, linetypes, text styles, dimension styles, user coordinate systems, and views. Take these steps:

- Start the **Rename** command.
- Select database table to be renamed from the option bar.
- Enter the name of the object to rename.
- Type in the new name of the item.

```
> RENAME
Rename symbol (Block, Dimstyle, Layer, LType, Style, Ucs,
or View): LA
Current name: dimensions
New name: dim
```

**Note:** You can rename these named symbols with the **Drawing Explorer**.

## Purging the Drawing

The **Purge** command cleans the drawing file of unused named objects. You can remove blocks, layers, linetypes, text styles, or dimension styles that you have defined or loaded before but are not used in the drawing.

```
> PURGE
Purge unreferenced symbols (Blocks, Dimstyles, Layer, LTypes,
Textstyles or All):
```

The **Purge** command can *selectively* purge the drawing, as follows:

- You can use the command for deletion of unused named objects within only one of the drawing database tables. To do so, first select the type of table to be purged. You are prompted for the names -- one after the other -- of unused named objects of the specified table.
- If you want to purge all drawing database tables, reply **All** to clean up the entire drawing in one operation. You may respond with a name pattern to limit the selection to certain names by using wildcard specifications (for example: TEMP\*).

### To purge a drawing file

To purge all unreferenced symbols in any drawing database table, respond as follows:

```
> PURGE
Purge unreferenced symbols (Blocks, Dimstyles, Layer, LTypes,
Textstyles or All): ALL
Names to delete <* = All>: *
Confirm each name? <Y>: N
```

**Notes:** Purging a drawing can significantly reduce its file size. This operation is a good habit before archiving or backing up a drawing file.

Some standard objects cannot be purged: These include the layer named 0, the linetype CONTINUOUS, and the text style STANDARD.

The **Purge** command does not handle named views and user coordinate systems, because these named objects are never referenced by another component of the drawing. Instead, you delete unused views and UCSs via the **View** and **Ucs** commands.

Only one level of nested blocks is purged at a time. You may need to repeat the **Purge** command.



### **DRAWING XPLORER \***

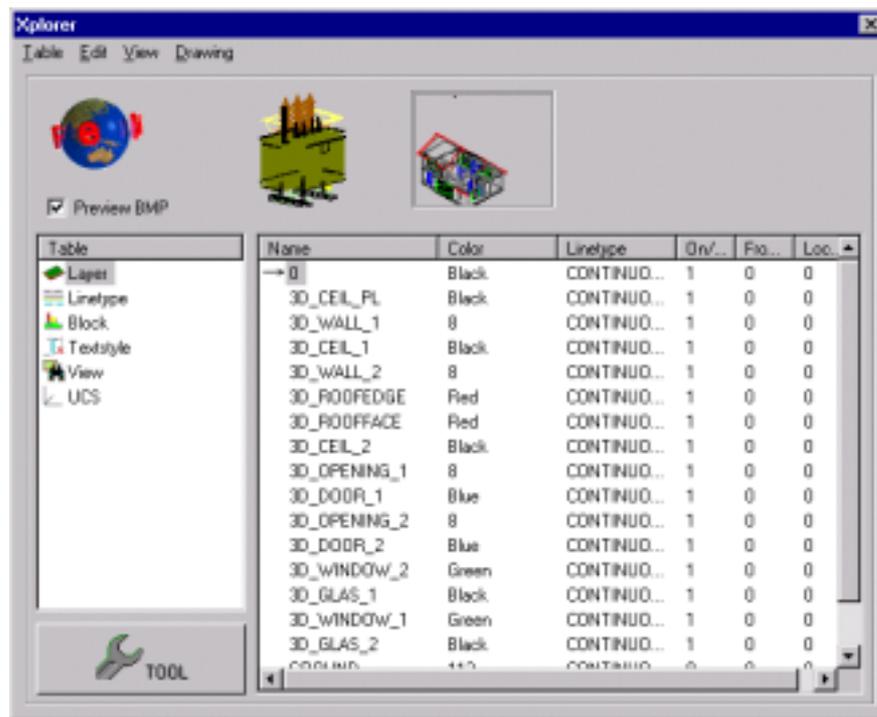
The Drawing Xplorer allows you fast and uncomplicated access to important information about the currently open drawing(s). The Xplorer can be used to alter properties of the current drawings.

More precisely, the Drawing Navigator is a tool to manage the named objects of the drawing database tables. You can handle the tables and table entries of layers, linetypes, text styles, views, user coordinate systems, and blocks.

To activate this tool, type the command **Xplorer** or select **Drawing>Explorer** from the **Edit** menu.

The **Xplorer** window displays a preview bitmap of the currently open drawings in the area under the menu bar. To switch to another drawing, click the appropriate bitmap. A tooltip over the bitmap displays the drawing name, including drive and path.

The **Table** list box allows you to select a specific table. The named objects for the selected table are shown in the **Name** list.



The Drawing Xplorer permits the following:

- Delete unused, unreferenced table entries (allowed only for the active drawing).
- Rename table entries (allowed only for the active drawing).
- Set the layer, linetype, text style, user coordinate system, or named view (allowed only for the active drawing) by double clicking to one of the table entries.
- Insert a block by double clicking to its name.

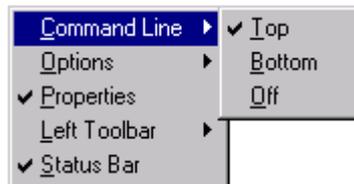
Click the **Tool** button to display the corresponding dialog box associated with the table name. For example, if **Layer** is the table selected in the Drawing Navigator, the **Tool** button displays the **Layer Manager** dialog box. This lets you create new layers.

## Customizing the User Interface

You can customize FelixCAD's user interface, so that the program better matches your needs and habits. You may remove (and reinstate) components, customize all the buttons in the tool areas, and change the menu system.

### Modifying the Desktop Layout

The **DeskConf** command (**File > Desktop Layout**) displays a pop-up menu that lets you rearrange the desktop. The menu allows you to determine the layout of the command line, option menu or bar, property bar, left toolbar (control panel), and status line.



The Desktop Layout pop-up menu

#### Command Line

The *command line* can be placed at the top of the screen (below the menu bar), or at the bottom of the screen (above the status line). If you are an advanced user familiar with the program's command set, you might even turn off the command line.

#### Options

The *options menu* can be displayed as a bar (under the command line) or as a floating menu. Once again, if you are an advanced user familiar with the command options, you can turn off the option menu and bar.

#### Properties

The *property bar* can be turned on or off. We recommend you turn it on.

#### Left Toolbar

The *control panel* (also known as the left toolbar) can be turned on or off.

#### Status Bar

The *status bar* can be turned on or off. We recommend you leave it on.

## Customizing Toolbars

The toolbars (we call them palettes) assemble buttons that represent a common set of commands, or sequence of commands.

### Dragging an Icon to a Toolbar

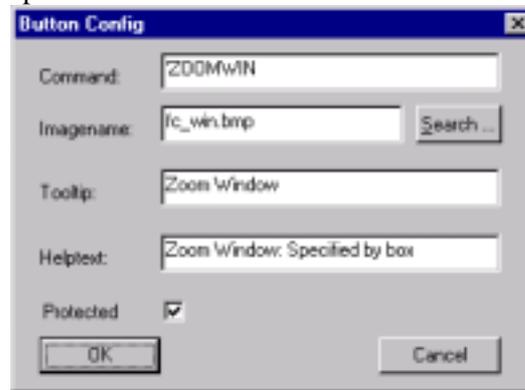
You add command icons (or symbols) to a toolbar by simply dragging a button from a palette into a toolbar. Follow these steps:

1. Open a palette containing the icon of a command which you want to place into a toolbar. Type the **PalMan** command and select palettes from the **Palette Manager**.
2. Hold down the left mouse button on the icon you want and drag the icon to the desired location within a toolbar.
3. Release the left mouse button to drop the icon at that position.

### Adding an Icon to a Toolbar

Follow these steps to add an icon to a toolbar:

1. Right-click the button you want to place a new icon on. The following dialog box opens:



Specifying the properties of a toolbar button

2. Specify the command name, the bitmap file, and the messages for the tooltip of the button and the help string displayed in the status bar.
3. Click **OK**.

If you want to protect the button from being overwritten, enable the *Protected* check box.

### Deleting an Icon from a Toolbar

You can delete an icon from the toolbar only when the button is not protected. You unprotect a button by right-clicking the button and then disable the **Protect** option in the **Toolbar Configuration** dialog box.

Follow these steps to delete an icon from a toolbar:

1. Hold down the left mouse button over the icon you want to delete.
2. Drag the icon onto the drawing window, or to the background area of the drawing area of the desktop.
3. Release left mouse button.

The icon is removed from the toolbar. You may then place another icon onto that button.



### Configuring FelixCAD

The **Config** command (**File>Configure**) initiates a dialog allowing you to set up a variety of system parameters. A series of 5 tabs along the top of the dialog allows you to set:



#### Common Tab

**File locking** - If checked, other users trying to access the current drawing across a network will be denied permission.

**Safety** - Allows you to select auto-saving of drawings, along with the file extension and interval for automatic backup. Note that each time a file is auto-saved, the system will experience a slight delay (depending on the power of your computer and the size of the drawing), and the contents of the undo list are cleared. For this reason, it is advised you set the interval to 20-30 minutes.

#### Directories Tab

This is where all working directories of the system are set. Unless you are an advanced user, it is advised you do not alter the default settings.

## Alias Commands Tab

The alias command tab allows you to quickly and easily assign keyboard command "shortcuts" for any FelixCAD command.

The standard command alias includes a variety of one or two character command shortcuts designed to speed up your work. An example would be typing ZW at the FelixCAD command line will execute the recursive command Zoom, then Window.

You may add new command aliases, or edit existing aliases as needed.

**Note:** Command aliases are saved in files with the extension .KEY. FelixCAD 4.01 or higher ships with an AutoCAD command alias (Facad14e.key), which can be used in place of the default FelixCAD .KEY file. Refer to the FelixCAD for AutoCAD guide for further details.

## HotKeys Tab

FelixCAD allows user defined setting of "hotkeys" to further improve your productivity. You may reassign the default values to suit your own preferences.

## Colors Tab

This is where you can reassign the default screen colors for the coordinate icon (all 3 axis), the background screen for the drawing window, the drag cursor for drawing, and the selection color for editing.

You can also assign alternate fonts for the system and tooltip text.

# Drawing with Precision

During the CAD drawing process, it is often necessary to specify precise coordinates and other values used in drafting. When FelixCAD needs input from you, a prompt will appear in the text window. The prompt asks you to enter a value, examples of which include:

- Coordinates for defining points of the drawing, such as the start and end points, center points, insertion points, base points, or target points.
- Distance.
- Angle.
- Widths, for example the width of a line.
- Numbers.

In this chapter, you learn how to provide data input to FelixCAD. Also, this chapter describes commands that serve as drawing aids, such as:

- Precision aids (grid, snap grid, and orthogonal mode).
- Object snap functions.
- Deleting and undeleting drawing objects.

Most of the commands described in this chapter do not create or modify entities. The settings (grid, snap) and methods of selecting and snapping objects make the drawing much more precise.

## **DATA INPUT**

FelixCAD lets you input data, such as coordinates and distances, via the keyboard or the cursor.

### **Coordinate Input**

When you provide a FelixCAD command with coordinate data, they can be entered as *absolute* or as *relative* coordinate values.

#### **Absolute Coordinates**

Absolute coordinates use the origin of the current coordinate system as their reference (starting) point. In the case of angles, the reference is the angle to the X-axis of the coordinate system.

An example of absolute X, Y-coordinates (2D) is 34, 4. An example of absolute X, Y, Z-coordinates (3D) is -0.12, 5.7, 1.0.

### Relative Coordinates

Relative coordinates use the last defined point as their starting point. Relative coordinates are specified by adding the @ sign before the value.

An example of relative coordinates is @10, 12 – this defines a point which has a distance of 10 drawing units in the positive X-axis and 12 in the Y-axes from the last entered point.

Moving an entity can use relative coordinates, and looks like this:

```
Select objects: [pick]
Basis point: 5,2
Target point: @3,0
```

### Cartesian Coordinates

Cartesian coordinates define a 2D or 3D point by its X, Y, and Z values. These values are entered separated by a comma. A dot is used as a decimal point. Numbers may be positive or negative. If no Z value is entered, it is interpreted as Z=0.

A sequence at the keyboard might look like this:

```
> line
From point: 2.5,0
To point: 5.75,0
To point: 0,3.25,2.5
To point: [Enter]
```

### Polar Coordinates

Polar coordinates define a 2D point by its distance from the origin, and the angle in the X,Y-plane (from the X axis). The < sign separates the two values.

For example, 1.41<45 defines a point at a distance of 1.41 drawing units from the origin of the coordinate system and an angle of 45 degrees from the X axis.

### Spherical Coordinates

Spherical coordinates define a 3D point by its distance from the coordinate origin, the angle *in* the X, Y-plane, and the angle *to* the X, Y-plane.

For example, 10<60<45 describes a 3D 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 3D point by its distance from the coordinate origin, the angle in the X,Y-plane and its Z value. Distance and angle are separated by < , angle and Z-value by comma.

For example, 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 X, Y-plane, and a Zvalue of 5 drawing units.

### **Distance**

Some commands, such as the **Trim** command, require that you input a distance. You do this in two ways:

- Enter a numeric value, either a whole number (an integer), or a real number. You can use decimal numbers (12.34), scientific notation (1.234E+10), or as a fraction (12-3/4).
- Point the distance by picking two points with the cursor.

### **Widths**

Some commands, such as the **Polyline** command, may require that you input a width. You do this in the same way as defining a distance.

### **Angles**

When a command, such as the Arc command, asks for an angle, you can provide the angle in the same way as distances. Angles are normally specified in degrees, counterclockwise from the starting point. The starting point, 0 degrees, is located at “East,” the positive X-axis.

Entering a distance with an angle requires the use of the < sign before the angle, such as 10<45. Relative angles also go in the counterclockwise direction.

## Numbers

Some commands require a discrete number. For example, the **Polygon** command asks for the number of sides. The value you provide must be a whole number.

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 Format	Example
<b>Absolute Coordinates</b>		
Cartesian (2D or 3D)	X, Y X, Y, Z	3, 8 3,8,6
Polar (2D)	distance<angle	7.5<45
Spherical (3D)	distance<angle,Z	7.5<45,12.2
Cylindrical (3D)	distance<angle1<angle2	7.5<45<33
<b>Relative Coordinates</b>		
Cartesian (2D or 3D)	@deltaX,deltaY, @deltaX,deltaY,deltaZ	@3,8 @3,8,6
Polar (2D)	@distance<angle	@7.5<45
Cylindrical (3D)	@distance<angle,Z	@7.5<45,12.2
Spherical (3D)	@distance<angle1<angle2	@7.5<45<33

## Pointing with the Cursor

*Pointing* is using the cursor to specify data on the screen. You can use the cursor to specify points, straight lines, angles, and other quantities. FelixCAD always interprets the cursor position as a coordinate: either X, Y, Z or distance<angle.

Pointing has one clear advantage over keyboard entry: dynamic preview (or dragging) provides a kind of preview image of the result of your action.

We recommend that you use pointing you must relate to entities that already exist in the drawing. For example, the center of a circle could be defined by the intersection of two lines. In this case, it is easier to indicate the center point by pointing at the intersection.

Using object snap ensures that the accuracy of the coordinate input by pointing is improved. Object snap modes recognize geometric features, such as start and end points, intersection, center point, and perpendiculars. Using object snap modes results in greater precision – and faster drafting -- than entering coordinates at the keyboard. Object snap modes are summarized later in this chapter.

## Coordinate Filters

Coordinate filters lets you mix keyboard entry with cursor picks. You can, for example, point the X-coordinate on the screen and type the Y-coordinate value. Here is an example with the **Line** command:

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

The **x, 10** is an example of using coordinate filters. You provided the Y-coordinate via the keyboard (10), then picked the X-coordinate on the screen.

In addition, this example shows the use of object snap modes. The words **end** and **cen** are endpoint and center point, respectively. So, when you picked the X-coordinate, you used the ENDpoint object snap to pick at the end of an open object, such as a line or arc.

The coordinate filters are:

Filter	Meaning
X	Provide Y (and Z), ask for X.
Y	Provide X (and Z), ask for Y.
Z	Provide X and Y, ask for Z.
x,y	Ask for X and Y, optionally provide Z.
y,z	Ask for Y and Z, provide X.
x,z	Ask for Z and Z, provide Y.
x,y,z	Ask for X, Y, and Z.

## From

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

```
> line
From point: from
Base point: [pick]
Offset point: @3,3
To point:
```

The **From** option is different from object snap modes in that it does not expect you to pick a geometric features. Instead, it serves to retrieve a relative point from a temporary base point. After this, control returns to the command in progress.

When prompted for the “Base point:”, you can use absolute and relative coordinates, as well as object snap modes.

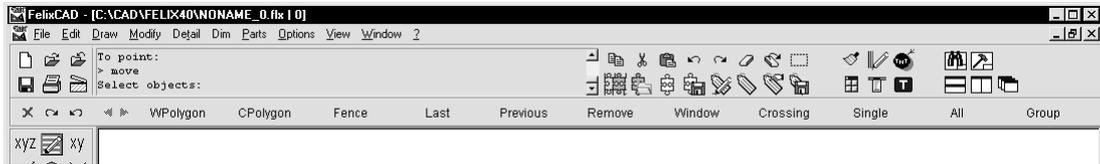
But when you are prompted for the “Offset point:”, you should only use *relative coordinates* (use the @ prefix). You can specify an absolute coordinate, however, this cancels the specification of a relative point. As an alternative, you can invoke an object snap option at this prompt.

## OBJECT SELECTION

Most editing commands require that you select one or more entities in the drawing. This happens any time you see the following prompt:

```
Select objects:
```

FelixCAD provides a rich variety of methods for selecting multiple objects. The option bar displays these options:



You can pick one or more entities at the “Select objects:” prompt. (The prompt keeps repeating until you press **Enter**.) All entities you select are added to the *selection set*. The **Remove** and **Add** options allow you to remove and add entities from the selection set. Selected entities are shown in color (magenta, by default).

### Object Selection Modes

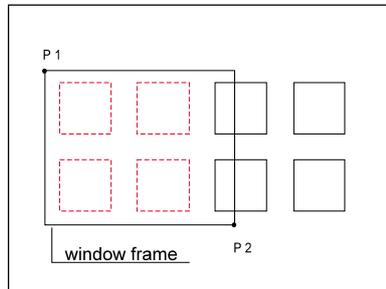
To select options, you can simply pick them, one at a time. It is more efficient to use an object selection mode to pick multiple entities. The options are:

Object Selection Mode	Abbreviation
Add	Ad
All	A
Cpolygon (crossing polygon)	Cp
Crossing	C
Fence	F
Last	L
Previous	P
Remove	Re
Single	Si
Window	W
Wpolygon (window polygon)	Wp

## Window

The **Window** selection mode defines a rectangle by two diagonal corner points. All entities completely within the window are selected.

```
Select objects: w
First corner: [pick]
Other corner: [pick]
Select objects:
```

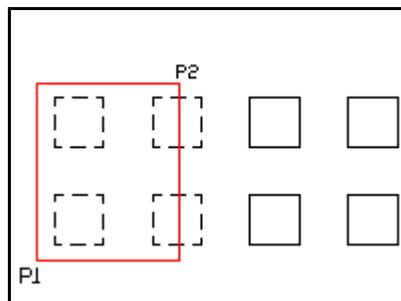


Continue object selection or terminate it by pressing **Enter**.

## Crossing

The **Crossing** selection mode defines a rectangle by two diagonal corner points. All entities within and crossing the window are selected.

```
Select objects: c
First corner: [pick]
Other corner: [pick]
Select objects:
```

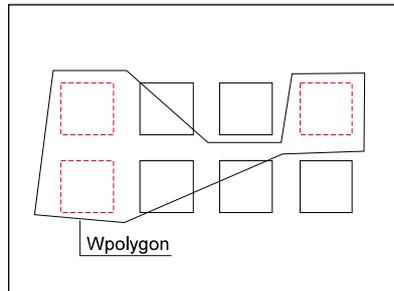


Continue object selection or terminate it by pressing **Enter**.

## WPolygon

The **WPolygon** (short for “window polygon”) selection mode defines a polygon by two or more vertex points. All entities within the polygon are selected.

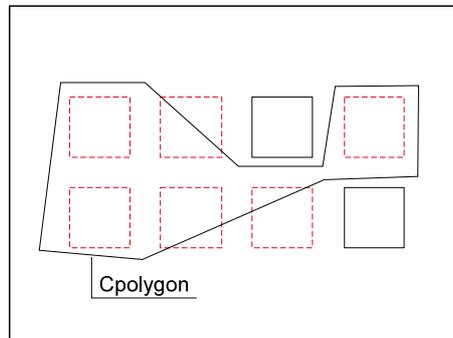
```
Select objects: wp
First point: [pick]
Next point: [pick]
Next point: [pick]
Next point: [pick]
Next point: [Enter]
Select objects:
```



## CPolygon

The **CPolygon** (short for “crossing polygon”) selection mode defines a polygon by two or more vertex points. All entities within and crossing the polygon are selected.

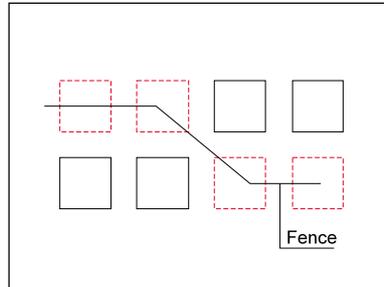
```
Select objects: cp
First point: [pick]
Next point: [pick]
Next point: [pick]
Next point: [pick]
Next point: [Enter]
Select objects:
```



## Fence

The **Fence** selection mode defines a polyline by two or more vertex points. All entities crossing the polyline are selected.

```
Select objects: f
First point: [pick]
Next point: [pick]
Next point: [pick]
Next point: [Enter]
Select objects:
```



Continue object selection or terminate it by pressing **Enter**.

## Last

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

```
Select objects: l
Select objects:
```

It is not possible to repeatedly use the Last mode to add to the selection set, since it only works for the *one* last drawn entity.

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

## Previous

The **Previous** option restores the previous selecting set. All entities are marked (and highlighted) from the previous selection operation. This option makes it fast to select the same selection set over and over.

```
Select objects: p  
Select objects:
```

**Note:** If the previous selection set lies on a frozen layer, you receive the message in the command line area that no entity was selected.

## All

The **All** option selects **all** entities in the current drawing -- with the exception of those located on frozen layers.

```
Select objects: a  
Select objects:
```

## Remove & Add

The last field of the option bar contains a toggle button to switch between adding and removing entities to the current selection set.

```
Select objects: r  
Remove objects: [pick]  
Remove objects: add  
Select objects:
```

Click the **Remove** button on the option bar (or type **r**) to change the selection mode to remove entities. You can only remove entities that are already in the selection set; entities already selected and marked are removed from the selection set. The **Add** mode allows you to select additional entities. This option only makes sense after you use the **Remove** option.

## Single

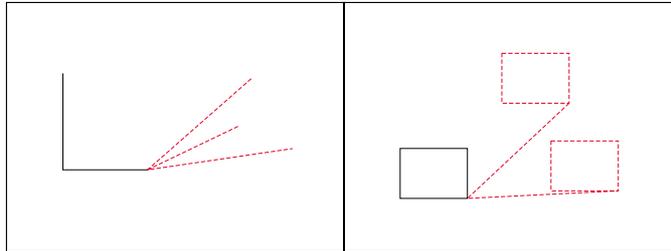
The **Single** option deactivates all object selection modes for the next and subsequent operations, with the exception of the standard modes Window and Crossing.

```
Select objects: si  
Select objects:
```

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

## DYNAMIC PREVIEW

Dynamic preview mode is a visual aid that allows you to see the result of a pending operation before it is executed. If necessary, you can make corrections. Dynamic preview also aids you in understanding the effect of a command on your drawing.



## Rubberband

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.
- Radius or diameter of a circle.
- Distance between starting- and ending-point of an arc.
- Next corner point of a 2D or 3D plane.

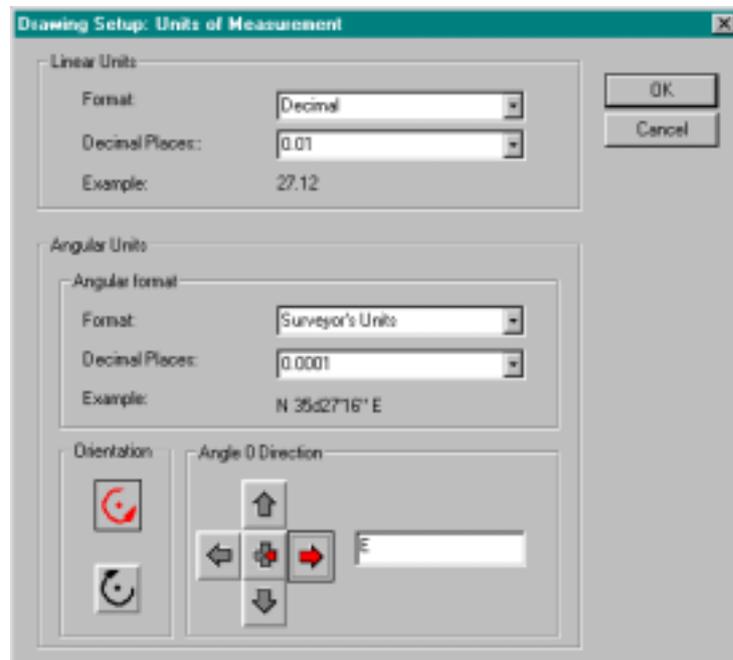
When used during modification of objects, the rubberband shows the distance and angle from one base point to a second point, as well as the target point for moving, copying, scaling or other modifying steps.

## Object Preview

The *object preview* 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.

## PRECISION AND UNITS SETTINGS

FelixCAD allows you to set the format and display accuracy of numbers and angles. The settings are saved with each drawing, so they can be set differently in each drawing. The **Setup** command displays the **Drawing Setup** dialog box:



### Linear Units

Linear units can be given a display accuracy ranging from zero to eight decimal places. In the case of fractional units, the display accuracy ranges from 0 (no fractions) to 1/256. You can select five kinds of units:

- Decimal units (metric) – 12.3456.
- Fractional units (unitless fractions) – 12 3/4
- Architectural units (feet and fractional inches) – 12' -3/4"
- Engineering units (feet and decimal inches) – 12-3.456'
- Scientific units (exponential notation) – 1.234E+02

## Angular Units

The **Angular Units** can be given a display accuracy ranging from zero to eight decimal places. FelixCAD recognized these angular measurement systems:

- Decimal degrees – 12.3456
- Degrees - minutes - seconds – 12d34'56.
- Grads (400 grads in a circle) – 12.34g
- Radians – 12.34r
- Surveyors units – N12d34'56 E

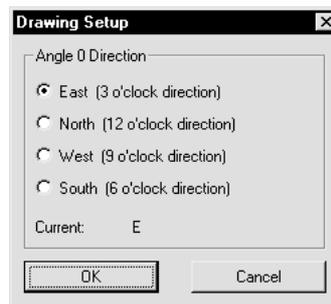
## Orientation

In most drawings, angles are measured in the counterclockwise direction. This is the default in FelixCAD and is called **Standard Angles**.

You can, however, change the orientation so that positive angles are measured in the clockwise direction, called **Compass Angles**.

## Angle Zero Direction

In most forms of drafting, zero degrees is located at the positive X-axis (East, or 3 o'clock). FelixCAD allows you to change the location of zero degrees. For example, surveyors normally set zero degrees as up on the page (North, or 12 o'clock). The display of the status bar is updated depending upon the settings of the angle zero direction and the orientation.



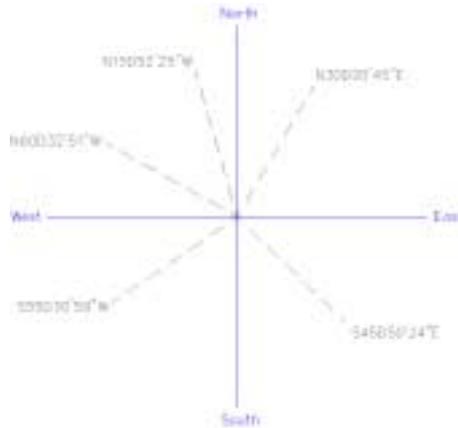
## Surveyor Units Entry

Because the surveyor units are so different from the other forms of angle entry, we have included a few examples:

Type of coordinate	Input	Example (Surveyor Units)
<b>Absolute Coordinates</b>		
Polar	distance<angle	7.5< N34D30'45"W
<b>Relative Coordinates</b>		
Polar	relative distance<angle	@7.5< N34D30'45"W

In surveyor's notation, the N stands for North. The 34D is 34 degrees from North. The 30' is an additional 30 minutes, and the 45" is an additional 45 seconds from North. The W indicates that the angle is towards the West.

There are 60 seconds in a minute, 60 minutes in a degree, and 360 degrees in a circle. Surveyors divide the circle into four parts, each with 90 degrees measured from the North or South poles in the direction of East or West.



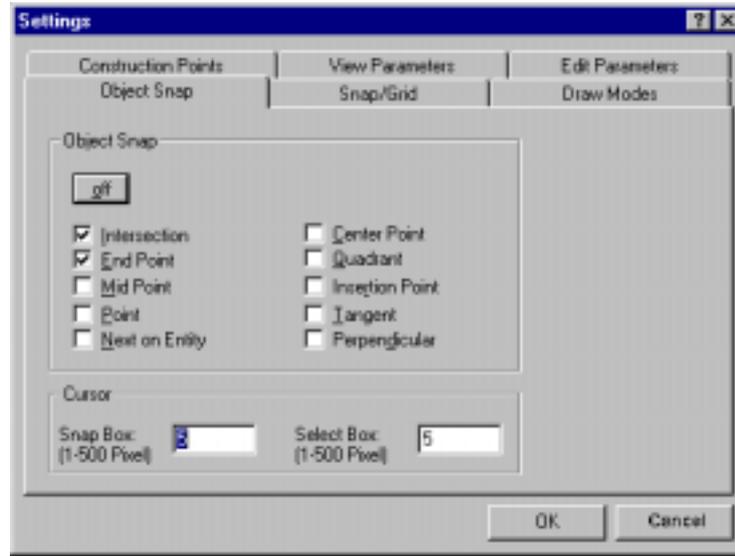
In the figure above, we see the four points of a compass: North, South, East, and West. As examples of how the Surveyor's Units work, several lines have been drawn in various directions. We labeled these lines with the correct bearings. Note that all the letters used in the bearings are in capital letters, as required by FelixCAD.

**Note:** When entering the N, S, E, W and D, be sure to enter them as capital letters -- lower-case does not work.

## ***DRAWINGS SETTINGS***

The **Settings** command (**Options > Settings**) is the primary command for setting up the drawing environment. The command displays the **Settings** dialog

box, which has six tabs that group together related settings. *AutoCAD users will see this commands similarity with the Preferences option (previously the CONFIG command) for system parameters.*



Tab	Command	Reference
Object Snap	PrecPar	Chapter 2 "Drawing with Precision"
Snap/Grid	PrecPar	Chapter 2 "Drawing with Precision"
Draw Modes	DrawMode	Chapter 5 "Basic Drawing Elements"
Construction Points	DrawMode	Chapter 5 "Basic Drawing Elements"
View Parameters	ViewPar	Chapter 3 "Viewing the Drawings"
Edit Parameters	EditPar	Chapter 6 "Modifying Drawing Objects"

## PRECISION DRAWING AIDS

Working with CAD involves drawing new entities, modifying existing objects, and doing detailing operations (dimensioning, for example). To assist you, FelixCAD provides you with precision drawing aids that help you precisely define specific points in the drawing.

The precision aids include snapping to a grid, drawing orthogonally, and object snap modes. These aids can be turned on, off, or changed – even in the middle of a command.

With **Snap** activated, your cursor picks are made to the nearest snap points. For example, if snap is set to 0.1mm, you make picks to the nearest 0.1mm. You can set the snap spacing to any value. For example, if you want your drawing accurate to the nearest 1/4", you would set the snap spacing to 0.25.

The **Grid** display covers the drawing area with a visible array of dots. This helps you in several ways: (1) you see the limits of the drawing; (2) you get a sense of the drawing scale; and (3) you see the snap distance, a multiple thereof. As with the snap distance, you can set the grid spacing to any value. For example, if you want a 1-foot grid, you set the grid spacing to 12 inches.

The grid does not cover the entire drawing. Instead, its extent is limited by the **LimMin** and **LimMax** system variables:

- **LimMin** sets the lower left corner (limits) for the grid display area.  
**LimMax** sets the upper right corner (limits) for the grid display area.

**Ortho** (short for “orthogonal”) mode constrains the movement of the cursor to the horizontal and vertical direction. This makes it easier to design and place parallel and orthogonal lines and other entities.

**OSnap** (short for “object snap”) modes allow you to work with geometrically significant points of entities. These geometric features include end points, intersections, center points, and so forth. You use osnap modes in two ways: turn them on permanently (until turned off again), or temporarily (active for a single pick). You can display the object snap modes by clicking the middle mouse button, or via the **Precision Aids** dialog box.

## CONTROLLING PRECISION PARAMETERS

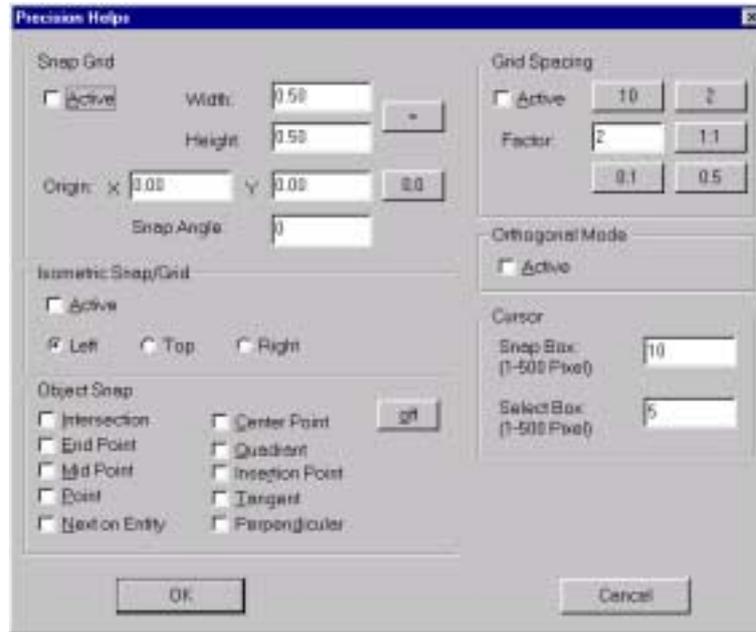


The **PrecPar** command (short for “precision parameters; **Options > Draw with Precision**) displays a dialog box that contains settings for drawing with precision. You can also access the precision drawing aids by clicking the symbol (shown above) in the control panel.

To *toggle* (turn on and off) **Ortho**, **Snap**, and **Grid** mode during a command, click on one of these buttons:



These control panel buttons toggle **Ortho**, **Snap**, and **Grid** mode.



## Snap Grid

The **Snap Grid** area controls the spacing of the snap. With **Active** turned on, the cursor can only select points that are positioned directly on the grid. **Width** and **Height** lets you specify the snap distance. **Origin** lets you change the origin of the snap. The **Snap Angle** field lets you rotate the snap, which also rotates the **Grid** and **Ortho** mode. The snap grid always follows the axes of the current coordinate system.

**Note:** The snap distance does not influence points defined by keyboard coordinate input. Even with snap activated, you can define points not on the grid by keyboard entry.

### Width & Height

The **Width** and **Height** fields let you enter values for the horizontal and vertical distance between the snap points. Units of measurement for these values are drawing units. The value 1.0000 represents the distance of one drawing unit. The = button sets the height value equal to the width value.

### Origin

The **Origin** fields allow you to define the origin of the snap on the X and the Y-axes.

**Note:** This setting also affects the origin of the grid and hatch patterns.

### Snap Angle

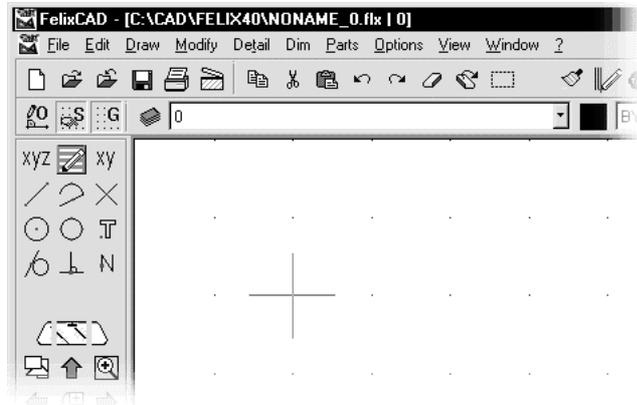
The Snap Angle field allows you to change the angle of the snap. To rotate the snap grid counterclockwise, enter a positive number (usually between 0 and 90 degrees) in the edit box. To rotate the snap grid clockwise, enter a negative value (usually between 0 and -90 degrees).

**Note:** This setting also affect the angle of the grid, hatch patterns, and ortho mode. The snap can also be rotated with the **Z-Direction** option when setting a User Coordinate System (see **UCS** command).

You can also set the snap origin and angle with the **SnapBase** and **SnapAng** system variables.

## Grid Spacing

Turning on the Active checkbox displays an array of dots on the drawing called the *grid*. The grid offers you a visual impression of the distances, proportions, and other relations of the drawing.



The small dots in the drawing area make up the grid display

Often the grid spacing is a multiple of the snap distance. For example, the snap distance is 1 inch while the grid distance is 1 foot. The grid is meant to be a guide; when the grid is dense, the drawing is difficult to see. When the grid is too dense, FelixCAD does not display it.

## Factor

The **Factor** field defines the ration of the grid spacing to the snap spacing. You can select a pre-defined value by clicking one of the buttons.

**Note:** The grid and snap settings become effective after the dialog box is closed.

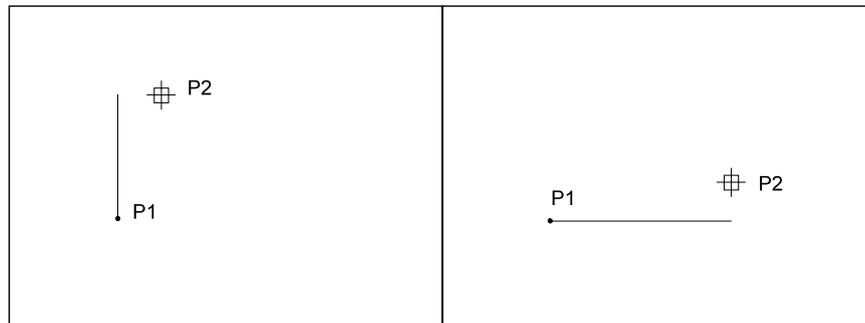
The grid only appears in the area specified by the **LimMin** and **LimMax** system variables. These two variables control the area of the drawing covered by the grid, when turned on.

## Orthogonal Mode

Orthogonal mode is useful while drawing or modifying entities: it forces the cursor to move in horizontal (along the X-axis) and vertical (Y-axis) directions.

You can rotate ortho mode by specifying an angle via the **Snap Angle** field of the **Snap Grid** area in this dialog box.

**Note:** Ortho mode is effective only when picking points using the pointing device. You override ortho mode when you type coordinates at the keyboard.



Ortho 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.

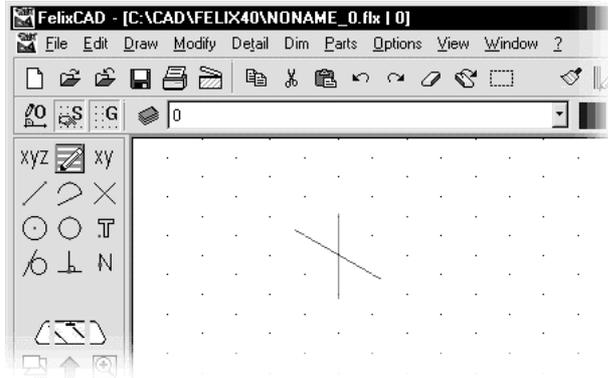
When you are modifying objects, ortho mode means that points for scrolling, scaling, rotating, and so forth, are exactly parallel or perpendicular to the axes of the defined base point.

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

## Isometric Snap/Grid

FelixCAD allows you to draw in isometric mode, which produces the illusion of 3D. The isometric snap style assists you in isometric drawing. In addition, isometric mode skews the shape of the cursor (see figure).

The lines aligned to the angled, isometric axes give the impression that you are looking at front, top, and side views of an object. Rectangular objects, for example, are usually drawn as parallelograms with 30, 90, or 150-degree angles.



The isometric grid

The **Left**, **Top**, and **Right** radio buttons determines the current isometric plane:



- **Left:** isometric crosshairs at 90° and 150°
- **Top:** isometric crosshairs at 30° and 150°
- **Right:** isometric crosshairs at 90° and 30°

**Note:** You must turn on the grid to see isometric grid.

The isometric snap grid has three major axes. By default (as long as no rotation of the snap grid is applied), the axes of the isometric style are 90° (the vertical axis), 30° and 150°.

The cursor moves in only two of the three axes at a time. Each of the three isoplanes has an associated pair of axes.

## OBJECT SNAP MODES

For precise drafting, many commands require that you select an exact position in the drawing. You can pick the precise location by typing the coordinates, or by picking a point in conjunction with an *object snap*. Object snap allows you to pick geometrically significant points on existing entities.

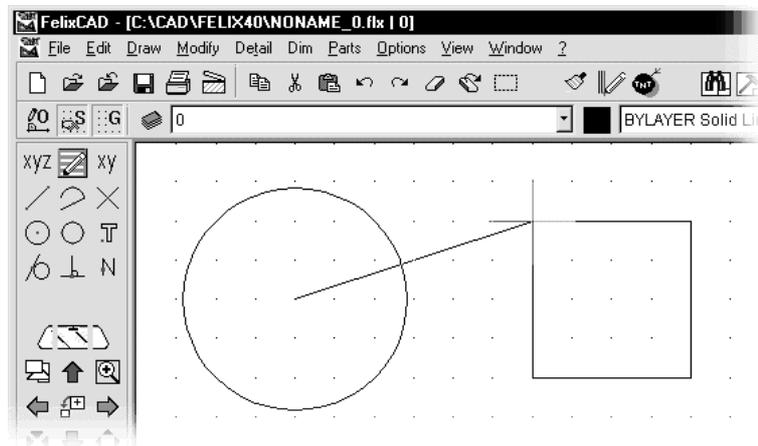
Object snap mode can be activated permanently or called for a single pick. The **OSnap** command lets you turn on object snap modes; transparent commands – such as **mid** and **qua** -- temporarily activate an object snap mode.

The procedure for using the object snaps is always the same:

1. Start a command that requires a point, such as “From point:”.
2. When the point is requested, reply with an object snap mode; the command prompts for the point with “of”.
3. Select the object with the cursor.

Here is an example of using object snap. A line is drawn from the center (**cen**) of the circle to the corner (**int**) of the rectangle:

```
> line
From point: cen
of [pick]
To point: int
of [pick]
To point: [Enter]
```



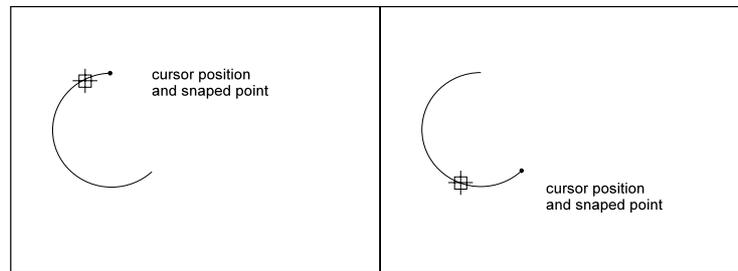
Drawing a line using object snap modes

Some object snap modes make no sense. For example, you can not define the end point of a circle (use the **qua** mode instead), nor can you pick the center of a line segment (use the **mid** mode instead).

On the other hand, you can use several osnap modes on entities. For example, a line can use the **end** mode (twice, one for each end), **mid** mode, and **per** (perpendicular) mode. An arc can use **end** mode (again, twice), **mid**, **cen**, **per**, **qua** (quadrant points) and **tan** (tangent) modes.

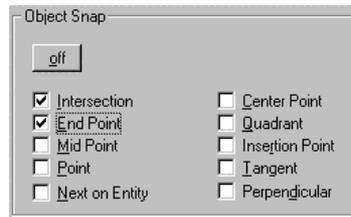
The **end** object snap mode does not differentiate between starting or ending point of an entity; you must be careful with your pick point. The object snap selects the nearest of the two possible points.

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



## Running Object Snap

Using the **Precision Aids** dialog box, object snap modes are turned on. This is called *running mode*.



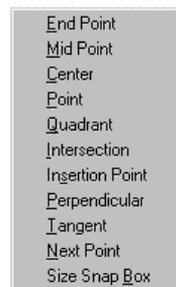
Click the checkboxes for the desired mode(s). A check mark indicates the mode is active. Clicking the **off** button deactivates *all* running snap modes.

When several object snap modes are on, you differentiate between them by the way you select an entity. For example, **End Point** and **Intersection** are on; pick a point nearest an end point, then the end is selected. If you pick a point near the intersection of two entities, then the intersection is selected. Properly configured, this feature can speed the drawing of entities significantly.

## Temporary Object Snap

You can choose between several options for a one-time activation of an object snap mode. You can use the **Snap Palette** to activate an object snap mode just one time during a command.

An effective method of calling an object snap mode is to bring up the cursor menu. Press the middle mouse button (or hold down the **Ctrl** key and press the left mouse button on a two-button mouse). The following menu is displayed, from which you select an object snap mode:



Use the keyboard to type a command abbreviation. The object snap modes and their abbreviations are shown in this table:

Object Snap Mode	Abbreviation
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

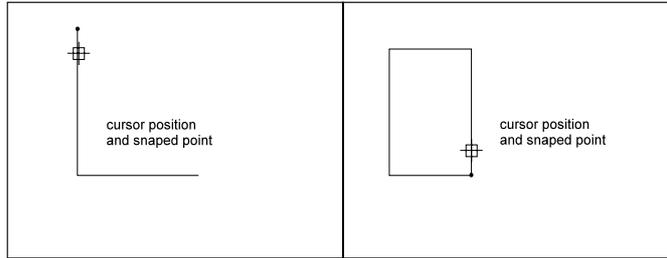
### Object Snap Modes in Detail

There is a difference between object selection and object snap modes. Object selection treats a group of entities, such as a rectangle, chain, and polyline, as a single object. In contrast, object snap modes treat these as individual entities. For example, you can snap to each segment and vertex of a polyline. This is also true for polygons and 2D and 3D faces.; the edges are interpreted as by object snap as individual lines In this way, the object snap modes can be used for the edges of planes as well as for segments of chains and polylines.

For simplicity, when this text refers, for example, to the “end point of a line” the term also refers to polyline segments and the edge of a plane.

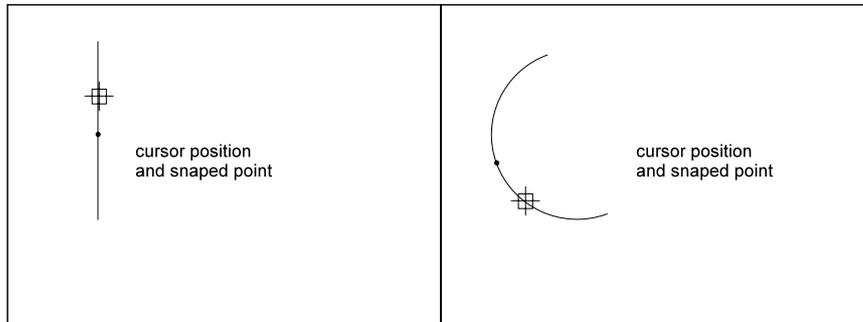
**ENDpoint** 

The **End** object snap snaps to the endpoint of a line or of an arc. The endpoints of planes and faces can be accessed.



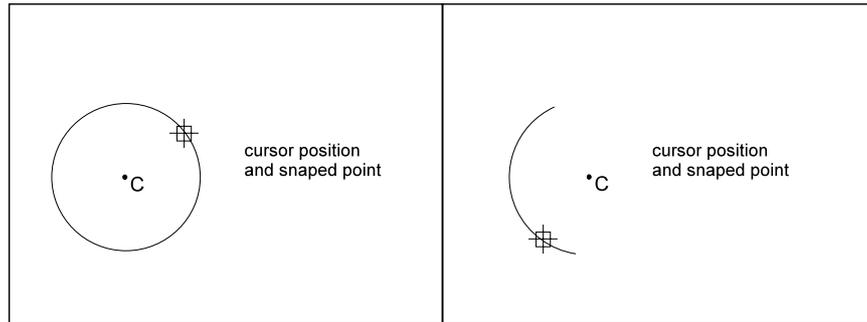
**MIDpoint** 

The **Mid** mode snaps to the center of a line or an arc. The edges of faces may also be snapped.



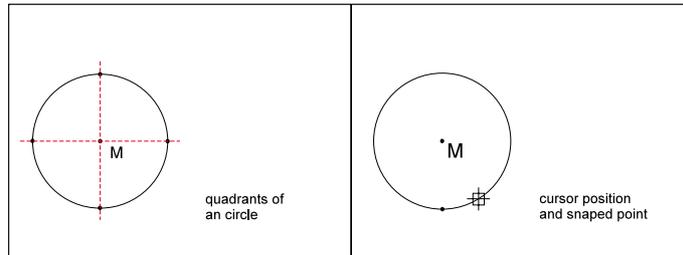
**CENterpoint** 

The **Cen** mode snaps to the center point of a circle, arc, and polyarc.



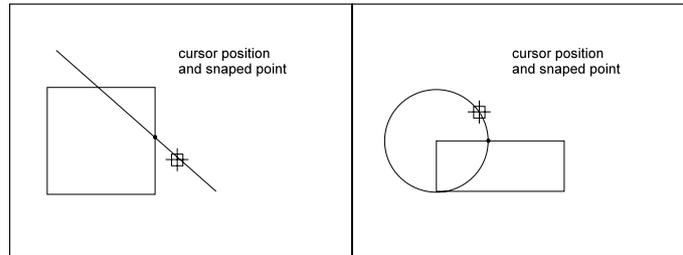
**QUA**drant 

The **Qua** mode snaps to the nearest quadrant of a circle or an arc. *Quadrant* refers to the points situated on the circumference at 0, 90, 180 and 270 degrees. Only the next visible quadrant of an arc can be snapped.



**INTersection**

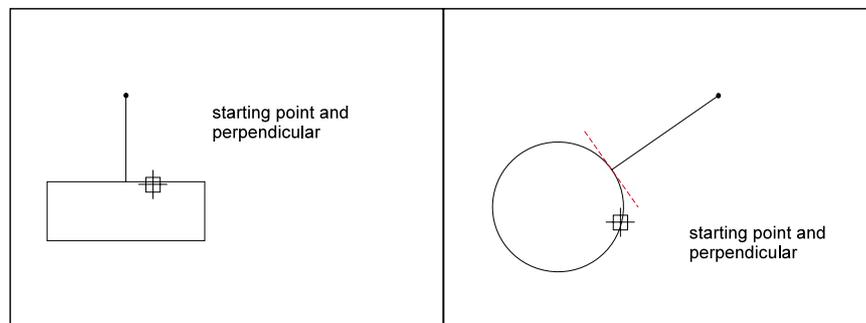
The **Int** mode snaps to the intersections of a pair of lines, arcs, and other entities -- or any other combination of them. This includes dashed and centerlines where the intersection point may not have line segments actually touching.

**IMPIed**

The **Int** object snap mode only returns an intersection point of entities that have a real intersection. In contrast, the **Imp** mode snaps -- in addition -- to an *implied* (intersection point. This is the point where the extension of two entities would intersect.

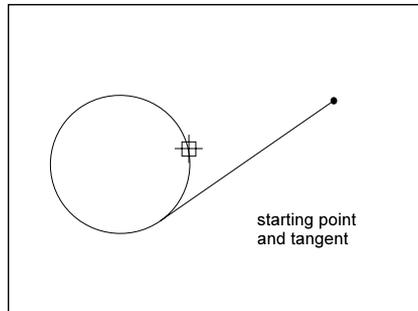
**PERpendicular**

The **Per** mode snaps to the perpendicular to an entity. This mode requires that a starting point already been selected. For example, when a perpendicular is desired, the *ending* point for the perpendicular is picked. The result is a perpendicular line from the ending point to the chosen entity.



## TANgent

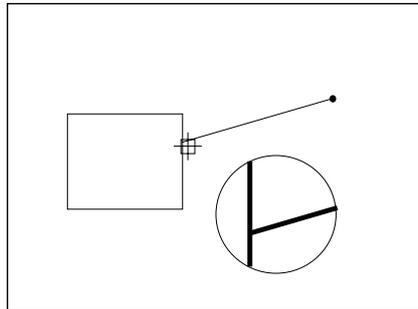
The **Tan** mode snaps to the tangent point of a curve. This mode requires that a starting point has already been selected. For example, when a tangent to a circle is desired, the *ending* point for the tangent line is picked. The result is a tangent line from the ending point to the chosen entity.



## NeXT

The **Nxt** 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 that are designed or modified by use of the object snap mode share a point with the selected object.

(If you select a point in 3D space without using this object snap mode, 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** 

The **Ins** mode snaps to the insertion point of a part, an attribute, or a text object.

**PoiNT** 

The **Pnt** mode snaps to a point entity, which was created by the **Point** command.



## Viewing Drawings

Unlike a paper drawing, using a CAD program involves a trade-off between seeing all of the drawing (with little detail) or seeing a detail portion (of a small part of the drawing).

Also, unlike a paper drawing, a CAD program allows you to view the drawing in three dimensions, and from an infinite variety of 3D viewpoints.

In this chapter, you learn how to:

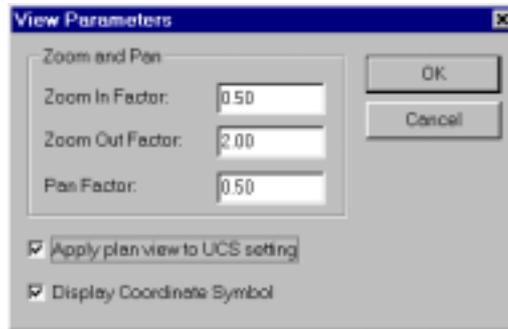
- Work with windows, such as creating new views and switching windows.
- View the drawing from different 3D directions.
- Enlarging and reducing the view, as well as moving the visible drawing portion.
- Create and use a coordinate system.

FelixCAD has commands, such as **Zoom**, that change the viewing size of the drawing, which enlarge and or reduces the visible portion of the drawing. Another command, **Pan**, keeps the size of the drawing constant, but moves the drawing around in the window.

Other commands change the viewpoint, so that you can see the left side or top view of a 3D drawing.

## SETTING VIEW PARAMETERS

You can preset parameters for the **Zoom** and **Pan** commands. The **ViewPar** command (**View > View Parameters**) displays the View Parameters dialog box.



### Zoom and Pan

The **Zoom In Factor** and **Zoom Out Factor** fields lets you specify how far FelixCAD zooms with the **ZoomIn** and **ZoomOut** commands.

The **Pan Factor** field lets you specify how far the drawing moves (called *panning*) with the **PanUp**, **PanDown**, **PanLeft**, and **PanRight** commands. A pan factor of 0.5, for example, moves the drawing by about half of the visible drawing area.

### Drawing Viewports

FelixCAD is capable of opening as many as four drawings at the one time. Each drawing is viewed in its own window. Further, you can open as many as four viewing windows (*viewports*) of each drawing. Viewports allow you to see the drawing from different angles, or see different details – at the same time.

A number, ranging from 0 to 3 identifies the windows. This number appears immediately behind the filename on the title bar, separated by a vertical line, as in **Filename | 2**.

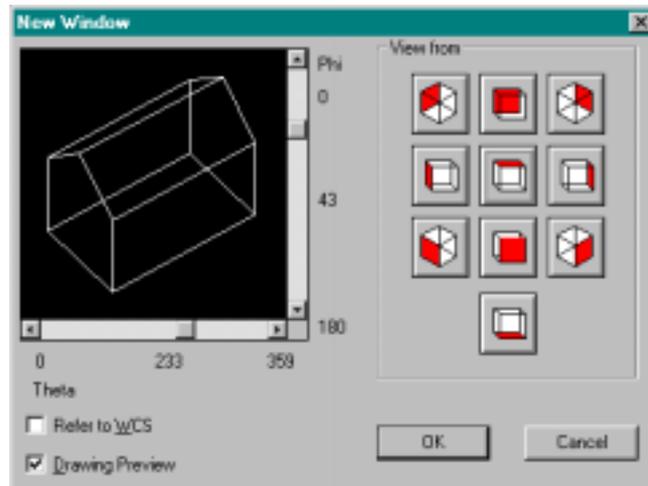
The windows can be arranged in tiles, cascades, or enlarged to fill the entire drawing area.

When more than one window exists, only one window is the *active* window. The active window is identified by the color of the title bar. You pick another window to make it the active window; click on any part of the window with the

cursor. FelixCAD's standard viewing angle is the top view (also called the *plan view*). At this angle, the X, Y-plane is viewed straight down the Z-axis. The drawing or *construction plane* is parallel to the screen.

### Creating a New Viewport

Once a drawing is open, you create additional viewports with the WOpen command (**Window > Open New Viewport**). Before the program opens a new window, the **New Window** dialog box asks for the view direction. *The AutoCAD equivalent to this command is the MVIEW command.*



Select a standard view from the **View from** buttons, or specify a custom view with the **Theta** and **Phi** sliders. (The view you select can be changed at any time.) Click on **OK**. The new window is created in standard size and arranged in the manner proscribed with the existing windows (tiled, cascaded, etc.).

#### Theta / Phi

The preview window, showing a house, lets you interactively change the 3D-view direction. Move the **Theta** and **Phi** sliders and watch the house rotate in three dimensions. When satisfied with the viewpoint, click **OK**.

#### View From

Select a standard view from the **View From** buttons.

## Plan View

To switch quickly to a plan view, use the **PlanView** command (**View > Plan View to Construction Plan**).

## Changing the View

The **3dView** command (**View > 3D View**) displays the **View Direction** dialog box, which is the same as the **New Window** dialog box discussed above. It 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.

To change the 3D viewpoint at the command line, such as in macros or FLISP routines, use the **SetViewDir** command.

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

The **Rotate** option allows you to specify a new view direction, based on two angles entered as real numbers, as shown by the following prompts:

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

The **Angle theta** is measured from the X-axis in the XY-plane. The **Angle phi** is specified up and down from the XY-plane.

**Note:** The **SetViewDir** command cannot be used in paper space.

## Changing the Active Window

Once you have more than one window open, there are two ways you can change the active window:

1. Click any part of a window to make it the active window.
2. Click the **Window** menu, which lists the names of all open windows. The active window is marked by a checkmark in front of its name. 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 Window

The **QWOpen** command (**Window > Copy Viewport**) opens a new window and allows you to make a quick copy of the active window (**Copy** option) or of the plan view (**Plan** option).

```
> qwopen  
New Window: Copy
```

## Closing a Window

The **WClose** command (**Copy > Close Current Viewport**) closes the current viewport. Or, you may use the standard Windows methods of closing a window: type **Ctrl+F4** or click the **x** in the upper right corner of the 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

**Transparent Zoom**

The **Zoom** command, like most other view commands, is *transparent* commands that can be used at the same time another command is being executed. For example, you can use the **ZoomOut** command during the **Line** command. Remember that transparent commands need an apostrophe before the command name, such as '**zoomwin**'.

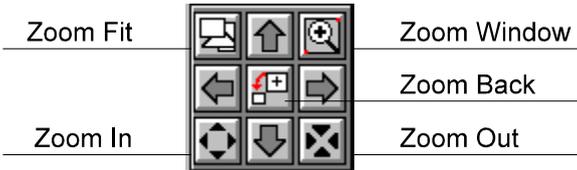
```

> LINE
From point: P1
To point: P2
To point: 'ZOOMOUT'
To point: P3
To point: [Enter]

```

**The Zoom Palette**

You can select the zoom commands from the palette. The symbols used to activate the zoom options are:



To activate a zoom command, click on its symbol. If you prefer to type commands at the keyboard, the zoom variations are found in the following table.

Zoom Options	Command
Zoom window	ZoomWin
Zoom to previous view	ZoomBack
Zoom to fit	ZoomFit
Zoom in	ZoomIn
Zoom out	ZoomOut
Zoom factor	ZoomFac

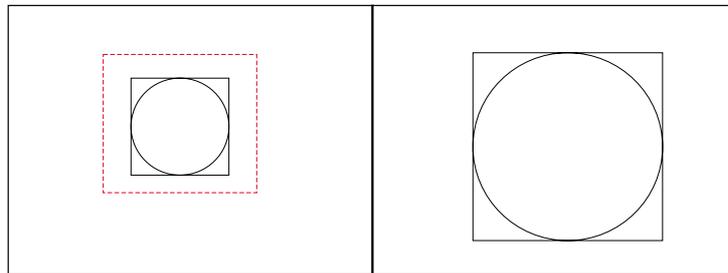
### Zoom Window

The **ZoomWin** command allows you to zoom into a rectangular area that you specify by opposite corner points with the cursor. As an alternative, you can type coordinates.

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

There may be a difference between the window you chose, and the view you see on the screen. This is due to different ratios of height and width of your chosen window and the screen.

The following figure shows the usage and result of the **Zoom Window** command.

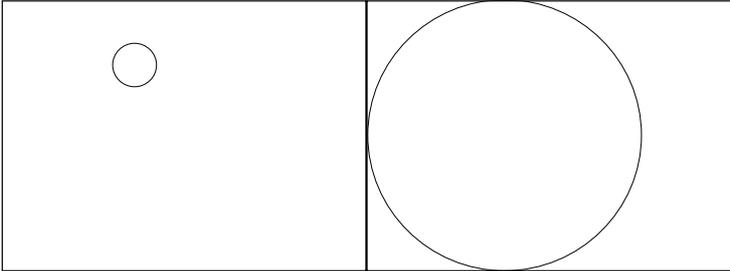


### Zoom Back

The **ZoomBack** command is like using the **Undo** command: it backs up to the previous view. Using this command allows you to switch quickly back and forth between two views to see the drawing as a whole, or to see its details more clearly.

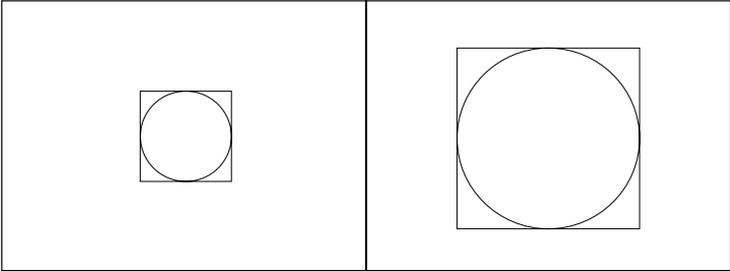
**Zoom to Fit**

The **ZoomFit** command allows you to see the entire drawing as large as possible in the window.



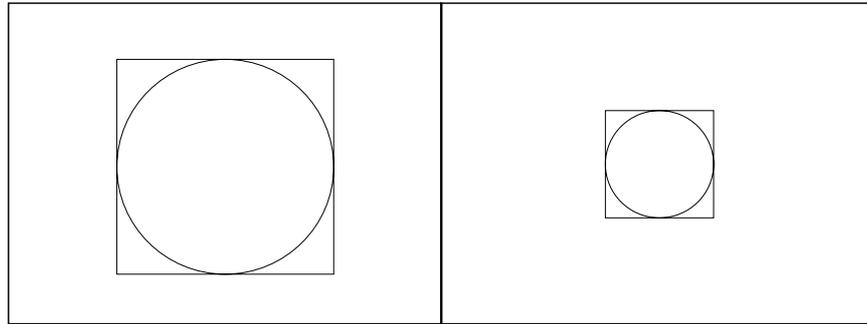
**Zoom In**

The **ZoomIn** command enlarges the view with each successive use, by a factor defined by the **View > View Parameters** command. By default, this is 0.5.



## Zoom Out

The **ZoomOut** command is the opposite of the **ZoomIn** command. It reduces the view with each successive use, by a factor defined by the **View > View Parameter** command. The default is 2.0.



## Zoom Factor

The **ZoomFac** command prompts you to enter the zoom factor. Numbers less than 1.0 reduce the size (equivalent to **ZoomOut**) and greater than 1.0 enlarge the size (like **ZoomIn**).

```
> zoomfac
Zoom factor: 1.2
```

## Zoom Page

The **ZoomPage** command zooms out to the page size defined for your printer (or plotter).

## PAN

The **Pan** command moves the visible portion of the drawing. The drawing can be moved to the right, to the left, upward, or downward. You now see objects that were previously not visible because they were off the viewing surface. In contrast to the **Zoom** command, the view scale does not change.

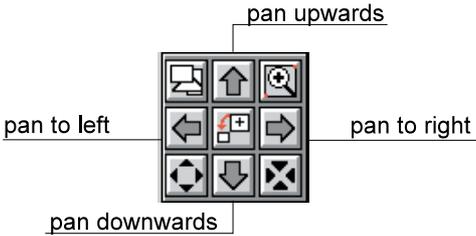
The **Pan** command, like most other view commands, is a *transparent* command that can be used at the same time another command is being executed. For example, you can use the **PanDown** command during the **Line** command. Remember that transparent commands need an apostrophe before the command name, such as **pandown**.

```
> LINE
From point: P1
To point: P2
To point: 'PANDOWN
To point: P3
To point: [Enter]
```

You can think of the **Pan** command like holding a camera. Think of being the photographer looking through the camera’s viewfinder, and moving the camera left and right, up and down.

**The Pan Palette**

You can select the pan commands from the palette. The symbols used to activate the pan options are:



To activate a panning command, simply click its symbol.

If you prefer to type commands at the keyboard, the pan variations are found in the following table.

Pan Options	Command
Move view to the left	PanLeft
Move view to the right	PanRight
Move the view down	PanDown
Move the view up	PanUp

After selecting one of these commands, the visible drawing portion is immediately moved in that direction. The amount of movement is determined by a factor defined by the **View > View Parameters** command. By default, this is 0.5.

## DYNAMIC ZOOM AND PAN

Two commands allow you to zoom and pan directly, compared to the commands described above.

### DZoom

The **DZoom** command lets you zoom in and out in *real-time* -- nearly seamlessly, in one operation (in the current drawing window). Dynamic zoom can be used transparently (in the middle of another command).

Select **View > Dynamic Zoom**. Notice that the cursor changes to a magnifying glass with a plus and minus sign. To start the dynamic zoom process, drag (click and hold the left mouse button) up and down. To zoom in, drag up (in the direction of the plus sign). To zoom out, drag down (in the direction of the minus sign).

When you reach the edge of the drawing window, release the mouse button, move the cursor away from the edge, and drag again.

To exit dynamic zoom mode, press **ESC** or **ENTER**. You can also right-click to open the option menu, which contains the **Exit** option.

Notice that the cursor menu also contains options for **Dynamic Pan** and **Zoom Extents**.

### Dpan

The **DPan** command lets you pan around in *real-time* -- nearly seamlessly, in one operation (in the current drawing window). Dynamic pan can be used transparently (in the middle of another command).

Select **View > Dynamic Pan**. Notice that the cursor changes to a hand. To start the dynamic pan process, drag (click and hold the left mouse button) around in any direction.

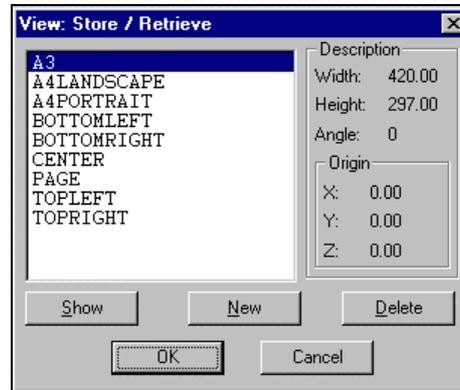
When you reach the edge of the drawing window, release the mouse button, move the cursor away from the edge, and drag again.

To exit dynamic pan mode, press **ESC** or **ENTER**. You can also right-click to open the option menu, which contains the **Exit** option.

Notice that the cursor menu also contains options for **Dynamic Zoom** and **Zoom Extents**.

## SAVING VIEWS

When working with many drawing portions and view directions, it can be convenient to save (and later to retrieve) the viewpoint. You do this with the **View** command (**View > View Manager**).



### Saving the Current View

First, use the zoom and pan commands described previously to set up a view direction. To save the view, start the **View** command. Then:

1. In the **Save/Retrieve View** dialog box, select the **New** option.
2. The **View** option save the current view; the **Window** option prompts you to select a rectangular window as the view.
3. Enter a name for the view. Later, you use the name to retrieve the view.
4. Click **OK**.

During the drafting process, named views can be applied in two situations:

- **Printing:** the **Print** command has an option for printing a named view. *Hint: this is useful in printing different 'sheets' (i.e. 'sheet1', 'sheet2') that may exist in one drawing.*
- **Paperspace:** when calling the command **View** in layout mode outside the **Viewport** command, you can create named views within paper space (for example to zoom fast to portions of the drawing sheet).

### Restore a Named View

A previously saved view is retrieved with the View command. Follow these steps:

1. In the **Save/Retrieve View** dialog box, choose one of the saved views from the list box.
2. Click the **Show** button. The saved view then becomes the present view.

### Delete a Named View

When you no longer need a named view, you can delete as follows:

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

## USER COORDINATE SYSTEMS

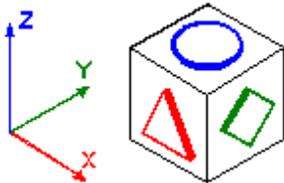
A *user coordinate system* is a coordinate system that you define. It is primarily used for 3D drafting. Creating your own UCS allows you to draw in “plan view” at any angle in space.

Until you create a UCS, you are drafting in the *world coordinate system*. Via a UCS, you can:

- Move the origin (0,0,0) to any point in the world coordinate system.
- Turn one or more axes relative to the world coordinate system.
- Move and angle the user coordinate system.

The user coordinate system, for example, makes it possible to change the location of the drawing plane so that it always lies parallel to the objects to be drawn. This makes it easier to construct 3D objects.

To determine the coordinates of a point, distances are measured from the point to the X- and Y-axis; in a 3-D system, measurements are also taken along the Z-axis. FelixCAD differentiates between a world coordinate system (WCS) and the user coordinate system (UCS).



**World Coordinate System**

The world coordinate system is the permanent Cartesian coordinate system 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. You cannot change the world coordinate system.

Instead, it is possible to create many user-defined coordinate systems for a drawing. The user coordinate system is saved along with the drawing and can be used again later.

**Creating a New UCS**

A UCS is created in two steps. First, you specify the origin and direction of the axes for the new UCS. Second, you have the UCS by name, so that you can reuse the UCS without having to reenter its parameters.

The **UCS** command creates a new user coordinate system. At the “UCS origin <0,0,0>” prompt, enter the coordinates for the origin of the new user coordinate system. Or, press **Enter** to use 0,0,0.

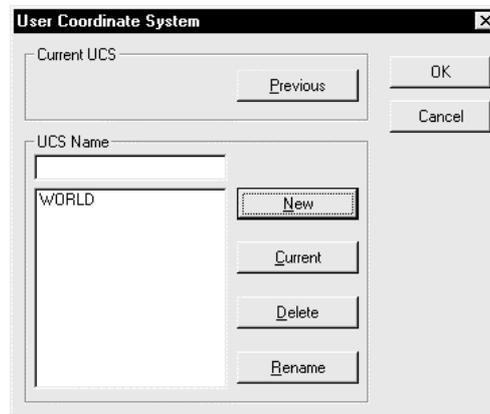
```
> ucs
UCS origin <0,0,0>:
```

Or, use the following options to interactively determine the origin of the user coordinate system and the direction of its axis:

Control	World	Z-Direction	3Point	View	Rotation
---------	-------	-------------	--------	------	----------

## Control

The **Control** option displays the User Coordinate System dialog box to select an existing UCS. The dialog box also lets you create a new UCS, rename it, or even delete it.



## World

The **World** option changes the UCS back to the world coordinate system.

## Z-Direction

The **Z-Direction** option changes the parameters of the coordinate system in two steps:

1. You determine the origin by entering the new coordinates (in X,Y,Z format) or by picking a point .
2. The direction of the axes of the user coordinate system is determined by fixing the z-axis. To do that, choose a point through which the positive Z-axis should go. The X- and Y-axis remain the same. The XY-plane is orthogonal to the Z-axis.

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

## 3Point

The **3Point** option is meant for drawing planes that are neither parallel to the viewing plane nor to the axis of active coordinate system. We recommend you use the cursor to pick the coordinates:

1. You determine the origin by entering the new coordinates (in X,Y,Z format) or by picking a point .
2. Pick a point (or type coordinates) through which the positive X-axis should run. This determines the axis unambiguously.
3. The direction of the other axes is determined by fixing 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 axes of the UCS so they are oriented with the plane of the drawing object. For example, choose the corner points of this plane with object snap. The selected plane becomes the drawing plane. All entities are drawn on this plane.

```
> ucs
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

The **View** option orients the user coordinate system parallel to the active view. The UCS command does not ask you for any coordinates. The X-axis of the coordinate system lies parallel to the horizontal axis of the viewing plane; the Y-axis is oriented vertically; and the z-axis is orthogonal to it

```
> ucs
UCS origin <0,0,0>: View
```

## Rotation

The **Rotation** option rotates the active coordinate system around one axis. You may choose the angle of rotation and the direction of the rotation.

```
> ucs
UCS origin <0,0,0>: Rotate
Rotate around x-, y-, or z-axis: Z
Rotation around Z-Axis <0>: 30
```

To rotate the coordinate system around two or three axes, repeat these steps for each axis:

1. Determine the axis around which to rotate the coordinate system. Either type its character or click its option (X, Y, or Z) from the option bar.

2. Determine the direction and angle of the rotation. Entering or pointing an angle does this.

You specify the angle of the rotation by typing the angle or by picking two points in the drawing. 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.

### Controlling the UCS

Of course, it would not be very effective if you had to make the above adjustments every time you needed to change the coordinate system. Therefore, FelixCAD allows you to save pre-defined user coordinate systems by name, which are saved with the drawing. Once a UCS is saved, you can always reuse it; the UCS names can also be renamed or deleted.

All this is possible via the **UCS** command's **Control** option. Or, from the menu bar, select **Options > User Coordinate System > UCS Manager**.

### Creating a New UCS

You create a new user coordinate system in these steps:

1. Create the UCS with an option as described above: **Z-Direction**, **3Point**, **View**, or **Rotate**.
2. Open the dialog box and type a name in the **UCS Name** field. The name can be up to 31 characters long, as well as dash ( - ) and underscore ( \_ ), but no spaces.
3. Click **New**.
4. Click **OK**.

### Renaming a UCS

If you need to rename a UCS:

1. Select a UCS name from **UCS Name** list.
2. Click the name and type a the new name.
3. Click **Rename**. This verifies the renaming; the new name of the UCS appears in the field.
4. Click **OK**.

**Note:** the world coordinate system cannot be renamed.

### Deleting a UCS

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

1. Select a UCS name from the **UCS Name** list.
2. Click **Delete**. The UCS is then deleted; its name no longer appears in the list.
3. Click **OK**.

**Note:** the world coordinate system cannot be deleted.

### Changing the Active UCS

You can create as many user coordinate systems as you need in a drawing, but only one user coordinate system can be the *active* coordinate system. To change the active UCS:

1. Select a UCS name from the **UCS Name** list.
2. Click **Active**. The user coordinate system becomes the active coordinate system. The field shows the name of the active coordinate system.
3. Click **OK**.

### Reverting to the Previous Coordinate System

To change to a previous coordinate system, click the **Previous** button. It enables you to quickly change between two coordinate systems. When you click this button, FelixCAD changes to the previous coordinate system without verification.

# Layers and Object Properties

The concept of working with *layers* in FelixCAD is analogous to manual drafting using layers of transparent Mylar. Parts of the drawing are drawn on different sheets of Mylar, such as the base plan, the electrical, the plumbing, the roadwork – each on its own Mylar sheet. When all sheets are imposed on top of one another, they form the entire drawing.

Layers are useful in separating objects, such as parts, sub-assemblies, and structural components like pipes and building materials. In CAD drawings, layers separate drawing elements, such as construction lines, center lines, outlines, and dimension lines. The advantages of creating a drawing in this manner – whether manually or digitally – lie in a clearer drawing and improved control over drawing on a single sheet.

This chapter describes how to create and use layers, as well as ways of controlling layers. *Control* refers to the visibility of the layers, their color, and the type of lines used. Management of layers is accomplished through the Layer Manager.

In the second half of this chapter you find explanations on object properties and the advantages of working with them. An *object* may be a part, a sub-assembly, a group of similar entities (such as parking spaces in an architectural drawing), or any other entity that you wish to separate from other entities in the drawing.

## **LAYER BASICS**

Instead of working with sheets of Mylar, as you might when drafting by hand, CAD software works with *layers*, which have the characteristics similar to Mylar. Layers make it possible to structure a drawing through logical or content characteristics.

Drawing layers always lie exactly on top of each other. All aspects of a drawing, such as the coordinate system, the zoom factor, the view arrangement, and fonts, are valid for all layers. Other aspects may differ for each layer, such as the color and linetype of entities drawn on a particular layer. In addition, layers can be “turned off,” which means that all entities on that layer become invisible.

In CAD software, a new object is always drawn on the *current* layer. When the object is drawn, the layer's properties (specifically, linetype, visibility, and color) apply to the object. FelixCAD has commands that let you change the properties of all objects on a layer, or of each object independent of the layer's global setting.

Layers are identified by name. All drawings contain at least one layer, given the name "0" (pronounced "layer zero"). All other layers can have their own number or name. Layer names help describe the purpose of the layer and can be up to 31 characters long.

You can create up to 4,095 layers in a FelixCAD drawing, a number that should be more than enough for any practical drawing. Layers do not necessarily follow one another in some sequential order. It is not necessarily true that layer 2 lies "on top" of layer 1 or "underneath" of layer 3. All layers are arranged in one single plane, which is not possible when drawing manually on different Mylar. When one entity hides other entities on different layers, the rule is that the object last drawn lies over already existing objects and "hides" them.

As mentioned earlier, every drawing contains one default layer name layer **0**. This layer has several special properties that distinguish it from all other layers. It is generated automatically by FelixCAD when a new drawing is created. The color 7 (black) and the linetype Continuous are automatically established for this layer, although you may change the color and linetype. This layer cannot be erased or renamed. Layer 0 has a special property when defining and inserting parts. You can find more on that subject in the "Parts and Attributes" chapter.

One layer in the drawing is always the *current layer*. All new entities are automatically drawn on this layer. Naturally, you change the current layer whenever you need to. As long as layer 0 is the only one existing, it is also the current layer. If there is more than one layer in a drawing, as there should be, you switch between layers using the **Layer Manager** or the **Properties** toolbar.

Although you can only draw entities on a single layer (i.e., the current layer), you edit entities residing on any layer at any time – with one exception: you cannot edit entities on layers that have been *frozen*, *turned off* or *locked* (more later).

The layer names, their status, and their properties are part of the drawing. They are saved with the drawing and can be shared with other drawings via the Drawing Explorer.

## **LAYER PROPERTIES**

Layers have four properties: name, color, linetype, and visibility setting.

### **Layer Name**

To identify layers and differentiate between them, each layer has a distinctive name. You can assign any name you want up to 31 characters long. The name can consist of letters, numbers, and two special characters: \_ (underline) and \$ (dollar). All characters in the name are automatically capitalized.

If you are not sure what layer names to use, you can check with the regulating body for your discipline. Some organizations have Standing Committees on the subject and have layer guidelines. One of these is the International Standards Organization (Standards ISO 13567-1 & -2:1998, ISO/DTR 13567-3:1998)

<http://www.iso.ch/isob/switch-engine-cate.pl?searchtype=general&KEYWORDS=CAD>

Another is the American Institute of Architects for the AEC industry. Their WEB site for purchasing the AIA standard is

<http://www.aiabooks.com/detail.asp?productid=R809-97&search=layer&position=1>. In some cases, your client or your office may have a layer naming specification. For informal drafting, you can create a few layers with self-explanatory names like:

- **Existing** for the pre-construction floorplan.
- **Constr** for the planned construction.
- **Text** for text.
- **Dim** for dimensions.

### **Color**

Every layer has a color; the default color is **black**. You can choose one of 255 color shades. Every color has its own number. The standard color of the layer can be marked in the color box as BYLAYER.

Since it is possible to draw on the current layer only, objects still to be drawn are in the standard color of the current layer, unless a new color is chosen with the **Object Color** option. 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 assumes the new color.

### Linetype

Similarly, every layer has a linetype; the default is Continuous (solid). 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 assumes the new linetype.

While every drawing contains all 255 colors, a new drawing contains only the Continuous linetype. Before you can assign other linetypes to a layer, you must load the linetype definitions into the drawing.

#### How to Load Linetypes

Linetypes vary by discipline. The linetypes used for an electrical drawing differ from a civil engineering drawing. For this reason, FelixCAD does not store with a fixed set of linetypes, but allows you to load linetype files specific to a particular discipline.

Linetype files have the extension *.lin* and can be found in the *\applic* folder. FelixCAD ships with two linetype files: **Mm.Lin** for metric drawings and **Inch.Lin** for imperial drawings. Your office or your client may provide you with project-specific linetype files. In addition, you may use most existing AutoCAD *.lin* files or create customized linetypes by following the syntax of the *.lin* files.

To load a linetype file:

1. Select **Linetypes** from the **Options** menu, or type the **Linetype** command. FelixCAD opens the **Open Linetype File** dialog box.
2. Type the name of the linetype file in the **Filename** input field or choose a *.lin* file from the file list.
3. Next, choose one (or more or all) linetypes from the list in the **Linetype** dialog box. The preview area at the top of the dialog box assists you to find the appropriate linetype.



4. After you choose and load the linetype(s), you assign them to a layer by using the **Layer Manager**. Or, they may be associated with an existing drawing objects with the **PropEdit** command

## Layer Visibility

One of the primary advantages of working with layers is that they aid you in visualizing the part, or drawing. The visual clarity of a drawing can be improved by turning off the visibility of some layers; this “hides” the entities residing on those layers.

Every layer can be turned on or off independent of all other layers. Changing the visibility is done through the **Layer Manager (Layer command)**.

### On / Off Status

When a layer is turned *off*, you no longer see the entities drawn on that layer. New layers are created with the status **on**.

### Locked / Unlocked Status

It is possible to *lock* layers so that entities on the layer are visible but may not be edited. New layers are created with the status **unlocked**.

When a layer’s status is **locked**, it is not possible to change, add, or delete entities assigned to this layer. This function makes it difficult to make an accidental change to the drawing. The status must be changed to **unlocked** if you want to work with the entities of this layer.

You can draw new entities on a locked layer. The standard color of the layer may be changed. A locked layer may be designated active, visible or invisible, frozen or thawed.

## Thaw / Freeze Status

The **Thaw** and **Freeze** status controls visibility in a manner somewhat differently from **On** and **Off**.

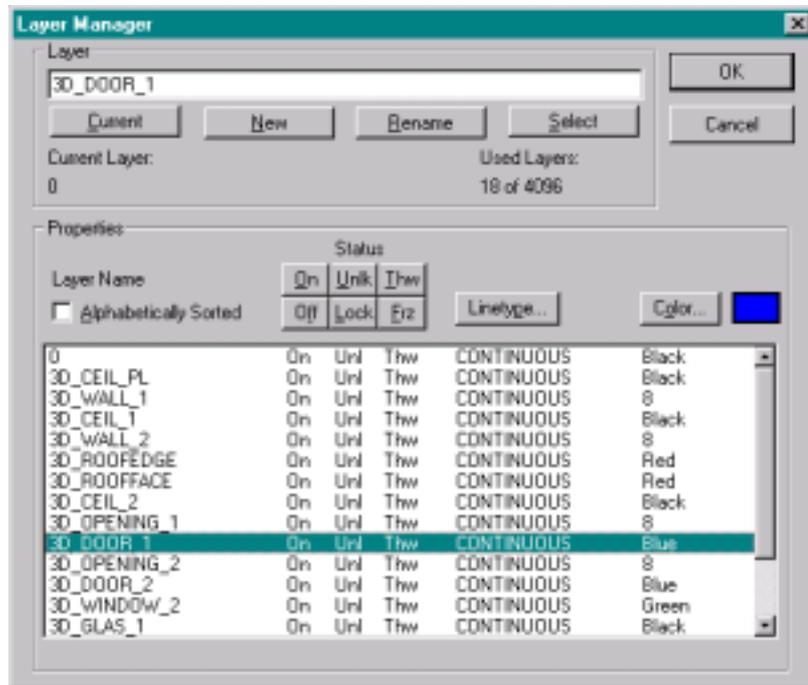
A *frozen* layer is both invisible and locked and thus impossible to change. Entities on a frozen layer are regenerated. When working with complex drawings, freezing unneeded layers helps increase the speed of drawing considerably.

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

**Note:** The current layer cannot be frozen.

## LAYER MANAGER

Creating new layers, choosing a current layer, and controlling layer properties is accomplished through a dialog box called the **Layer Manager**. You access the dialog box with the **Layer** command (**View > Layer Manager**).



The top section of the **Layer Manager** dialog box contains a field for the name of the current (active) layer and a report of the number of layers already defined in this drawing. In a new drawing, this reads:

```
Current Layer:      Used Layers:
0                  1 of 4096
```

Buttons allow you to create new layers, rename layers, and select the current layer. Other buttons change the state of the layer and select linetype and color.

The **Alphabetically Sorted** checkbox lists layers in alphabetical order (when on) or in the order they were created (when off, no check mark).

The lower section of the dialog box lists the names of all the existing layers in the drawing. The status, color, and linetype are also shown for each layer. Select a layer with your cursor.

The meanings of the abbreviations used are:

Abbreviation	Status
On	On: entities on the layer are visible.
Off	Off: entities on the layer are invisible.
Unlk	Unlocked: entities can be edited.
Lock	Locked: entities cannot be edited.
Thw	Thawed: entities are visible and can be edited.
Frz	Frozen: entities are invisible, cannot be edited, and are not regenerated.

### Creating a New Layer

You can create a new layer at any time. You do not have to draw entities right away on a newly created layer. Remember that you can only draw on the current layer.

A new layer is created in two steps:

1. Type the name for the new layer in text entry box at the top of the dialog box. Remember the limits of naming (see earlier in this chapter).
2. Click the **New** button. Notice that the layer appears in the list. Every new layer has color 7 (black) and the linetype Continuous. You may assign other linetypes and colors, as described later in this chapter.

### Renaming a Layer

A layer is given a different name in three steps:

1. Select the layer from the list of layer names.;
2. Notice that the name appears now in the text entry box. Type the new name over the old name.
3. Finish renaming by clicking the **Rename** button. Notice that the renamed layer appears in the list.

### Selecting the Current Layer

You select the current (working or active) layer in two ways:

1. Click on the **Layer Manager** Icon and from the dialog box, select the layer you want to become current. Finally, click the **Current** button.
2. Click the **Layer Drop-Down List**. Once it opens, scroll down and click on the layer you want to become current.

Alternatively, you can type in SETLAYER (See SETLAYER command, later in this chapter).

### Changing the Layer's Linetype

A newly created layer has the linetype Continuous automatically assigned to it. You change the linetype in three steps:

1. Select the layer(s). Hold down the **Shift** key to select a range of layers, or hold down the **Ctrl** key to select a dispersed range of layers.
2. Click the **Linetype** button. Notice that the available linetypes are shown in the **Linetype** dialog box. *Note:* you must first load linetypes into the drawing, as described earlier in this chapter.
3. Select the linetype to be used and click **OK**. Notice that the selected layer(s) have the new linetype assigned to them.

This makes the selected linetype the standard linetype of this layer. All entities on this layer take on that linetype.

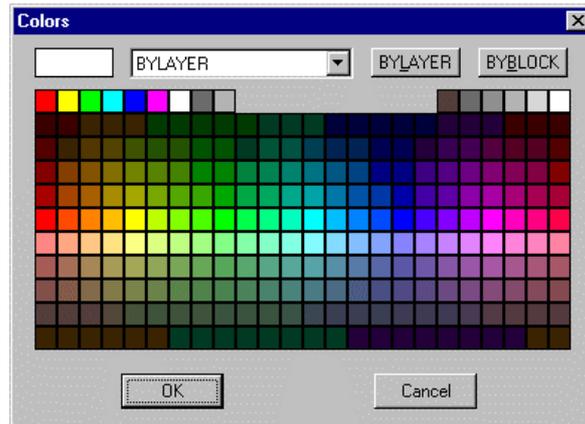
Elements of the layer drawn with a linetype different than the BYLAYER linetype are *not* affected by the change. Further instruction on drawing elements with a linetype different than the standard linetype can be found in the *Object Properties* section in this chapter.

## Changing the Layer's Color

When a new layer is created, it is automatically assigned the color 7 (black). A new color can be assigned to the new layer via the **Color** button. You change the color of a layer in three steps:

1. Select the layer(s). Hold down the **Shift** key to select a range of layers, or hold down the **Ctrl** key to select a dispersed range of layers.
2. Click the **Color** button. The **Colors** dialog box appears.
3. Select another color from the colors displayed, or type a color number (in the range of 1 to 255). The chosen color appears at the upper left corner.

Click **OK** to dismiss the dialog box. Notice that the layers have the new color assigned to them.



This makes the chosen color the standard color for this layer. All entities on this layer now appear in this color.

Entities drawn with a different color are not affected by the change.

## Controlling the Layer Status

The layer properties that determine the status of the layer were discussed earlier in this chapter. Changing the status of a layer is as simple as clicking the proper button. The status of each layer is displayed next to the layer's name.

- The **On** and **Off** buttons control the display visibility of the layer.
- The **Unlk** and **Lock** buttons control access to the layer. **Unlk** allows editing; **Lock** prohibits editing.
- The **Thw** and **Frz** buttons control the ability to see, edit, and regenerate the layer. **Thw** unlocks and makes visible; **Frz** locks and makes invisible.

The layer status is changed in two steps:

1. Select the layer(s). Hold down the **Shift** key to select a range of layers, or hold down the **Ctrl** key to select a dispersed range of layers.
2. Click the button appropriate to the change in status. For example, to freeze a layer, click the **Frz** button.

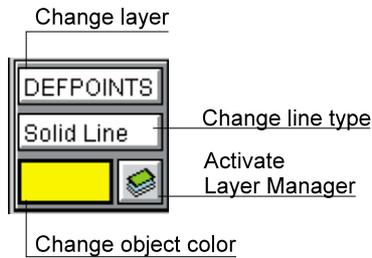
Remember that the current layer *cannot* be frozen.

## QUICK ACCESS TO LAYERS

In addition to the Layer Manager, FelixCAD provides a palette (toolbar) with the three most important layer functions:

- Change the current layer.
- Draw with a linetype other than BYLAYER.
- Draw with a color other than BYLAYER.

To access these functions, select the buttons found on the palette on the left-hand of the desktop.

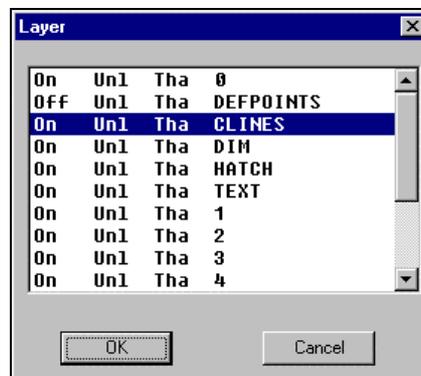


## SETLAYER Command: Set Current Layer

This function allows you to set a new current layer either from the palette (toolbar) or typing SETVAR and pressing the <Return> key.

**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 *Layer Control* section.

Selecting this button opens the **Layer** dialog box, where you can select another current layer.



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

<b>Abbreviation</b>	<b>Meaning</b>
On	On: visible.
Off	Off: invisible.
Unl	Unlocked: full access.
Loc	Locked: no editing access.
Tha	Thawed: not frozen.

A frozen layer cannot become the current layer, so that choice does not appear for the current layer. Select a layer name in the list box and click **OK**.

### **Set Current Linetype**

You may want to override the linetype assigned to the layer (known as the **BYLAYER** linetype).

(To change the linetype of an entity after it has been drawn, use the **PropEdit** command, described later on in this chapter.)

To change the current linetype, click the **Linetype** button in the control panel. The **Layer Manager** dialog box appears, which is the same dialog box as described earlier. Choose a linetype from the list and click **OK**.

### **Set Current Color**

You may want to override the color assigned to the layer (known as the **BYLAYER** color).

(To change the color of an entity after it has been drawn, use the **PropEdit** command, described later on in this chapter.)

To change the current color, click the **Color** button in the control panel. The **Layer Manager** dialog box appears, which is the same dialog box as described earlier. Choose a color from the list and click **OK**.

## Set Layer by Reference

The **LayRef** command (**Options > Set Layer**) provides an alternative form of layer control which you may find more appropriate and direct. You can set, turn off, freeze, lock, or unlock a layer by picking a reference entity.

Choose one of the options from the options bar:

Off	Lock	Freeze	Unlock	On...	Thaw...
-----	------	--------	--------	-------	---------

The **Set** option is the default option for **LayRef**; it is used if you select no other option. When you choose **Set**, the routine prompts:

Select reference entity to set layer:

The layer on which the picked entity is located is now 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 is turned on or frozen.

## MODIFYING OBJECT PROPERTIES



The **PropEdit** command (**Edit > Modify Object Properties**) changes the properties of objects. The characteristics of objects are said to be the “properties.” Objects may have a different:

- **Layer:** default = 0.
- **Color:** default = 7, black.
- **Linetype:** default = Continuous.
- **Thickness:** extrusion in the Z-direction; default = 0.0 units.

All of these characteristics are automatically associated with an entity when it is created. You may want to, in some cases, to change (or override) these parameters. You do this with the command.

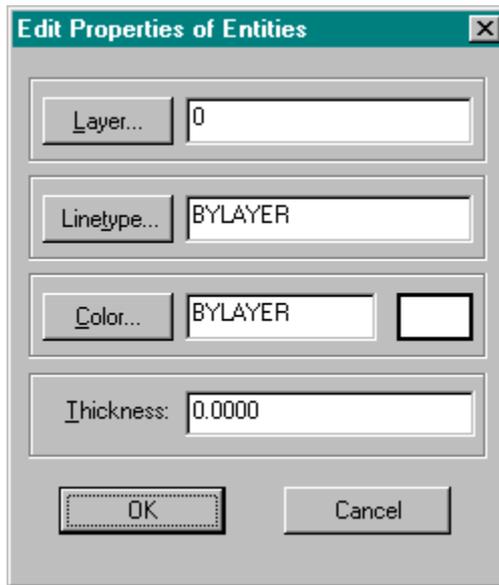
The **PropEdit** command allows you to make these changes:

- Objects may be placed on a different layer.
- Objects may be given a different color.
- Objects may be given a different linetype.
- The thickness of objects may be modified.

You may change any or all of these properties at a time. The changes apply to one, several, or all objects in the drawing.

Select the entities whose properties you want to change. Remember that all selected objects are changed. If three objects are selected, all three receive the same changes.

Press **Enter** to end object selection. FelixCAD displays the **Edit Properties of Entities** dialog box. Choose the object properties that are to be changed by clicking the proper button.



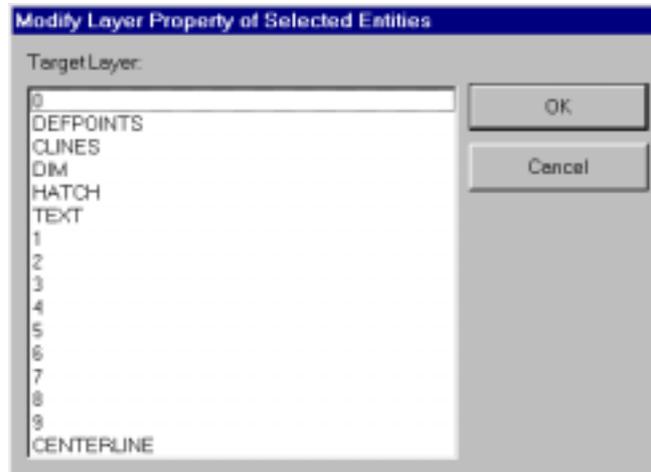
The color, layer, and linetype properties each have a separate dialog box. These dialog boxes are identical to those opened for layer linetype and color control. Choose the new characteristic in each case. Thickness is applied directly and does not require a dialog box.

## MODIFY OBJECT LAYER



The **PropLayer** command (**Edit > Modify Object Layer**) transfers selected objects from their current layer to another layer.

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



## COPY OBJECT PROPERTIES



The **CopyProp** command (**Edit > Copy Object Properties**) allows you to change the properties of entities to match those of another entity. This makes it easy to make a group of entities take on the same look as a red dotted entity. Only properties that are common to all objects (layer, linetype, and color) are transferred to the selected objects.

To change the properties of the selected objects takes two steps:

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

## THICKNESS

By applying the thickness property to entities, you create a three-dimensional effect called “2½D.” The extrusion of entities creates the illusion of depth. Thickness is measured in the Z-direction. You can give thickness to the following entities:

- Arc.
- Circle.
- Line.
- 2D polyline.
- 2D face.

If any of these entities are drawn with a non-continuous linetype pattern (such as dash-dot) or if they are a wide polylines, then the pattern and width is extruded.

Thickness can be assigned to other drawing objects, such as 3D polylines, 3D faces, text, attributes, attribute definitions, block insertions, dimensions, and hatches, but FelixCAD does not display the extrusion. To set a preference for subsequent drawing, set the **Thickness** system variable. Type the system variable name and enter a numeric value. To turn off thickness for further drawing, set the value to zero. To alter the thickness of drawing object, use the **PropEdit** command (see earlier in this chapter).

### Applying Object Selection and Object Snap to Extruded Entities

When selecting or snapping to entities with an extrusion thickness, keep these two rules in mind:

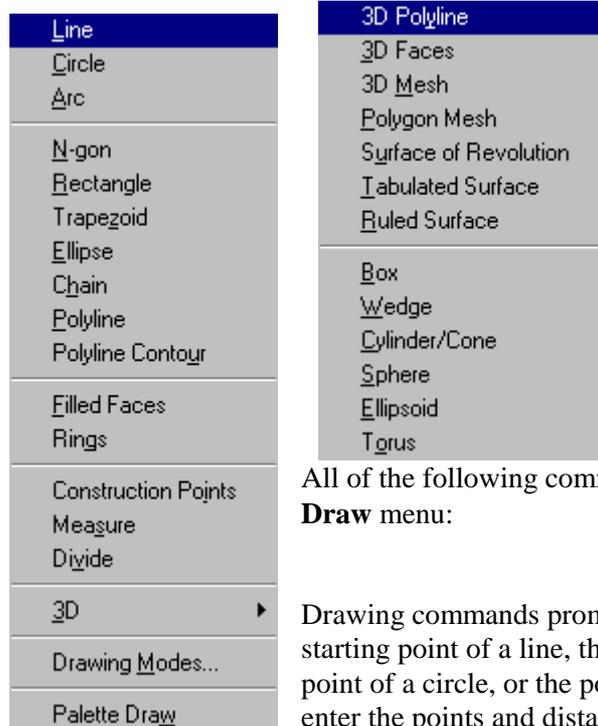
- **Object Selection:** When requested to select objects, FelixCAD recognizes extrusion lines.
- **Object Snap:** When you snap to extrusion line of entities with thickness, FelixCAD finds the appropriate point of the entity’s base.

### Treatment of Extruded Objects in Rendering and Hidden Line Removal

Objects with thickness are recognized by the rendering and hidden-line removal commands: **ARender**, **Render**, **Hide**, and **FHide**.

## Draw: Drawing Basic Entities and Objects

In this chapter, you learn about the commands that draw basic geometric entities. FelixCAD draw lets you drawing with simple entities, like lines and circles, as well as advanced 3D entities, such as surfaces of revolution and three-dimensional polylines. The entities you draw may be changed later using the editing commands (see Chapter 6).



All of the following commands are available from the **Draw** menu:

Drawing commands prompt you for input, such as the starting point of a line, the length of the line, the center point of a circle, or the points to define an arc. You enter the points and distances by pointing with the cursor or by typing discreet coordinates or values on the keyboard.

Options for each command are shown in the option bar. The default is the first option, which is selected by the left mouse button.

### Using Object Snaps

When defining points by pointing with the cursor, you should use object snap options in ensure precise drawings. For more information about object snap, refer to Chapter 2 “Drawing With Precision.”

## Loading Linetypes

The **Linetype** command allows you to choose linetype definitions for later use. The linetype chosen with this option does *not* become the active linetype. It is only 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:

1. From the **Options** menu, select **Line Types** (or type the **Linetype** command). Notice the **Open Linetype File** dialog box.
2. Select a file with the extension of *.lin*. Usually linetype files are found in the **\applic** folder. FelixCAD ships with two linetype files: **Mm.Lin** contains metric linetypes, while **Inch.Lin** contains imperial linetypes.
3. Click **Open**. Notice the **Linetype** dialog box.
4. Select one or more linetypes that you want to load into the drawing.
5. Click **OK**. FelixCAD adds the linetype definitions to the drawing. These are saved with the drawing and are available the next time you open the drawing.

To set a linetype, use the **Layer** command and assign linetypes to layers. Or, use the **SetLinetype** command to preset the linetype for all entities drawn after this point.

## Applying Text Styles

The **Fonts** option from the **Detail** menu allows you to load text styles from font files. A *text style* lets you adapt a font file for special drawing needs, such as presetting its size. You find a more detailed explanation in the “Text Objects.” chapter

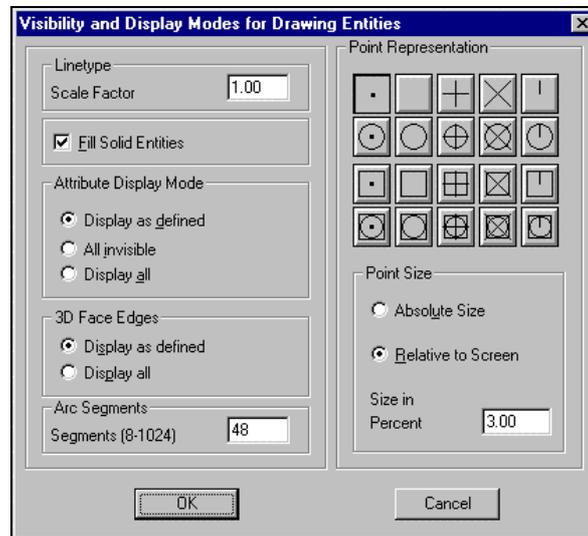
## Specifying the Dimension Type

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

## DRAWING MODES



**Draw > Drawing Modes (DrawMode command)** lets you specify the proper visibility and display control of drawing entities. The command displays the **Visibility and Display Modes for Drawing Entities** dialog box:



When you change a setting in this dialog box, the setting applies to all objects drawn from then on. The changes do not affect all drawn objects until the active view has been regenerated, using the **Regenerate** command found in the **Window** menu.

### Linetype: Scale Factor

The **Linetype Scale Factor (LtScale command)** determines the scale factor for lines drawn with a linetype. The default is 1.0. A larger number results in extended line patterns (dash lines are longer); a smaller number contracts the line pattern (dash lines are shorter). To change the scale factor, enter a new number in the text edit box.

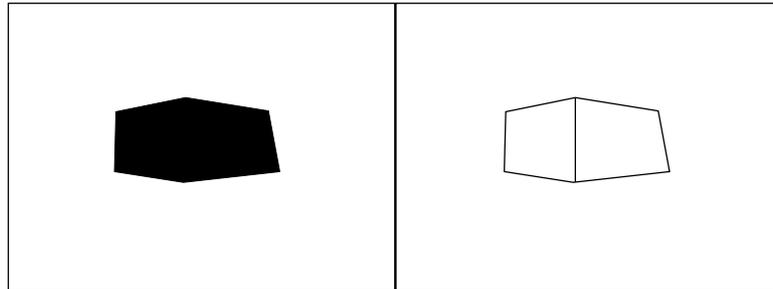
*Note: If the linetype scale factor is set to a very low number, dashed or center-lines may appear to be continuous type. In such cases, re-set the ltScale to a larger value and perform a regen afterwards to see the gaps in these types of lines.*

After a regeneration of the drawing, the scale of line patterns already drawn is adjusted to the new value.

## Fill Solid Entities

The **Fill Solid Entities** option specifies whether or not polygons are filled (*polygons* are polylines with a width greater than zero, or are 2D faces). A check mark in the box indicates the fill function is on. The default is on.

This option must be turned on to draw filled planes. If the **Fill Solid Entities** option is off, then the **Fill Polygons** option is not available in the **Print/ Plot** dialog box (more details in Chapter 13 “Printing and Plotting”). The figure below shows the effect of filling and not filling a polygon.



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

## Attribute Display Modes

This option allows you to choose how attribute text should be displayed. The options are:

- |                           |   |
|---------------------------|---|
| <b>Display as defined</b> | Attributes are displayed according to their definition (the default).         |
| <b>All invisible</b>      | All attributes are made invisible, e.g. their status is defined as invisible. |
| <b>Display all</b>        | All attributes are displayed, regardless of how they were defined.            |

Choose a display option by clicking its radio button.

## 3D Face Edges

In 3D views, if you want to hide intersections or lines that are behind other objects, select the **Displayed as defined** option (the default).

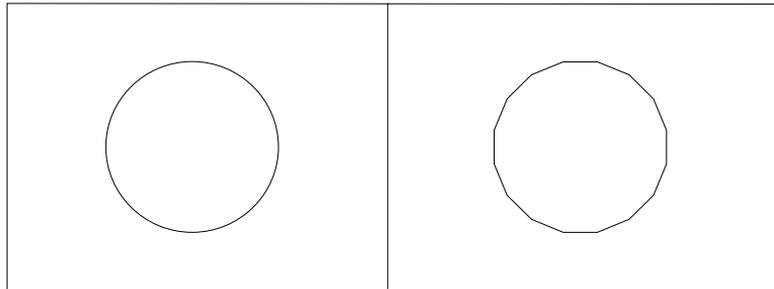
If all intersections are to be displayed, then select the **Display all** option.

## Arc Segments

All CAD programs display arcs and circles on the screen as a series of very short straight-line segments.

This option specifies the number of segments that FelixCAD uses to draw all arcs and circles. You can type any value between 8 and 1024. A smaller value makes the arcs or circles appear jagged, but regenerates the drawing more quickly. A larger number tends to smooth them out. The default is 48.

Note that this number affects the output quality of some printers and plotters.



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 from 20 types of point displays. You don't see the new point mode until you regenerate the drawing; all points take on the new display mode. *Note: AutoCAD users will know this point display option as the PDMODE command*

## Point Size

Although mathematically points have no size, you can specify the display size in two different ways:

<b>Absolute Size</b>	The size of points is determined in the absolute scale of 3 drawing units; the default.
<b>Relative to Screen</b>	The display of points is determined by a percentage relative to the size of the visible drawing plane.

**LINE**

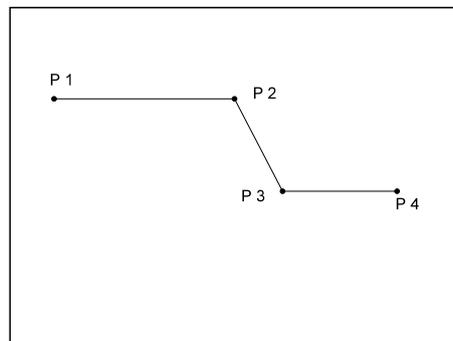
The **Line** command (**Draw > Line**) draws lines and line segments. You specify the line by two points: the starting point and the ending point of the line. The **Line** command displays four choices on the option bar:

Continuous	Segments	Append	TT		
------------	----------	--------	----	--	--

**Continuous Option**

**Continuous** is the default option when drawing lines; this option is used when no other option is chosen. The **Continuous** option allows you to draw line segments in sequence until the **Line** command is terminated by pressing the right mouse button or the **Esc** key.

The **Line** command requests a “From point:” (start point) for the first line segment, and continues to request “To point:” until the command is terminated. Each subsequent point is 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.)*

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 **Undo** and **Close** options are added to the option bar.

The **Undo** option can be used repeatedly to erase the previous line segment, until the line is completely gone.

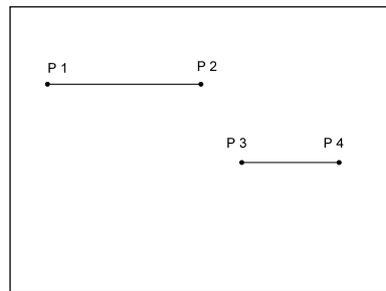
The **Close** option connects the last drawn point with the start point, to create a closed shape.

To terminate the **Line** command, press **Enter** or **Esc**.

### Segments Option

The **Segments** option creates single line segments, as opposed to a multi-segment (continuous) line.

As with continuous lines, the **Line** command prompts for the start and end points of the first segment. The program then repeatedly asks for start and end points. Note the difference: the lines drawn have start points independent of the previous end point; segments are usually not appended to 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>
```

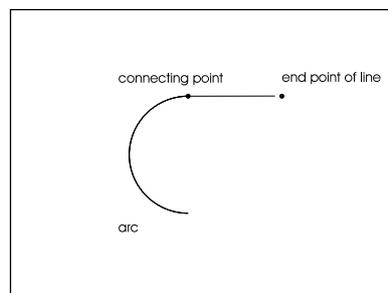
## Append Option

The **Append** option allows you to connect a new line to the end of an existing line or arc. Users familiar with AutoCAD will find this similar but exactly the same as joining a line segment to another with the PEDIT command and 'join' option. When prompted, select the line or arc to connect:

```
From Point: append
Select line or arc: [pick]
To Point: [pick]
```

The new line is connected to the end of the line or arc nearest to the pick point.

When appending the line to an arc, you are prompted the length of the line. The appended line is drawn tangent to the end point of the arc. You can either type the length, or pick the length with the cursor; for precision, it is better to type the length.



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

## Close and Undo Options

After you select one of the **Line** command's option listed above, FelixCAD displays the following options on the option bar:

Undo	Close				
------	-------	--	--	--	--

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

**Undo**

The **Undo** option allows you to undo the last step taken. In other words, the last-drawn segment is erased. The end point of the last line segment may then be used as a starting point of another segment if required. **Undo** can be repeated as long as a segment of the line exists. The last segment is erased with each successive **Undo**.

The end point of the previous segment can then be used as a starting point of the next segment.

**Close**

This option allows you to draw a closed polygon. Choose the **Close** option from the option bar or type **C** at the keyboard. FelixCAD draws a line segment that connects the end point of the last line to the start point of the first line. (The **Close** option also allows you to create closed 3D polylines. )

**TT Option**

The **TT** option is short for “Tangent Tangent.” It allows you to draw a line tangent to two arcs or circles. The **TT** option is useful in creating compound or reverse curves with a line segment in between the arcs.

You are presented with two prompts, as follows;

```
Tangent from:  
Tangent to:
```

After picking the two arcs or circles, the **Line** command continues with the “To point:” prompt.

**CIRCLE**

The **Circle** command (**Draw > Circle**) draws circles. There are several ways to geometrically describe a circle. FelixCAD allows you to choose from among several methods to draw the circle in the most convenient manner. A circle can be drawn by any of the following options:

- Specify the center point of the circle, and the radius or diameter.
- Specify three points on the circumference of the circle.
- Specify two points on the circumference (and, hence, the diameter) of the circle.
- Specify two points tangent to other lines, arc, or circles.

The following options appears on the option bar when calling the command **CIRCLE**:

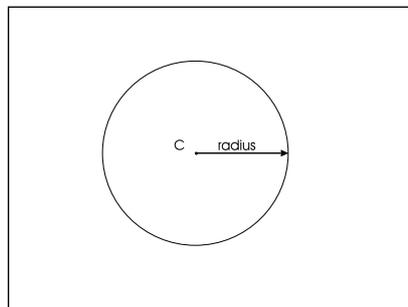
Center	3Points	2Points	Radius	Diameter	Tangent
--------	---------	---------	--------	----------	---------

**Center Option**

Since the **Center** option is chosen most often in drafting, it is the default: it is activated automatically when no other option is selected. You are first prompted for the location of the center point; the second prompt asks for the radius:

```
> circle
Center point: [pick]
Radius <1.00>: [pick]
```

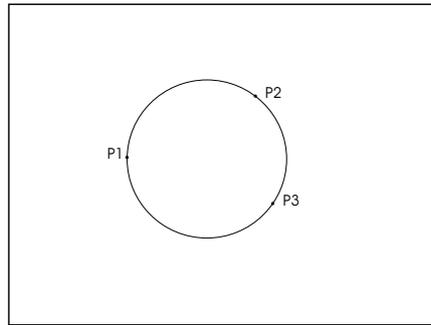
The radius prompt displays the radius, such as <1.00> above, you entered earlier. To accept the default radius, press **Enter**; to change the radius, type or pick a new value.



To enter a diameter value, select the **Diameter** option from the option bar.

### 3Points

The **3Points** option draws a circle that passes through three points on its circumference. This option is convenient when the circle must intersect three tangent lines, or when drawing a circle within a polygon.



The prompt asks for three points:

```
> circle  
Center point: 3P  
First point: [pick]  
Second point: [pick]  
Third point: [pick]
```

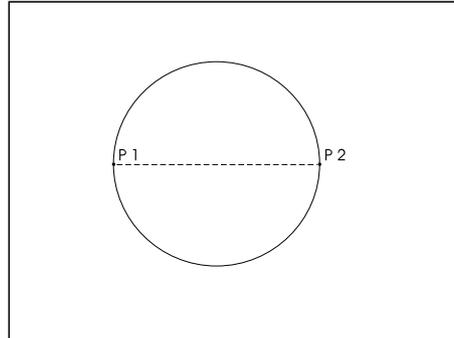
You may pick the points on the screen (we recommend you use object snaps), or enter the coordinates through the keyboard.

## 2Points

The 2Points option draws a circle whose diameter passes through two points:

```
> circle
Center point: 2p
First point: [pick]
Second point: [pick]
```

The points may be picked using the object snap functions, or may be entered from the keyboard.



## Radius Option

The **Radius** option is useful when the radius of the circle is a known, but the center point is undefined, or when the radius follows from geometry already established. To define the radius:

- Verify the last radius by pressing **Enter**.
- Enter a new value for the radius manually.
- Pick two points for the radius in the drawing. This is especially useful when the radius follows from existing geometry.

```
> circle
Center point: Radius
Radius <1.00>: 1.24
Center point: [pick]
Center point: [pick]
Center point: [Enter]
```

This option repeatedly draws circles based on the center points that you pick. Press **Enter** or **Esc** to end the **Circle** command.

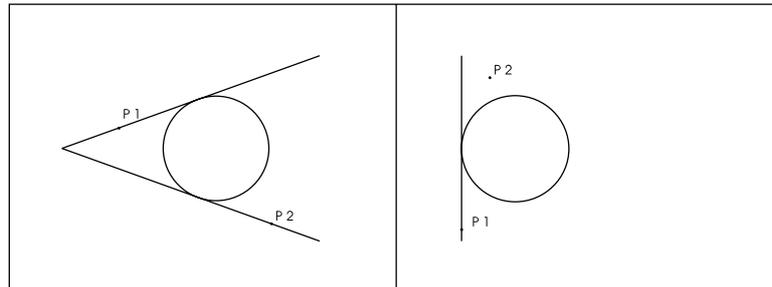
## Diameter Option

The use and the implementation of the **Diameter** option follows exactly the **Radius** option, except that you specify the diameter.

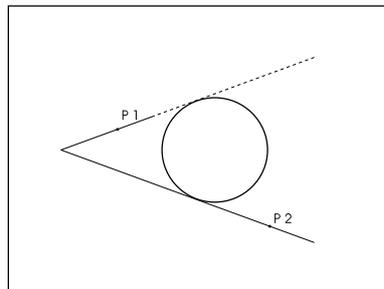
## Tangent Option

The **Tangent** option is used to draw a circle of given radius or diameter tangent to two lines.

The first prompt asks 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 is 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 is moved in the direction of the imaginary point of intersection, so that the defined radius touches 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



The **Arc** command (**Draw > Arc**) draws arc objects. There are many ways to geometrically describe an arc. FelixCAD provides you with these options:

- **3Point**: specify three points, the starting point, a second point, and the end point.
- **SEnd**: specify two points and the radius, or the angle of the arc, or a tangent through the starting point.
- **SCenter**: specify two points and the center point.
- **CStart**: specify the center point, the starting point, and the angle.
- **CRadius**: specify the center point, radius, the starting angle, and the ending angle.
- **Append**: continue an arc from an existing arc or line.

### TIP

Arcs are measured counterclockwise (to the left), starting from the positive X-axis.

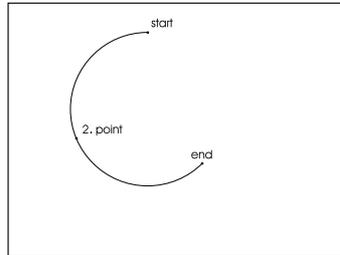
To pick points for the construction of arcs, we recommend that you use object snaps to ensure accuracy.

The **Arc** command displays these options on the option bar:

3Point	SEnd	Scenter	CStart	CRadius	Append
--------	------	---------	--------	---------	--------

### 3Point Option: Start Point, Second Point, End Point

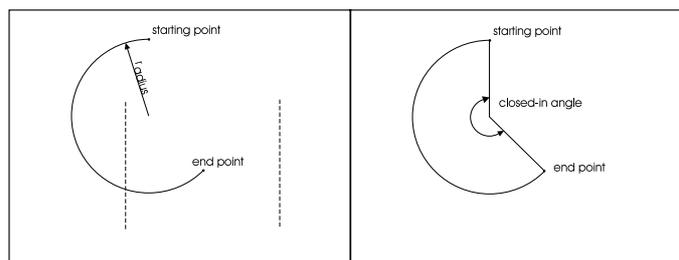
The most common method for drawing arcs is to select three points on the arc. The arc is defined by three points: the starting point, a second point on the arc's circumference, and the end point of the arc. This is the default in FelixCAD; it is used if you chose no other option. Remember the direction convention: the arc is drawn counterclockwise starting at the positive X-axis.



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

### SEnd Option: Start Point, End Point, Radius/Angle/Tangent

The **SEnd** option creates arcs with the starting and end point defined. After that, you can specify a radius, an angle, or a tangent. The defaults of this option simply follow the prompts: entering the starting point, end point, and the radius of the arc by picking or by entering coordinates at the keyboard.



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

Instead of the radius, you can enter an angle or a direction (from the starting point) by selecting **Angle** or **Direction** from the option bar.

### Angle Option

The **Angle** option defines the arc by the included angle. The *included angle* is the angle between the starting point and its end point; the vertex of the angle is at the arc's center point.

```
> arc
Start point: SEnd
Start point: [pick]
Endpoint: [pick]
Radius: Angle
Included angle: 45
```

Enter the value through the keyboard or by picking points on the screen. When picking a value, remember that the X-axis is the point of reference, *not* the last point entered.

### Direction Option

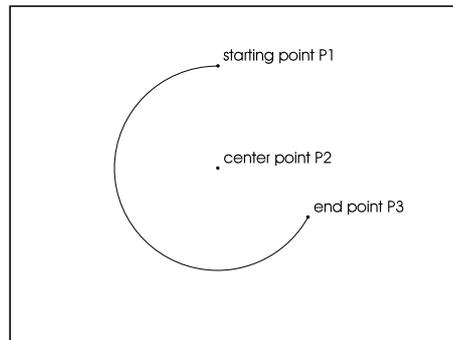
The **Direction** option defines the arc by a tangent that touches the arc at its starting point. Since the starting point of the arc is already defined, you need only define a tangent by keyboarding or picking another point on the entity to be tangent.

```
> arc
start point: SEnd
Start point: [pick]
Endpoint: [pick]
Radius: Direction
Direction from start point: [pick]
```

## SCenter Option

### Start Point, End Point, Center Point

The **S**Center option draws the arc based the starting point, the end point, and a center point.



Select **S**Center from the option bar when prompted for the start point, as follows:

```
> arc
Start point: SCenter
Start point: P1
Center point: P2
Endpoint: P3
```

You may 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 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 Option

### Center Point, Start Point, End Point

The **CStart** option draws the arc beginning with the center point, followed by the start and end point.

After choosing **CStart** from the option 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: P3 or angle specification
```

## CRadius Option

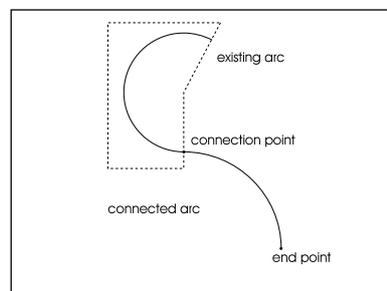
### Center Point, Radius, Angle

The **CRadius** option starts an arc by defining its center point. After that, you specify the radius, and the starting and ending angle of the arc. A rubberband line helps you to determine the angle.

```
Center point: P1
Radius: radius specification
Starting angle: P2
Ending angle: P3
```

## Append Option

The **Append** option appends an arc to lines and other arcs. The arc is determined by two points: the starting point is the connection with the existing line or arc; the end point belongs to 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.



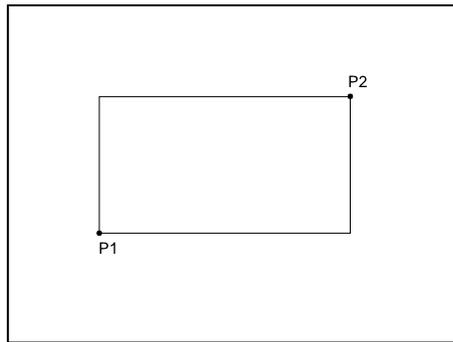
```
Choose the line or arc: P1
End point: P2
```

**RECTANGLE**

The **Rectangle** command (**Draw > Rectangle**) allows you to construct rectangles of any size and aspect ratio, as well as squares. When the command is active, the option bar has a single option:

Width					
-------	--	--	--	--	--

Rectangles are drawn by one of two methods. The first is by entering via keyboard input (i.e. @X,Y or X,Y for relative or absolute co-ordinates). The second is by picking two opposite corners. The resulting rectangle is a single object.



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

The **Width** option allows you to enter or pick a numeric value for the thickness of the lines making up the rectangle. To use the last entered value (as offered by the prompt) just press **Enter**.

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

**NGON**

The **Ngon** command (**Draw > N-gon**) is used to draw a regular polygon of three to 1,024 sides – more than enough for any practical need. This command provides options for three different principles of construction:

- **Side:** specify the length of the sides of the polygon.
- **Outside:** specify the center point and a radius of an external circle.
- **Inside:** specify the center point and a radius of an internal circle.

Select the command for drawing a polygon, and indicate the number of sides. You may specify a width, if desired. The resulting polygon is treated as a single object.

Width					
-------	--	--	--	--	--

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

Side	Outside	Inside			
------	---------	--------	--	--	--

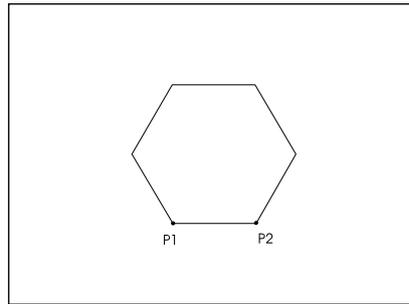
**Side Option**

The **Side** option is the default option for drawing regular polygons. (It is used if no other option is chosen.) The polygon is defined by the length and placement of one side.

To draw a regular polygon, first enter the number of sides: enter a value between 3 and 1024. The default is four sides; if you previously drew a polygon, that number appears in the prompt. To reuse the same number of side, press **Enter**.

After specifying the number of sides, determine the starting point and the end point of the first side. You may either type the coordinates or pick points with the cursor.

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

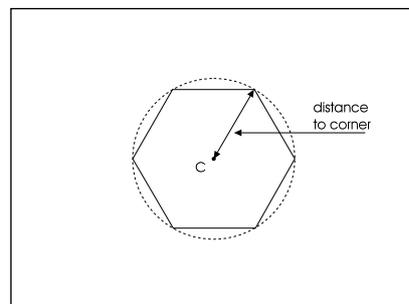


Number of sides <4>: **6**  
 First corner point of the side: **P1**  
 Distance or second point of the side: **P2**

### Outside Option: Circumscribing the Polygon

The **Outside** option 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 **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 is drawn counterclockwise, starting from the first corner point.

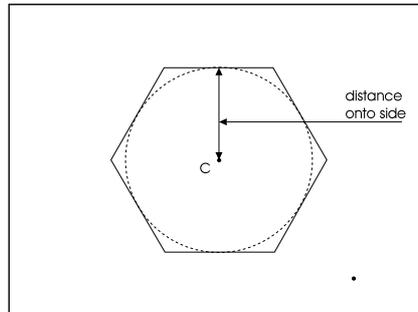


Number of sides <4>: **6**  
 Center point of the polygon: **P1**  
 Distance to its corners: **P2**

**Inside Option:  
Inscribing the Polygon**

The **Inside** option defines the polygon by a circle whose circumference touches each side of the polygon at its midpoint (fits inside the polygon).

Select **Inside** from the option bar, then enter the number of sides. Next, determine the center point of the polygon by entering its coordinates or picking them, then determine the radius of the circle. The polygon is drawn counterclockwise, starting from the defined side.



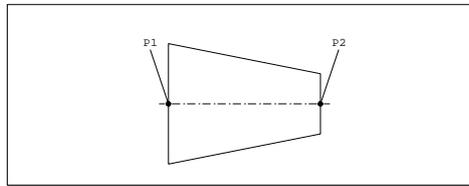
```
Number of sides <4>: 6  
Center point of the polygon: P1  
Distance to its corner points: P2
```

**TRAPEZOID**

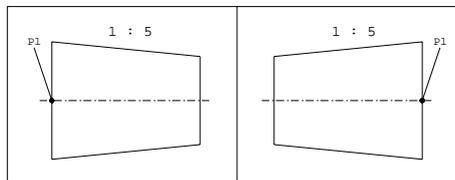
The **Trapezoid** command (**Draw > Trapezoid**) draws a quadrilateral with two parallel sides (trapezoid) on the current layer.

You create the trapezoid by specifying an insertion point and angle of inclusion, followed by the length of the cone and the diameters of its top and bottom edges. The *insertion point* is the midpoint of the first base. You determine the points through pointing or entering the values or coordinates at the keyboard.

The cone can also be drawn using a taper ratio. To do so, press **Enter** at the “Diameter of second side:” prompt. Type a ratio, which must be a real number greater than 1.



```
> trapezoid
Insertion point: P1
Insertion angle <0>: [Enter]
Diameter of first side: 5
Length: 6
Diameter of second side: 2.5
```

**Taper Ration Option**

```
> trapezoid
Insertion point: P1
Insertion angle <0>: [Enter]
Diameter of first side: 5
Length: 6
Diameter of second side: [Enter]
Ratio 1 : 5
```

## ELLIPSE

The **Ellipse** command (**Draw > Ellipse**) draws a full ellipse. FelixCAD provides two methods for drawing ellipse-shaped polygons:

- **Center:** specify the center point, then the radii; the default.
- **Endpoint:** specify the endpoints of the axes.

Center	Endpoint				
--------	----------	--	--	--	--

### Center Option

The **Center** option is the default. You first identify centerpoint of the ellipse, then specify the primary radius (endpoints of the longer axis), and finally the secondary radius (endpoints of the smaller axis). Note that specifying the primary radius locks down the orientation of ellipse.

```
> ellipse
Center point:
Primary radius: @3<45
Secondary radius: 2
```

### Endpoint Option

The **Endpoint** option is the alternative method of constructing an ellipse. You first specify the start and the end point of the major axis, then specify the secondary radius (by entering a numerical value or by rubberbanding and picking the radius from the midpoint of the major axis).

```
> endpoint
Center point: Endpoint
First axis endpoint: [pick end point of major axis]
Second axis endpoint: [pick end point of minor axis]
Secondary radius:
```

*Note: The ellipse will be created as closed polyline of many straight line segments. The CAD user therefore cannot later pick center points of the two radii.*

**CHAIN**

The **Chain** command (**Draw > Chain**) is a combination of the **Line** and **Polyline** commands. The differences between drawing a line, a polyline, or a chain are:

- Continuous *line* draws each segment as a separate entity, which is useful when you want to later select each segment separately.
- *Chain* draws continuous lines similar to the **Line** command with the **Continuous** option. The chain, however, is treated as a single object, which is selected as a whole for editing purposes.
- Like a *polyline*, you can specify a width for the entire chain object. The width last entered for a polyline is used as the default for the next chain, and vice versa. However, a single width must be specified for the entire chain.

A chain is drawn by specifying the points that are the starting and end point of each chain segment. The end point for each segment serves 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				
-------	----	--	--	--	--

**Width Option**

The **Width** option allows you to specify 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.

**TT Option**

The **TT** option draws the chain tangent to an arc or circle to the tangent of a second arc or circle. For more details, see the **Line** command's **TT** option.

## POLYLINE



The **Polyline** command (**Draw > Polyline**) draws 2D entities that consist of connected line and/or arc segments. Each segment can have a different width; in addition, the width of each segment can be tapered. Polylines are always drawn with solid fill.

A polyline is defined as a single object and is treated as such for editing operations, in contrast with entities drawn with the **Line** and **Arc** commands.

After you select the **Polyline** command, the option bar has these options:

Arc	Width	Halfwidth	Append	TT	
-----	-------	-----------	--------	----	--

Polyline segments are always drawn by defining their start and end points, and their width, regardless if they are arcs or lines. After drawing the first segment, you are prompted for the endpoint of the next polyline segment. (The start point becomes 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: [Enter]
```

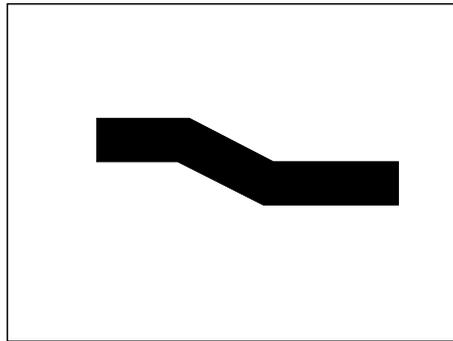
## Width and Halfwidth Options

The **Width** and **Halfwidth** options assist in defining the width of a polyline.

**Width** defines the entire width of a polyline segment, whereas *Halfwidth* defines the width from the centerline to one side of the polyline segment. Accept the default (last entered) value by pressing **Enter** or enter a new width manually or by picking it.

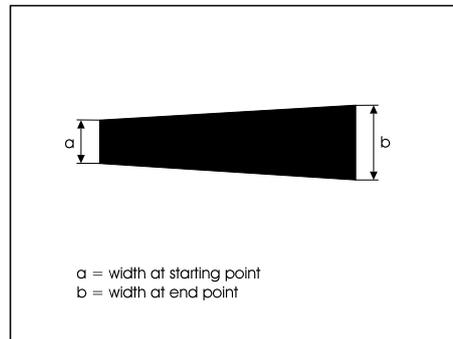
The following rules affect both width options:

- **Uniform Width:** When you choose the **Width** or **Halfwidth** option *before* picking the start point of the first polyline segment, then the width is in effect for the entire length of the polyline.



```
Width for the entire polyline <0.500>: [Enter]  
First point: P1  
Next point: P2  
Next point: P3
```

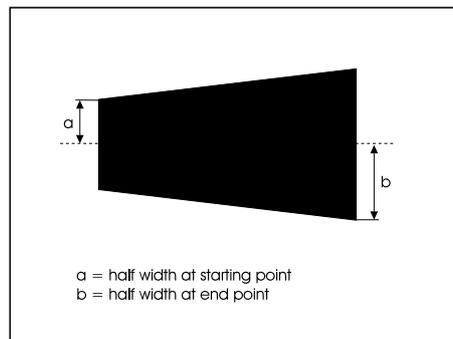
- **Tapered Width:** When you choose the **Width** or **Halfwidth** option *after* picking the start point for the first polyline segment, you may enter different values for the width at the starting point and end point.



```

First point: P1
Next point: Width
Starting width <0.500>: [Enter]
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

```

- **Changing Widths:** The most-recent width is used for all following polyline segments – until you change the width by selecting the **Width** or **Halfwidth** again.

### Arc Option

The **Arc** option lets you include one or more arcs in the polyline; you may give the arc(s) a width. The methods for drawing a polyarc are identical to drawing arc with the **Arc** command.

To draw polyline arcs with a line width, follow these steps:

1. Pick the first point of the arc. This can be the end point of the previous polyline segment.
2. Select the **Width** or **Halfwidth**, option, then specify the starting width and ending width.
3. Determine the other parameters for drawing arcs using the same procedure as drawing regular arcs.

## Append Option

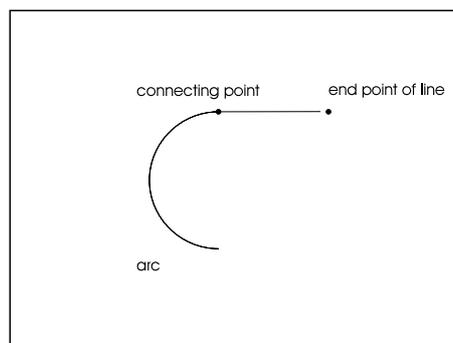
The **Append** option starts the polyline at the end of another line, polyline, or arc. Choose the element (line or arc) to which the polyline still to be drawn is to be appended. Select the **Width** or **Halfwidth** options, if required, and determine the starting and the ending width.

### Polyline to Line Option

To choose an object, select an existing line or polyline. Remember that the polyline is connected to the end point of the line *closest* to the point selected.

### Polyline to Arc Option

To append a polyline to an existing arc, enter the polyline length after selecting the existing arc. A polyline with the specified length is drawn tangent to the end of the arc.



```
Select line or arc: P1  
Length of line: P2
```

To end the **Polyline** command, press **Enter**.

## TT Option

The **TT** option allows you to draw from a tangent of one arc or circle to a second tangent of an arc or circle. See the **Line** command's **TT** option for more information.

**PCONTOUR**

The **Pcontour** command (**Draw > Polyline Contour**) creates a boundary polyline from existing objects that form an enclosed area. (You may recognize this command by the name Boundary or BPoly.) The original entities can be made from a combination of lines, arcs, circles, polylines, chains, and ellipses. They can be in any arrangement, as long as the boundary can create an enclosed area without looping back upon itself. The result is a closed polyline, created on the current layer.

This command can be used in conjunction with the **Area** command to make the job of area calculations easier.

When you start the **Pcontour** command, the following prompts are displayed:

```
Select boundary edges: [any valid selection method is allowed]
Select boundary edges: [Enter]
Ray cast to find start entity <+X>: [described below]
```

Next	+X	-X	+Y	-Y	
------	----	----	----	----	--

```
Select point inside boundary contour: Pick inside the enclosed area
```

When you pick a location that does not work or one of the boundary entities is not a valid entity, then you see the following error message:

```
Point located outside of a boundary contour or given geometry
too complex!
```

**Next Option**

The **Next** option runs a line from the point you specify to the nearest object and then traces the boundary in a counter-clockwise direction.

Usually the **Next** option produces a desirable result. It may not, however, be satisfactory when the space between two possible boundaries is very narrow. By selecting one of the other options, listed below, FelixCAD looks for the boundary in a specific direction from the point you select.

**+X Option**

The **+X** option projects an imaginary line in the positive X-direction. The line run from the point you specify to the first object encountered. After that, FelixCAD traces the boundary in the counterclockwise direction.

**-X Option**

The **-X** option projects an imaginary line in the negative X-direction. The line run from the point you specify to the first object encountered. After that, FelixCAD traces the boundary in the counterclockwise direction.

**+Y Option**

The **+Y** option projects an imaginary line in the positive Y-direction. The line run from the point you specify to the first object encountered. After that, FelixCAD traces the boundary in the counterclockwise direction.

**-Y Option**

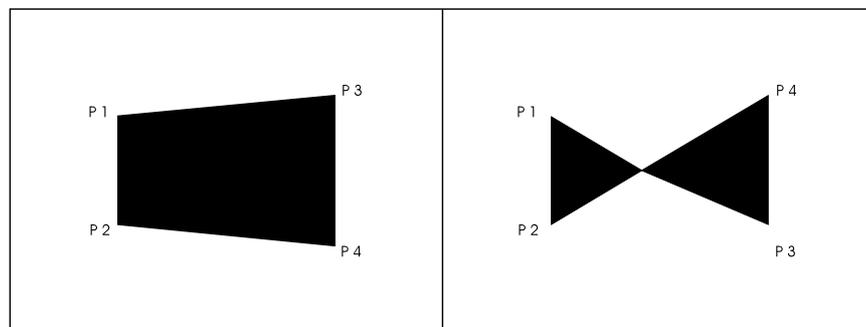
The **-Y** option projects an imaginary line in the negative Y-direction. The line run from the point you specify to the first object encountered. After that, FelixCAD traces the boundary in the counterclockwise direction.

**FILLED FACES**

The **2DFace** command (**Draw > Filled Faces**) draws a solid-filled plane with straight-line borders. The planes can have three or four sides. The command may be done continuously or using segments. It is also possible to connect 2D planes to existing 2D faces.

Two-D faces are always drawn by determining the corner points (there are either three or four corners). The **Close** option is for drawing triangular planes: the third entered point is connected with the starting point of the plane.

The **2DFace** command is sensitive to the order in which corners are picked. 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. The figure below shows the consequences of picking corners in two different ways.



Enter the command **2DFace** for drawing filled faces and select an option from the option menu:

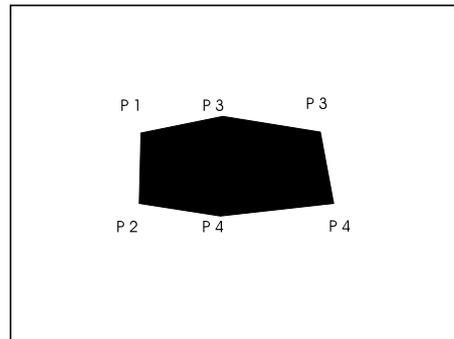
Continuous	Segments	Append			
------------	----------	--------	--	--	--

## Continuous Option

The **Continuous** option is the default for drawing filled planes; it is used automatically when no other option is chosen.

When drawing rectangular planes, determine the end points of the edges. Remember that the first and third points are connected on a single edge, as are the second and fourth points. The four edges will make up the plane.

After determining the end points of the first two edges, you may enter additional points. These are used as the third and fourth edge. The command repeats itself, allowing you to add more planes connected to the preceding plane. To terminate the command, press **Enter**.



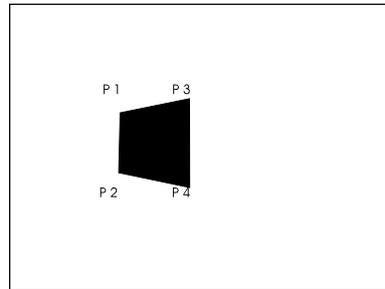
```
> 2DFACE
First point: P1
Second point: P2
Third point: P3
Fourth point: P4
Third point: P5
Fourth point: P6
Third point: [Enter]
```

To draw a triangular plane, specify three end points for two of the edges, then choose the **Close** option.

```
> 2DFACE
First point: P1
Second point: P2
Third point: P3
Fourth point: Close
```

## Segments Option

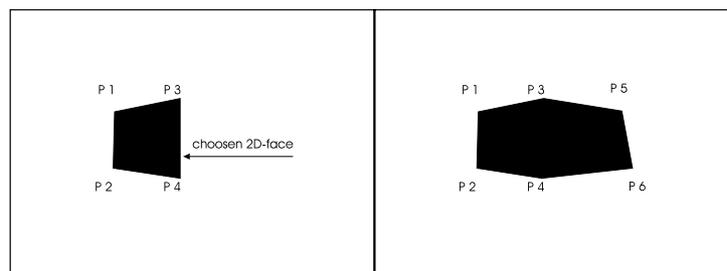
The **Segments** option draws a single rectangular or triangular filled plane. This option ends after the entry of the points required for one plane, in contrast to the **Continuous** option. For triangular planes, there is no difference from the **Continuous** option .



```
> 2DFACE
First point: P1
Second point: P2
Third point: P3
Fourth point: P4
```

## Append Option

The **Append** option allows you to append a 2D plane to existing planes. You can only append a plane to the edge of the last drawn plane. After choosing the option, select an edge of an existing plane, then determine the corner points of appending plane.



```
> 2DFACE
Choose a side: Identify edge
Third point: P5
Fourth point: P6
...
```

## RINGS (DONUT)



The **Ring** (or **Donut**) command (**Draw > Rings**) draw solid rings (solid-filled or thick-walled circles). The ring is a special case of a closed polyline; it is essentially two polyarcs with width. The filled circle is a special case of a ring: it is a ring with the interior diameter equal to zero.

Rings can be edited with any of the polyline editing commands. They behave just like a polyline object.

The last entered diameter values are used as an option for the next entry. If you want to keep using the same values, just pick a new center point. The **Ring** command repeats itself, always creating identical rings, until you press **Enter**.

```
> RING
Center point: P1
Center point: P2
Center point: [Enter]
```

### Diameter Option

The **Diameter** option determines new values for the inside and outside diameters.

Diameter					
----------	--	--	--	--	--

```
> ring
Center point: Diameter
Inside diameter <0.50>: 0.75
Outside diameter <1.00>: 1.25
Center point: P1
Center point: [Enter]
```

## POINTS



The **Point** command (**Draw > Construction Points**) draws a point. The points can be displayed in a variety of styles and sizes. With the **Point** command, you can:

- **Point**: insert individual points.
- **Measure**: insert points at a specified distance from each other along a line, arc, polyline, etc.
- **Segments**: insert a number of points along a line.

Point	Measure	Segments			
-------	---------	----------	--	--	--

The shape and size of the points is determined by the **DrawMode** command's **Drawing Parameters** dialog box (see the beginning of this chapter for more details).

The points are always be inserted on the current layer.

### Point

The **Point** option is the default option. You determine the position of the point by typing coordinates at the keyboard or by picking points on the screen. The command automatically repeats itself until you press the **Esc** or **Enter** key.

```
> point
Point: [pick]
Point: 1,2
Point: [Enter]
```

## Measure



The **Measure** option allows you to insert points along lines, polylines, circles, and arcs at a defined spacing. This dissects the entity into sections of constant size (the drawing object is *not* cut into pieces but remains a complete object.) Remember that all entities are not an exact multiple of the length of the segment, so there may be a remainder. After starting the **Point** command, select the **Measure** option from the option bar:

```
> point
Point: Measure
Select line, arc, circle, or polyline: [pick]
Length of segments: 0.1
```

Choose an entity and specify the spacing between segments (in drawing units), either by typing a numerical value or by picking points on the screen. The measurement of the distance between points begins from the start point of the segment, and progresses to the end point. When inserting points on a circle, the starting point lies at the intersection of the radius with the positive X-axis.

## Segments Option



The **Segments** option 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 **Measure** option, there is no remainder when using the **Segments** option (the object being segmented is *not* cut into different pieces but remains one complete object).

Select the **Segments** option from the options bar, then select an object. Specify the number of segments; the number of segments should be a whole integer between 2 and 4,096.

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.

## COMMAND MEASURE AND COMMAND DIVIDE

### Introduction

The command measure and the command divide orders points or a block multiple along a 2D-Object. This object can be an arc, a line, a circle or a 2D-Polyline. If a block is selected, it can be inserted aligned or non-aligned to the object. Both commands work very similar. While the command MEASURE asks for a distance, DIVIDE requests for the number of points and calculates the distances itself.

### Command sequence for MEASURE and DIVIDE

#### Measure

After commencing the **Measure** command (or selecting it from the menu DRAW), the following prompt appears:

```
> measure  
Select a line/circle/2d-polyline or an arc to measure:
```

After selecting the entity it is highlighted and the following question appears:

```
Block or Distance:
```

Here you can input either a distance directly (by putting in a real value or selecting two points) or you can type "Block" (or "B". "Bl" etc), to use a block. If inputting a distance, the points at each distance from the beginning to the end of the object are inserted and the command is now completed. If you type in "Block", you are then asked for the block name:

```
Name of Block to insert:
```

Now type in the name of an existing block. You will be asked for the offset angle for the alignment of the block along the object.

```
Offset angle for alignment the block along the object (or  
ENTER for none alignment):
```

Here you can press return to avoid the alignment or put in any offset angle which is added to the tangential angle of the object at the point of insertion. Finally, you are asked for the distance again.

```
Distance for dividing the Object:
```

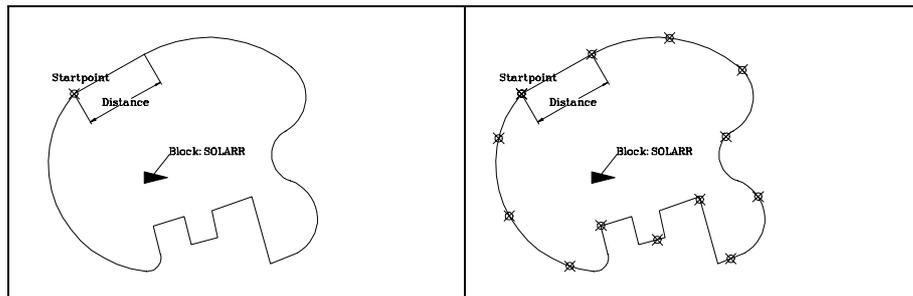
Here, you have to input a distance either by giving a real value or by selecting two points. Now the Block at each position is inserted and the command is complete.

### Divide

The sequence for the **Divide** command is the same. The only difference is that you are asked for a number of points or blocks to insert instead of a distance. In this case, the command will compute the length between the insertion points itself.

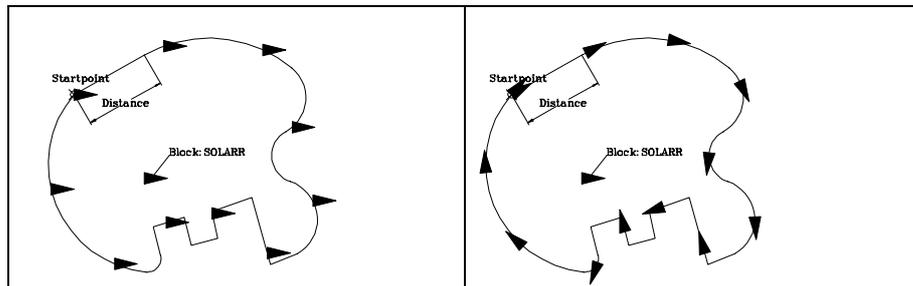
### Example

The following pictures show a sample polyline and how the commands MEASURE and DIVIDE work with it:



The initial state: The entity to measure is a polyline. Moreover the block SOLARR is defined.

After measuring the polyline using the distance, points at each distance along the polyline are sampled, started at the start point of the polyline



After dividing the polyline in ten pieces the block SOLARR is inserted ten times along the polyline, but it is not aligned.

In this case the polyline is divided into 12 pieces with the option block and aligned with an angle of 0 degrees, this means the block is inserted tangential at the object at each point.

## Technical Notes for Divide Command

The command **Divide** inserts number +1 entities along the object unless a circle is selected. So it might be necessary to delete the first or last inserted entity, if a closed polyline is selected. However, this is much easier than creating an additional entity manually.

The command is not an internal command and so it is available for usage inside the (command ...) - function. To use it inside of a LISP-application, use the related LISP-Function (MesDiv EntName Dist|Nr B1Name OffsAngle) instead, which is defined in MesDiv.LSP in the FCAD-Applic directory.

## 3D POLYLINE

The **Poly3D** command (**Draw > 3D > 3D Polyline**) draws an entity made of connected straight line segments. You can specify X, Y, Z coordinates for each vertex.

Unlike 2D polylines, you cannot include arc segments in a 3D polyline, nor can you give the 3D polyline a width. The entire 3D polyline has a width of zero.

As opposed to a line created with the **Line** command, a 3D polyline is treated as a single object. Terminate the command by pressing the **Enter** key.

```
> POLY3D
First point: P1
Next point: P2
Next point: P3
Next point: [Enter]
```

## Undo and Close Options

During the command, the option bar displays the following options:

Undo	Close				
------	-------	--	--	--	--

### Undo

The **Undo** option allows you to undo the last step taken. In other words, the last-drawn segment is erased. The end point of the last polyline segment may then be used as a starting point of another segment if required. **Undo** can be repeated as long as a segment of the polyline exists. The last segment is erased with each successive **Undo**.

The end point of the previous segment can then be used as a starting point of the next segment.

**Close**

This option allows you to draw a closed polyline. Choose the **Close** option from the option bar or type **C** at the keyboard. FelixCAD draws a polyline segment that connects the end point of the last line to the start point of the first line.

**3D FACES \***

The **3DFace** command (**Draw > 3D > 3D Faces**) creates a three- or four-sided surface in 3D space; you may specify XYZ coordinates for its vertices. The difference between 2D and 3D surfaces is that the 3D surfaces are not shown as solid areas but as a wireframe.

The points defining the 3D face are in either a clockwise or counterclockwise direction, so that one edge of a 3D surface is used as the base for the next 3D face. You may specify invisible edges, which is useful for improving the clarity of complex 3D models.

These options to draw 3D faces may be selected from the options bar:

Continuous	Segments	Append	Invisible		
------------	----------	--------	-----------	--	--

**Continuous Option**

The **Continuous** option is the default when drawing 3D surfaces. To draw a 3D face, define the corner points in a clockwise or counter-clockwise direction.

**Triangular 3D Faces**

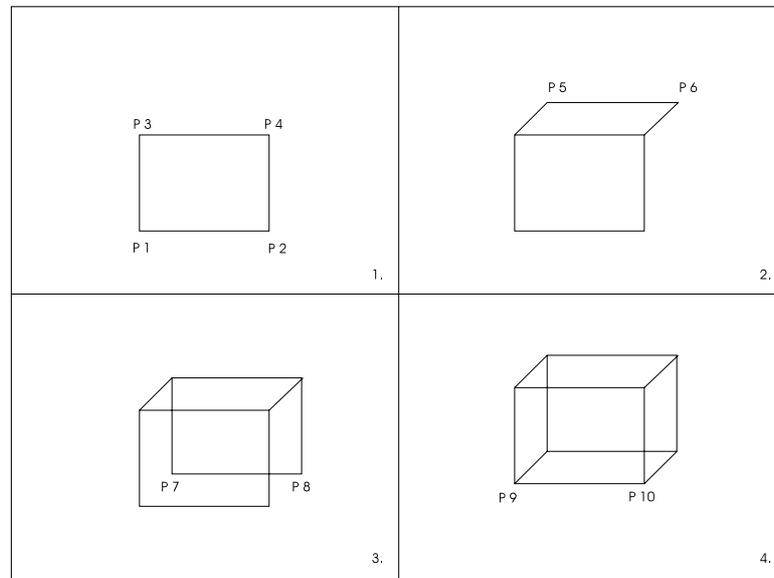
For a triangular 3D face, define three corners, then select the **Close** option. The last defined point is then joined to the first point, forming a triangle.

```
> 3DFACE
First point: P1
Next point: P2
Next point: P3
Next point: Close
```

**Rectangular 3D Faces**

For a rectangular 3D face, define four points. After having completing the fourth vertex, FelixCAD creates the four-sided object.

The command then continues. You are prompted for further points: the next 3D face uses the last drawn edge as the base for the next 3D face.



End the **3DFACE** command by pressing the **Enter** key.

```
> 3DFACE
First point: P1
Second point: P2
Third point: P3
Fourth point: P4
Third point: P5
Fourth point: P6
Fourth point: P8
Third point: [Enter]
```

## Segments Option

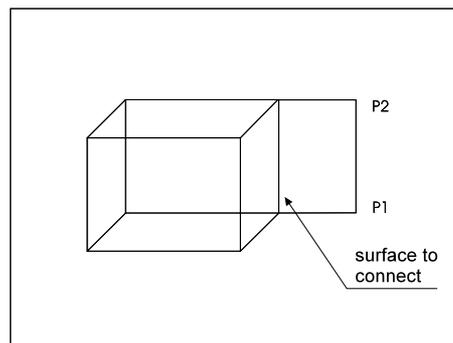
To draw a single 3D-face use the **Segment** option. The method is nearly identical to that just described:

```
> 3DFACE
First point: Segments
First point: P1
Second point: P2
Third point: P3
Fourth point: P4
```

In contrast to the **Continuous** option, the command ends with the fourth point. The **Segment** option permits the **Close** option, allowing you to create a single triangular 3D face. To add more 3D surfaces to an existing one, use the **Append** option.

## Append Option

The **Append** option allows you to connect new 3D faces to existing 3D faces. First, select the edge that should act as the base for the connection, then define the vertices for the new 3D surface. The option automatically changes to **Continuous** mode. This means that after having connected one 3D face, you can specify additional points to create more surfaces.



```
> 3DFACE
First point: Append
Select line or edge of 3DFace: [pick]
Third point: P1
Fourth point: P2
Third point: [Enter]
```

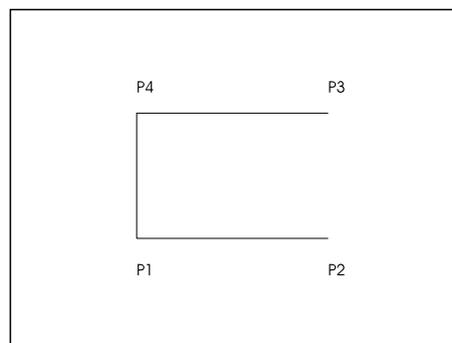
## Invisible Option

The **Invisible** option draws the 3D face, with invisible edges. This feature is useful when dealing with complex drawings containing a large number of 3D faces. The invisible lines makes 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
```

and results in the following:



**POLYGON MESH \***

The **PFace** command (**Draw > 3D > Polygon Mesh**) creates three-dimensional polygon meshes of arbitrary topology and surface properties.

To create a polyface mesh, first specify the control points (vertices). Then specify which vertex is on which face of the polyface mesh.

```
> PFACE
Control point 1: [Specify a point]
Control point 2: [Specify a point]
Control point 3: [Specify a point]
Control point 4: [Enter]
Face 1, Control point 1: 1
Face 1, Control point 2: 2
Face 1, Control point 3: 3
Face 1, Control point 4: [Enter]
Face 2, Control point 1: 1
Face 2, Control point 2: 3
Face 2, Control point 3: 4
Face 2, Control point 4: [Enter]
Face 3, Control point 1: 2
Face 3, Control point 2: 3
Face 3, Control point 3: 4
Face 3, Control point 4: [Enter]
Face 6, Control point 1: [Enter]
```

Pressing **Enter** twice in a row exits the command.

**Polyface properties**

You edit a polyface mesh as a single entity by applying editing commands, such as **Delete**, **Move**, **Copy**, **Rotate**, **Scale**, **Stretch**, **PropEdit**, and **ChProp**. You cannot use the **PolyEdit** command to edit polyface meshes, nor can you use the **Explode** command on polyface meshes.

Note that users rarely employ the **PFace** command. Instead, the command is meant for generating meshes by FLISP routines, based on 3D point lists (such as surveyor's measurement points).

Do not confuse polyface meshes created by **PFace** with polygon meshes created by the **3DMesh** command.

**3D MESH \***

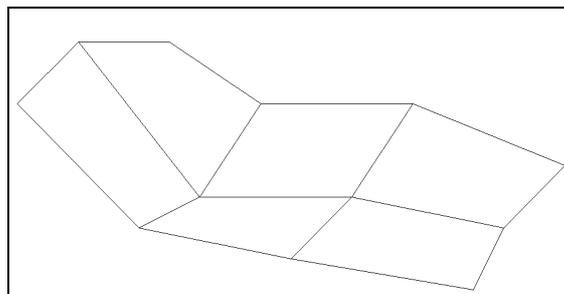
The **3DMesh** command (**Draw > 3D > 3D Mesh**) draws polygon meshes. Usually, it is used for generating irregular surfaces, vertex by vertex. For example, it is used in surveying and GIS applications to create topological surfaces.

A 3D mesh is a wireframe rectangular blanket composed of  $M$  column lines by  $N$  row lines passing through a matrix of  $M \times N$  three-dimensional points in space.  $M$  and  $N$  are indices specifying the number of rows and columns that make up the mesh, and determine the number of vertices required in the mesh.

To create a polygon mesh:

1. Specify the mesh  $M$  and  $N$  size.
2. The command prompts for vertex coordinates until you have defined all vertices in the  $M$  direction, then in the  $N$  direction. You are prompted for each vertex in order, starting with vertex (0,0), continuing with the second vertex in the first column (0,1), and so on.
3. Once all vertices of a column are specified, the first vertex of the next column is requested. For example, vertex (1,0) is the first vertex in the second column. In the figure below, the last vertex is (3,2).

```
> 3DMESH
M-value of the mesh (between 2 and 256): 4
N-value of the mesh (between 2 and 256): 3
Vertex (0,0): 110,170,4
Vertex (0,1): 130,190,4
Vertex (0,2): 160,190,6
Vertex (1,0): 150,130,0
Vertex (1,1): 170,140,0
Vertex (1,2): 190,170,2
Vertex (2,0): 200,120,0
Vertex (2,1): 220,140,-2
Vertex (2,2): 240,170,0
Vertex (3,0): 260,110,0
Vertex (3,1): 270,130,0
Vertex (3,2): 290,150,0
```



Meshes can be open or closed, depending on whether the mesh joins in either the  $M$  or  $N$  direction, or both. A donut-shaped mesh is an example of a mesh closed in both directions.

The **3DMesh** command is rarely used as an explicit command in drawing practice. However, the command provides an easy way to generate meshes by FLISP routines based on 3D point lists (like surveyor's measurement points). The command is primarily designed for usage within applications (either based on FLISP or on other programming languages) to construct general polygon meshes.

Do not confuse polygon meshes created by **3DMesh** with polygon faces created by the **PFace** command.

## **SURFACE OF REVOLUTION \***

The **RevSurf** command (**Draw > 3D > Surface of Revolution**) creates a 3D surface by revolving a path curve (the *profile*) around an axis of revolution. The curve sweeps around the axis to define the surface in the *N* direction of the mesh. The *M* direction of the mesh is determined by the axis of revolution.

The density of the mesh is based on the value of the **SurfTab1** and **SurfTab2** system variables.

To generate a surface of revolution:

1. Pick an entity, such as a line, arc, circle, or polyline. This entity determines the profile.
2. Pick an entity to be the axis, such as a line or open polyline. The profile revolves around the axis.
3. Specify the start angle, and the included angle through which to revolve the profile. The default is a full circle.

> **REVSURF**

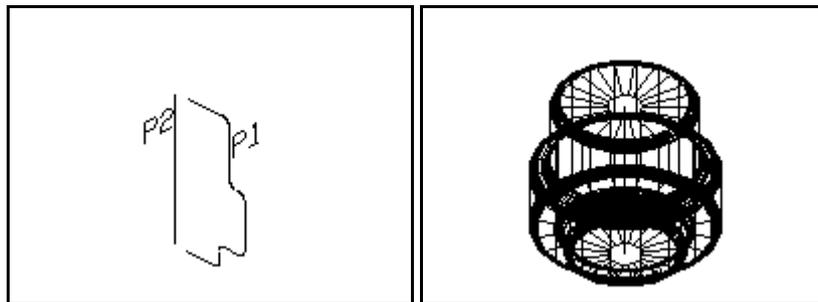
Select path curve to revolve: [**pick**]

Select axis of revolution: [**pick**]

Start angle <0>: [**Enter**]

Included angle to revolve entity (+=ccw, -=cw) <360>: [**Enter**]

**Note:** To close the mesh in the *N* direction, select a circle or closed polyline as the curve.



Revsurf Path Curve & Axis (P1 & P2) and Completed Revsurf Surface

## **RULED SURFACE \***

The **RuleSurf** command (**Draw > 3D > Ruled Surface**) creates a ruled surface between two boundary curves (two *profiles*). Valid entities that define the

edges of the surface can be points, lines, arcs, circles, 2D polylines, and 3D polylines. You can use a mixture of these entities, such as a point and a line.

The **SurfTab1** system variable determines the number of equal intervals between the resulting vertices on the defining profile entities. The number of intervals is the same for each curve.

The resulting surface is a polygon mesh.

To create a tabulated surface:

1. Pick an entity that defines the first profile.
2. Select another entity for the other boundary outline.

```
> RULESURF
```

```
Select first defining curve of ruled surface: [pick]
```

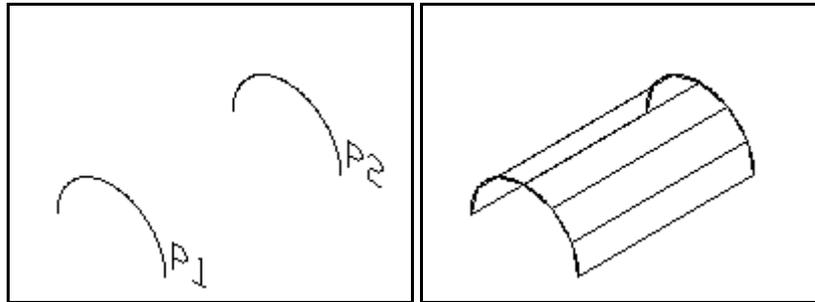
```
Select second defining curve of ruled surface: [pick]
```

**Tips**

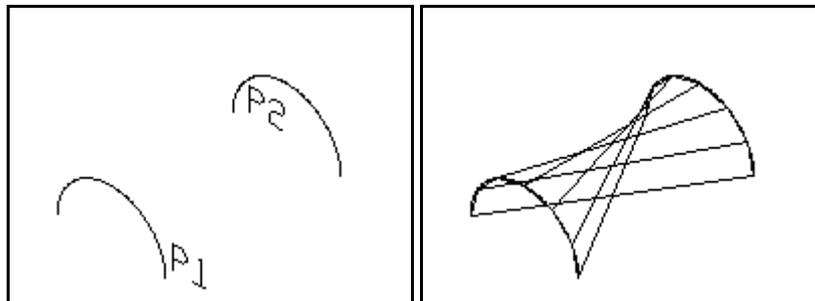
If the first entity is an *open* entity (such as an arc or line), the second entity must also be an open entity. If the first entity is a *closed* entity (such as a circle or a closed polyline), the second entity must also be a closed entity.

If one entity defining an edge is a point, the other entity cannot be a point entity.

To create a mesh, select each entity at the same end. If you select each outline entity at opposite ends, FelixCAD generates a self-intersecting polygon mesh.



Correct Method for Rulesurf command and result



Incorrect Method for Rulesurf command and result

**TABULATED SURFACE \***

The **TabSurf** command (**Draw > 3D > Tabulated Surface**) creates a tabular (or extruded) surface using a path curve and a direction vector.

You can extrude lines, arcs, circles, 2D, or 3D polylines. The entity provided as path curve defines the surface of the mesh. The direction vector (extrusion path) is determined by picking a line or an open polyline.

The **SurfTab1** system variable defines the number of *M*-direction surfaces on the extruded entity. When extruding a line, arc, or circle, the number of *M*-direction surfaces is evenly divided determined by the value of the **SurfTab1** system variable. When extruding a polyline, extrusion lines are drawn at the ends of the line segments, and the arc segments are divided evenly into the number of surfaces determined by the value of the **SurfTab1** system variable.

If the polyline has been curved (fitted) or splined, the number of *M*-direction surfaces is evenly divided using the **SurfTab1** variable.

The resulting surface is a polygon mesh (stored in the drawing database as a complex polyline entity).

To create a tabulated surface:

1. Select the entity for the path curve.
2. Specify the direction vector:

```
> TABSURF
Select path curve: [pick]
Select direction vector (line or open polyline): [pick]
```

The direction and length of the line determine the direction and length of the extrusion. Only the first and last points of a polyline are used when determining the extrusion path. The point at which you specify the entity determines the extrusion direction.



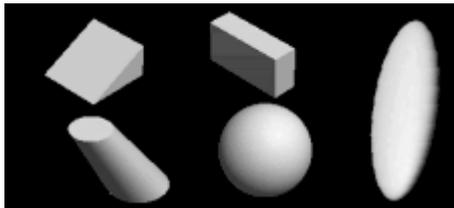
Tabsurf command. Two mouse click points (P1, P2) and resulting surface.

**RECOMMENDED MESH CONSTRUCTION \***

The system variables that control the M and N indices of meshes are **SurfTab1** and **SurfTab2**. **SurfTab1** sets the M index, and **SurfTab2** sets the N index.

Although you can specify values up to 256 for either index, it is not recommended to use high values in mesh construction. The reason is that a dense mesh increases the processing time of your drawing.

You cannot change the M and N values an existing mesh. If you are not satisfied with the mesh, you must erase the mesh, reset the **SurfTab1** and **SurfTab2** variables, and then recreate the new mesh.

**NEW 3D FUNCTIONS \***

Starting with FelixCAD version 4.01, a series of new 3D functions are available to the CAD user. Object creation is based on intuitive input options available to the user and/or programmer.

The new objects are under the 3D option in the DRAW menu are:

**Box \***

The command **3DBOX** (**Draw>3D>Box**) allows simple creation of a 3 dimensional box.

**Wedge \***

The command **3DWEDGE** (**Draw>3D>Wedge**) allows creation of a 3 dimensional wedge.

**Cylinder/Cone \***

The command **3DCONE (Draw>3D>Cylinder/Cone)** allows creation of a 3D cone, cylinder, prism, or pyramid. Object creation is determined by user input. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

**Sphere \***

The command **3DSPHER (Draw>3D>Sphere)** allows creation of a 3D sphere. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

**Ellipsoid \***

The command **3DELLIPS (Draw>3D>Ellipsoid)** allows creation of a 3D ellipsoid object. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

**Torus \***

The command **3DTORUS (Draw>3D>Torus)** allows creation of a 3D torus. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

## Modifying Drawing Objects

<b>M</b> ove
Copy
Rotate
Scale
Flip
Mirror
3D Rotate
3D Mirror
Stretch
Array
Offset Through Point
Offset Distance
Trim
Expand
Fillet
Chamfer
Intersect
Lengthen
Delete partial
Break
Rejoin
Edit Polyline ▶
Change
Modify Parameters...
Palette Modify

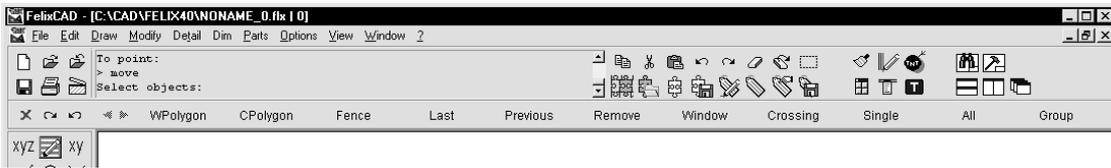
In this chapter, you learn about the modification and editing commands found in the **Modify** menu.

These commands let you modify entities by moving, rotation, scaling, mirroring, and copying them. They also provide a way easily perform time-consuming editing, such as fillet, chamfer, trim, and stretch.

Some commands are specific to some entities, such as editing polyline vertices. Before you can use any of these commands, a drawing must be open, and at least one entity must be in the drawing.

## Object Selection

All modify commands request that you select one or more objects for modification. The option bar displays all the methods of object selection available:



For most commands, you may select more than one object for editing. However, some commands, such as **Trim** and **Expand**, expect a single object for some of their prompts.

When multiple objects are selected, they are transformed together in the same relationship as they are currently placed. For more information on object selection, see Chapter 2 “Drawing with Precision”.

## MODIFY PARAMETERS



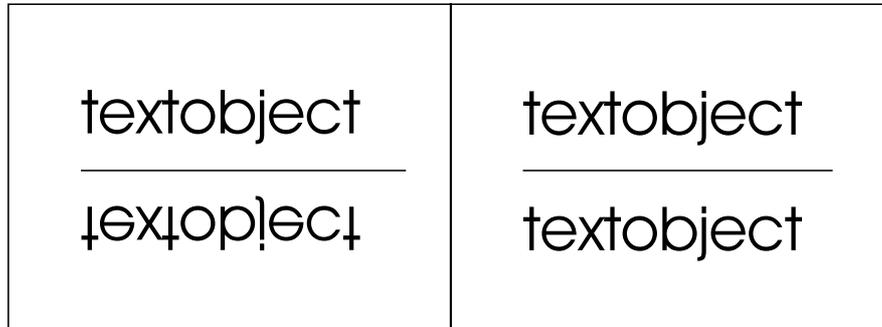
The **EditPar** command (**Modify > Modify Parameters**) allows you to define the default values for some modifying and editing commands. The command displays the **Edit Parameters** dialog box:

### Mirror / Flip

The **Mirror / Flip** area determines what happens to the orientation of text and hatch patterns after the commands that mirror or flip objects.

## Mirror Text

When the **Mirror Text** option is on (a check mark is shown), text objects are mirrored. When turned off (the default) text is copied, and not mirrored, about the specified axis. The following figure shows the influence of this option on text objects.



Mirroring of a text object, option **Mirror Text** activated (left) or not activated (right)

## Mirror Hatching

When the **Mirror Hatching** option is on (a check mark is shown), hatch objects are mirrored. When turned off (the default) hatching is copied, and not mirrored, about the specified axis.

## Offset & Fillet / Chamfer

The **Offset** and **Fillet / Chamfer** areas are discussed later in this chapter. The values set for in this dialog box are only default values, which can be changed by the **Offset**, **Fillet**, and **Chamfer** commands themselves.

## Common

The **Highlight objects when selected** option determines whether objects you select are highlighted by a different color. When on (the default), selected objects are displayed in the magenta (pink) color. When off, selected objects do not have the dashed line, which is useful for slower computers.

The **Display highlighted objects as a dashed line** option determines the look of highlighted objects. When on, selected objects are displayed by dashed lines, which is useful for monochrome displays. When off (the default), selected objects do not have the dashed line.

The **Let Groups be selected** option determines what happens when you pick an entity that is part of a group. When this option is on (the default), picking the entity picks the entire group; when off, only the entity is picked. Groups are discussed in detail in “Parts and Attributes, XRefs, & Groups.”

**Mouse Context ON/OFF.** Found under the TOOLS menu. When activated ('ON') this tool will automatically show any modifications available to any object via an option dialog box, by simply left-mouse clicking on it. It allows changes to objects without searching for the applicable commands.

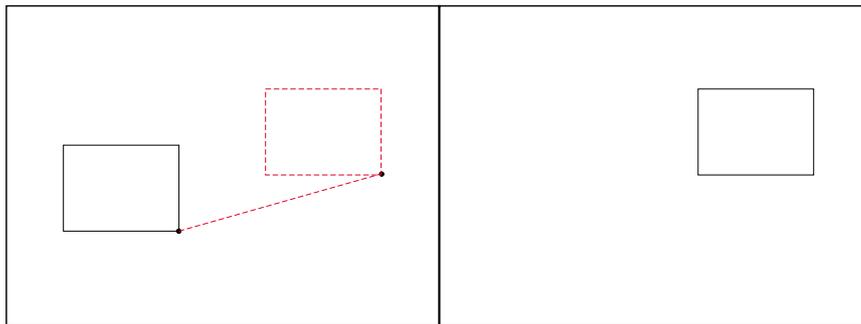
## MOVE



The **Move** command (**Modify > Move**) move drawing objects within the coordinate system.

```
> move
Select objects: [pick]
Select objects: [Enter]
Basepoint: [pick]
Second point of displacement: [pick]
```

Select the objects to be moved. You may select more than one object; they will be moved together in the same relationship as they are currently placed. End object selection by pressing **Enter**.



Position of an object before (left) and after (right) the moving

The next prompt asks for the location of the **Basepoint**. This is the point that you choose as the base from which to begin the move.

Basepoint	Rel.Point				
-----------	-----------	--	--	--	--

If, however, you need to move the objects a specific distance from their current location, select the **Rel.Point** option from the option bar. The prompt asks for distances in the X, Y, and Z directions:

```
Basepoint: Rel.Point  
Relative point dX,dY,dZ: [pick]
```

The third method is to select the objects with the cursor, and drag them to the new location. Remember to use object snap modes to make the move more accurate.

**COPY**

The **Copy** command (**Modify > Copy**) duplicates objects within the coordinate system. 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.

```
> copy
Select objects: [pick]
Select objects: [Enter]
Specify base point or select option (Relative Point,
Multiple): [pick]
Target point: [pick]
```

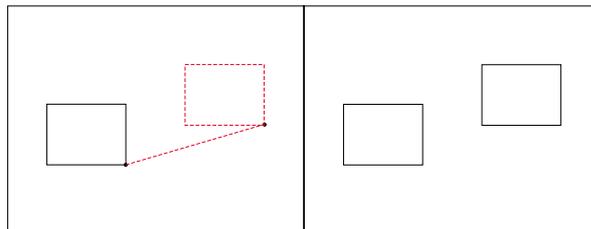
Select the objects to be copied, and end object selection with **Enter**. The option bar displays the **Basepoint** and **Rel.Point** option:

Basepoint	Rel.point				
-----------	-----------	--	--	--	--

The **Basepoint** option prompts you for base point and the target point (the position to which the copy is to move).

The **Rel.Point** option asks for the relative points by specifying the move in the X, Y, and Z directions. (You don't to specify a Z delta.)

```
> COPY
Select objects: P1
Select objects: [Enter]
Basepoint: Rel.Point
Relative point dX,dY,dZ: 2,-3
```



Position of objects before (left) and after (right) the copying

**ROTATE**

The **Rotate** command (**Modify > Rotate**) rotates entities through an angle about an axis.

```
> ROTATE
Select objects: P1
Select objects: [Enter]
Basepoint: P2
Rotation angle: 60
```

Select the objects to be rotated.

Angle	Reference				
-------	-----------	--	--	--	--

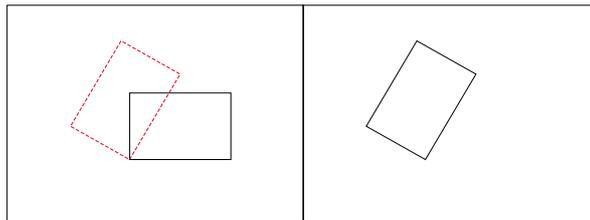
The command prompts for a basepoint, which is the point about which the entities are rotated. We recommend that you pick a corner or the center of an entity, but you may pick any point -- whether on or off of the geometry.

When prompted, enter the rotation angle. Rotation takes place to the specified angle.

As an alternative, you may select a reference angle:

```
> ROTATE
Select objects: P1
Select objects: [Enter]
Basepoint: P2
Rotation angle: Ref.Angle
Reference angle <0>: 44
New angle: 45
```

The **Reference** option asks for a starting angle, then prompts for a new angle to which the object will be rotated. Rotation occurs counterclockwise. As before, rotation occurs about the basepoint.



**SCALE**

The **Scale** command (**Modify > Scale**) changes the size of selected objects. The objects are scaled uniformly in the X and Y directions.

```
> SCALE
Select objects: P1
Select objects: [Enter]
Basepoint: P2
Scale factor: 1.5
```

Select the objects to be scaled.

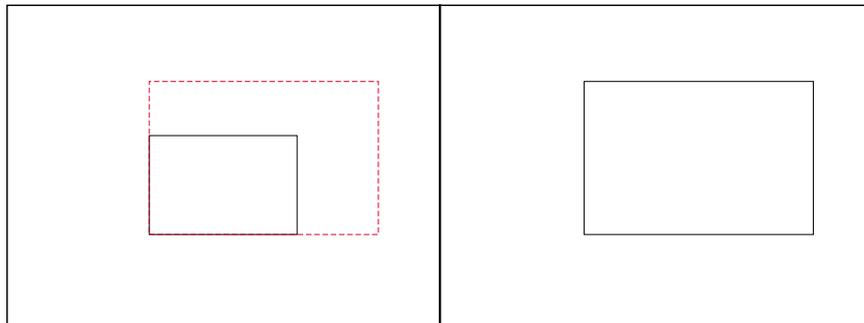
The command prompts for a basepoint, which is the point from which the entities are scaled. We recommend that you pick a corner or the center of an entity, but you may pick any point -- whether on or off of the geometry.

**Note:** If the basepoint is part of the selected object, it keeps its original position during the scaling, while all the other points are scaled by the same relative values in the X and Y directions. When the basepoint is not on the object, the distance from the basepoint to all other points on object are changed by the same scale factor.

The command then displays the options bar:

Sc.Factor	Reference				
-----------	-----------	--	--	--	--

The **Sc.Factor** (short for “scale factor”) option is the default option.



Enter a scale factor:

- A scale factor larger than 1 enlarges the selected objects.
- A scale factor between 0 and 1 reduces the objects by that factor.
- Negative scale factors are not permitted.

For example, a scale factor of 3.0 enlarges the selected objects by three times, while a scale factor of 0.25 reduces the objects to one-quarter of their original size.

The scale factor can be specified by indicating a second point. To do this, move the cursor after the fixing the basepoint. The distance (measured in drawing units) is interpreted as the scaling factor. The advantage is that you see the effect of the scaling while moving the cursor; the disadvantage is that the scale factor is less precise.

### Reference Scale

The **Reference** option allows you to change of the size of an entity without need to first calculate the scale factor. For example, if you need to enlarge an object with a length of 3.45 drawing units up to 6.9 drawing units, you would first calculate the scaling factor ( $6.9 / 3.45 = 2$ ).

The **Reference** option makes this calculation for you. Select the **Reference** option from the option bar. Enter the reference length, and then enter the new length:

```
> SCALE
Select objects: P1
Select objects: [Enter]
Basepoint: P2
Scale factor: Reference
Reference length <1>: 3.45
New length: 6.9
```

**FLIP**

The **Flip** command (**Modify > Flip**) flips selected entities about a specified axis. (This is unlike the **Mirror** command, where the original entities remain in place.) The flip axis can be at an arbitrary angle, or be parallel to the X- or Y-axis.

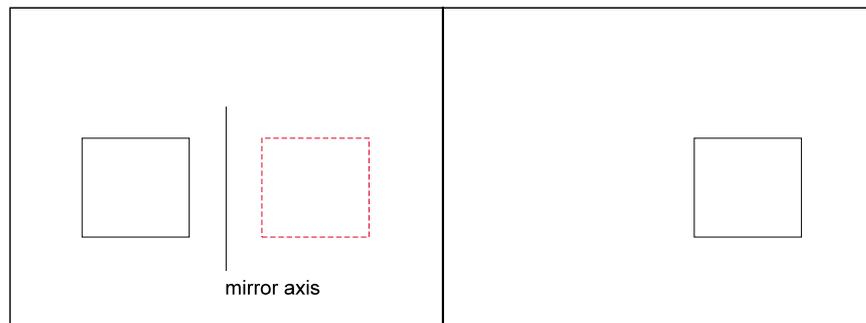
```
> FLIP
Select objects: P1
Select objects: [Enter]
First point of mirror axis: P2
Second point: P3
```

Select the object, then select an axis option from the option bar:

<b>Axis</b>	<b>Horizontal</b>	<b>Vertical</b>			
-------------	-------------------	-----------------	--	--	--

**Axis**

The **Axis** option allows you to specify the flip axis, and is the default option. Indicate two points or type the coordinates. The axis may have any alignment: aligned with the objects, or any of the axes.



If the axis is supposed to run through any points of existing geometry, we recommend that you use object snap options to define the points precisely.

## Horizontal

To flip the marked object about the horizontal axis, choose the **Horizontal** option. The flip axis runs parallel to the X-axis of the coordinate system. A single point is only required to define this axis.

```
> FLIP
Select objects: P1
Select objects: [Enter]
First point of mirror axis: Horizontal
Horizontal reflection point: P2
```

## Vertical

To flip the marked object about the vertical axis, choose the **Vertical** option. The flip axis runs parallel to the Y-axis of the coordinate system. A single point is only required to define this axis.

```
> FLIP
Select objects: P1
Select objects: [Enter]
First point of mirror axis: Vertical
Vertical reflection point: P2
```

## MIRROR



The **Mirror** command (**Modify > Mirror**) makes a mirrored copy of selected entities, and keeps the original entities in place. This is different from the **Flip** command, which erases the original entities.

```
> MIRROR
Select objects: P1
Select objects: [Enter]
First point of mirror axis: P2
Second point: P3
```

Select the object(s). Press Enter to end object selection.

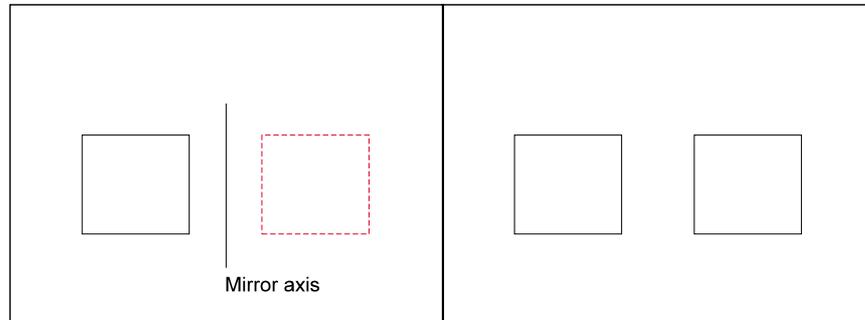
The next prompt asks for the definition of the mirror axis, displaying the options on the options bar.

Axis	Horizontal	Vertical			
------	------------	----------	--	--	--

## Axis

The **Axis** option allows you to specify the mirror axis, and is the default option. Indicate two points or type the coordinates. The axis may have any alignment: aligned with the objects, or any of the axes.

If the axis is supposed to run through any points of existing geometry, we recommend that you use object snap options to define the points precisely.



## Horizontal

To mirror the marked object about the horizontal axis, choose the **Horizontal** option. The mirror axis runs parallel to the X-axis of the coordinate system. A single point is only required to define this axis.

```
> MIRROR
Select objects: P1
Select objects: [Enter]
First point of mirror axis: Horizontal
Horizontal reflection point: P2
```

## Vertical

To mirror the marked object about the vertical axis, choose the **Vertical** option. The mirror axis runs parallel to the Y-axis of the coordinate system. A single point is only required to define this axis.

```
> MIRROR
Select objects: P1
Select objects: [Enter]
First point of mirror axis: Vertical
Vertical reflection point: P2
```

## Mirroring Text

If there is text among the selected objects, it is likely that you do not want that text mirrored. For this reason, FelixCAD does not, by default, mirror text entities.

If you need the text mirrored, use the **EditPar** command (**Modify > Modify Parameters**) to display the **Edit Parameters** dialog box. Click the check box next to **Mirror Text** and click **OK**.

## STRETCH



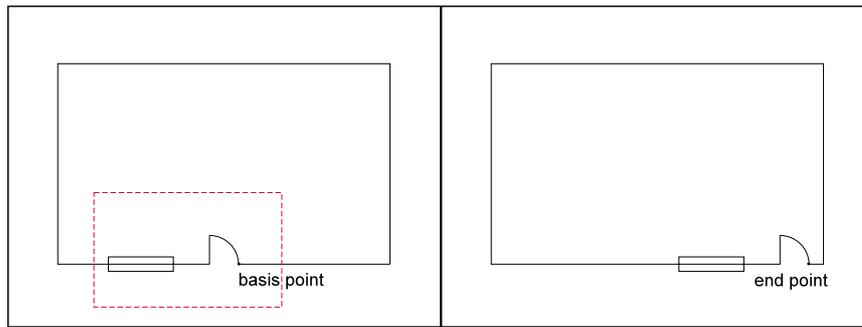
The **Stretch** command (**Modify > Stretch**) stretches entities that lie on the border of the selection rectangle, but moves entities that lie within the selection rectangle. (This command performs like a combination of the **Move** and **Lengthen** commands). Stretching is useful for enlarging or reducing entities in one direction. This is in contrast to the **Scale** command, which scales in both the X and Y directions).

Like other editing commands, you must select objects for stretching. The **Stretch** command, however, is more sensitive in the selection of entities. You must carefully plan which entities will be stretched and which not. Different types of entities may be stretched at the same time. A typical use of this feature would be stretching a hatched object with a dimension. The object, associated hatch and dimension lines would stretch in the desired direction. Additionally, the dimension text is updated automatically to the correct value.

During the command, the *method* of selection has an influence on which entities are moved and which are stretched, according to these rules:

Crossing	Cpolygon	Remove			
----------	----------	--------	--	--	--

- Entities totally enclosed by the crossing window (or polygon) are moved.
- Entities crossing the crossing window (or polygon) are stretched (lengthened).
- All other entities retain their position and shape.



After selecting entities, according to the rules listed above, the command displays these options:

Basepoint	Displacem.				
-----------	------------	--	--	--	--

### Basepoint

The default **Basepoint** option is useful for moving selected entities to a particular position, and for stretching entities on the selection border.

Select the basepoint for the stretch by typing the coordinates or pick the basepoint.

```
> STRETCH
Select objects: P1
Select objects: [Enter]
Basepoint: P2
Second point of displacement: P3
```

Finally, determine the end point of the stretch. The basepoint is moved to the end point.

### Displacement

The **Displacement** option stretches selected entities by a vector (a straight line defining length and distance). Enter the value for the move in the form of X, Y, and (optionally) Z coordinates.

```
> STRETCH
Select objects: P1
Select objects: [Enter]
Basepoint: Displace.
Displacement dX,dY,dZ: 2,-3
```

## ARRAY

The **Array** command (**Modify > Array**) creates multiple copies of selected entities at random or in a pattern. The pattern may be in horizontal rows and vertical columns, in a circle, or in an arc. In addition, the entities can be rotated when arrayed in the circular or arc patterns.

After entering the command and selecting the entities, select an option:

Individual	Rectangular	Circ.NRot	Ci.Rot		
------------	-------------	-----------	--------	--	--

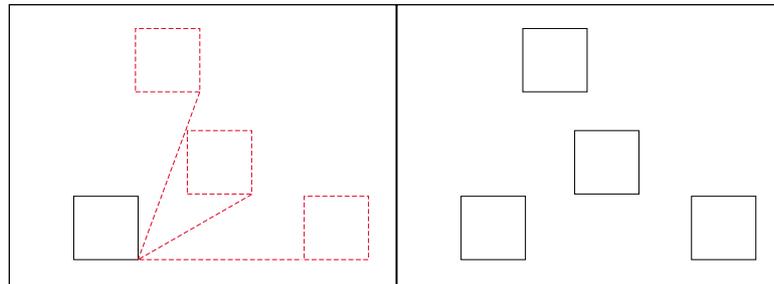
### Individual



The **Individual** option copies the selected entities repeatedly, just as if you were to use the **Copy** command over and over. This option is the default for the **Array** command.

```
> ARRAY
Individual> [Enter]
Select objects: P1
Select objects: [Enter]
Basepoint: P2
Target point: P3
Target point: P4
Target point: [Enter]
```

Select the basepoint. A discreet point on the geometry (such as an edge, corner, or intersecting points) is often more useful.



Type the coordinates of the target point, or pick the points for placement of the copies. The command creates a copy of the selected entities at each target point.

Until you terminate the command, you keep placing a copy, as required. Terminate the command by pressing **Esc** or **Enter**.

## Rectangular

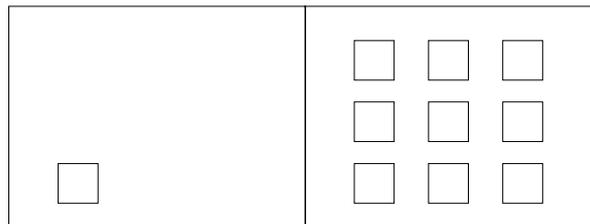


The **Rectangular** option arranges the array in rows and/or columns, called a *rectangular array*.

```
> array
Individual> Rectang
Select objects: [pick]
Select objects: [Enter]
_RECTANG
No. of columns: 2
No. of rows: 3
Distance between columns: 4
Distance between rows: 5
```

Start the **Array** command, then select **Rectang** from the option bar. After selecting the entities to array, specify the number of the columns and rows. Finally, define the distance between the columns and the rows.

**Note:** The rows and columns are normally array up and to the left. To array the entities down and to the right, use a negative number for the distance.



## Circ.NRot & Ci.Rot



The **Circ.NRot** and **Ci.Rot** options copy the selected entities in a circular path, called a *circular array*. The copying can be around a full 360-degree circle, or a partial circle (an arc).

The **Circ.NRot** option creates a circular arc without rotating the selected entity. The **Ci.Rot** option rotates the selected entities so that they always “face the center.”

Both options prompt you for the following parameters:

```
> ARRAY
Individual> Circ.NRot or Ci.Rot
Select objects: P1
Select objects: [Enter]
Center: P2
Number of entities: 4
Angle to fulfill <360>: [Enter]
```

- **Center:** the center point of the circular array.
- **Number of entities:** the number of times the Array is to reproduce the selected entities.
- **Angle to fulfill:** the distance the array should go around the circle; 360, if the array is to be a full circle.

When creating a circular array, the command uses a reference point on which to position the entities on the circumference of the circle, as shown in the following table:

Entity	Reference point
Arc	Center of arc.
Circle	Center of circle.
Line	Start or end point.
Part	Insertion point.
Point	Insertion point.
Rectang	Start point.
Text	Insertion point.

For example, when you array a circle entity (to represent bolt holes around a pipe flange, for example), the **Array** command arrays the circle at its center point.

If you wish to change this reference point, the Basepoint option lets you define an alternate reference point on the entity to be array. Remember to use an object snap mode with this option.

### Array Without Rotation

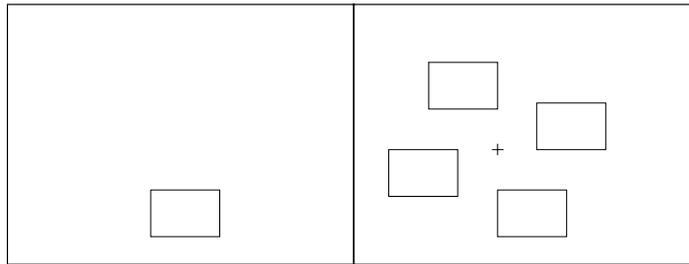


The **Circ.NRot** option (short for “circular, no rotation”) arranges the entities around a circle (or arc) path without changing the alignment of the entity.

```
> Array
Individual> Circ.NRot
Select objects: P1
Select objects: [Enter]
Center: P2
Number of entities: 4
Angle to fulfill <360>: [Enter]
Angle between entities: 60
```

Select the objects for the array, choose the **Circ.NRot** 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 arrayed object to be reproduced (measured **between** the reference points of the objects).



**Note:** Circular arrays are constructed *counterclockwise* from the starting point. For a clockwise rotation, enter negative values for the angle.

### Array With Rotation

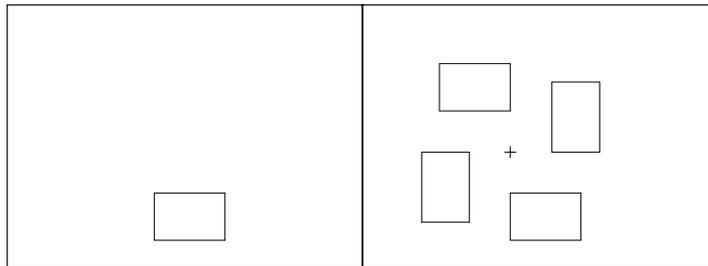


The **Ci.Rot** option (short for “circular, rotated”) rotates the arrayed entities when placing them along the circular path. The entities are rotated in the same direction as the circle describing the path. The rotation angle corresponds to the angle between the copies (measured between the reference points of the arrayed entities).

```
> ARRAY
Individual> Ci.NRot
Select objects: P1
Select objects: [Enter]
Center: P2
Number of entities: 4
Angle to fulfill <360>: [Enter]
```

After starting the **Array** command, select the entities, and choose the **Ci.Rot** option 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.

**OFFSET THROUGH POINT**

The **Offset** command (**Modify > Offset Through Point**) creates parallel copies of lines, circles, arcs, and 2D polylines. The command copies the selected entity and places it a specific distance from the original. The original entity stays in place.

**Note:** The **Offset** command can create unexpected results, depending on the entity selection:

- Line and straight 2D polyline: an identical copy is created, to make parallel lines.
- Circle and arcs: an copy is created, but is scaled by a factor to make concentric curves. The center of the original circle or arc is used as a basepoint for scaling.
- 3D polyline made of line and arc segments: due to the arc portions being scaled, the offset polyline may look very different from the original.

**Lines & 2D Polylines**

Start the **Offset** command, and select the line(s) or polyline(s) to be offset.

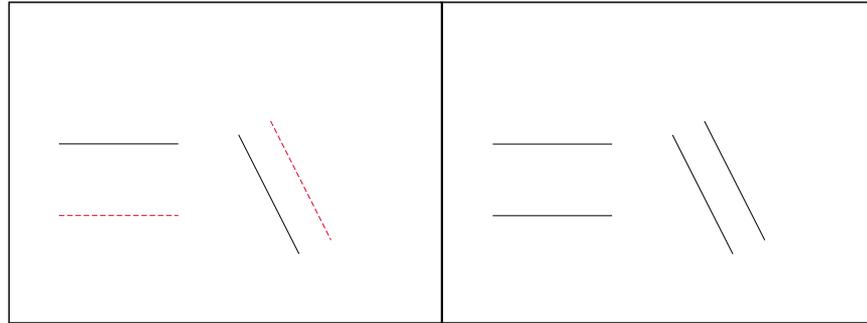
There are two ways to specify the offset distance. One method is to pick a point, using object snaps if necessary. FelixCAD offsets the copy at the pick location.

```
> OFFSET
Select objects (line, circle, arc, 2D-polyline): P1
Offset by point: P2
Select objects (line, circle, arc, 2D-polyline): [Enter]
```

The second method is to select the **Distance** option. Enter the offset distance. You are then prompt “Offset to Side?”. Pick a point on either side of the original entity.

```
> 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): [Enter]
```

In both cases, the **Offset** command continues, until you press **Enter** or **Esc**.

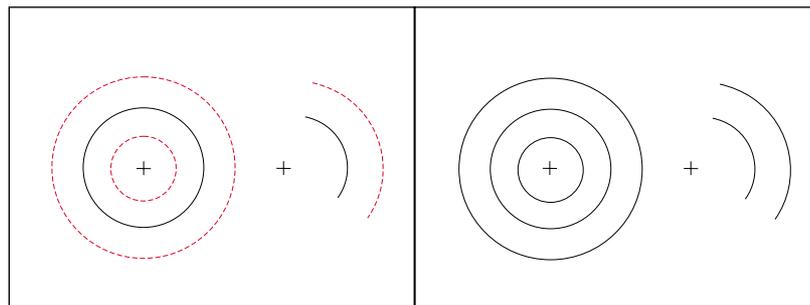


### Circles & Arcs

The **Offset** command creates concentric circles and arcs with a small or larger diameter than the original entity.

After selecting the objects to be offset, the position of the offset is determined by:

- Picking a point.
- Typing the coordinates of a point on the circle or arc.
- Using the **Distance** option.



```
> OFFSET
Select objects (line, circle, arc, 2D-polyline): P1
Offset by point: P2
Select objects (line, circle, arc, 2D-polyline): [Enter]
```

**Distance**

The **Distance** option 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 through point: Distance
Distance <5>: 10
Second point P3
Offset to side ? P4
Selection of an entity
(Line, circle, arc, 2D-Polyline): [Enter]
```

**Both Sides**

You can offset an entity to both sides by selecting the **BothSides** option when you are asked to specify the side to offset:

```
> OFFSET
Select entity (line, circle, arc, 2D-polyline): [pick]
Offset through point: Distance
Distance <0.98> [pick]
Second point: [pick]
Side to offset ? BothSides
Select entity (line, circle, arc, 2D-polyline): [Enter]
```

## OFFSET DISTANCE

The **DOffset** command (**Modify > Offset Distance**) creates lines and curves parallel to an existing entity.

The **DOffset** command is similar to the **Offset** command, but prompts you for the offset distance *before* you select entities. (This command has been added for convenience of users familiar with other CAD programs.)

```
> DOffset
Distance or select option ThruPoint <1.00>: 0.30
Select entity (line, circle, arc, 2D-polyline): [pick]
Side to offset ? [pick]
Select entity (line, circle, arc, 2D-polyline): [Enter]
```

## Through

To indicate a *through* point (such as an significant point in the geometry), select the **ThruPoint** option (or type T, for short):

```
> DOffset
Distance or select option ThruPoint <0.30>: T
Select entity (line, circle, arc, 2D-polyline): [pick]
Offset through point: [pick]
Select entity (line, circle, arc, 2D-polyline): [Enter]
```

## BothSides

You can offset an entity to both sides by selecting the **BothSides** option when you are asked to specify the side to offset:

```
> DOffset
Distance or select option ThruPoint < 0.50 > : [Enter]
Next point: [pick]
Select entity (line, circle, arc, 2D-polyline): [pick]
Side to offset ? BothSides
Select entity (line, circle, arc, 2D-polyline): [Enter]
Side to offset ?
```

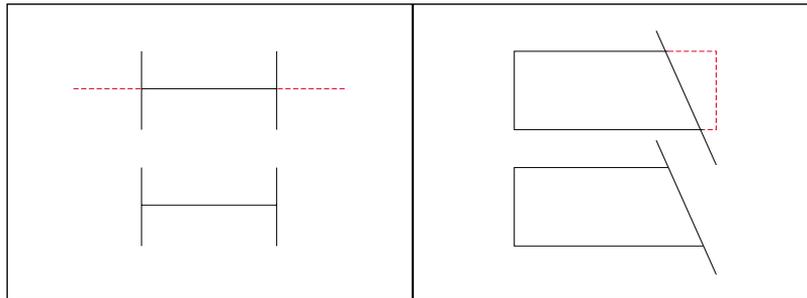
**TRIM**

The **Trim** command (**Modify > Trim**) cuts off entities that cross a “cutting edge.” The cutting edge is the intersection with another entity. The entities are trimmed on the side that you pick. The command keeps repeating until you end it with the **Esc** key.

```
> TRIM
Select entities for intersecting edges
(Line, circle, arc, 2D-Polyline): P1, P2
Select entities for intersecting edges
(Line, circle, arc, 2D-Polyline): [Enter]
Select entity for trimming
(Line, circle, arc, 2D-Polyline): P3, P4
Select entity for trimming
(Line, circle, arc, 2D-Polyline): [Esc]
```

Start the **Trim** command and select one or more entities that will become the intersection edge.

The next prompt asks you to select the entity that you want to trim. The selection is sensitive to location: the point you pick is the portion that will be trimmed away. As entities are selected, they are trimmed; the trimmed portion disappears from the drawing.



*Note: Some objects will not trim other entities, most notably Parts or Blocks.*

**EXPAND**

The **Expand** command (**Modify > Expand**) lengthens entities to a *boundary edge*. The boundary edge is another entity, such as a line, arc, circle, or 3D polyline. Use this command to close polygons, and to extend entities to the proper boundary edge. *Note: AutoCAD users know this command as EXTEND.*

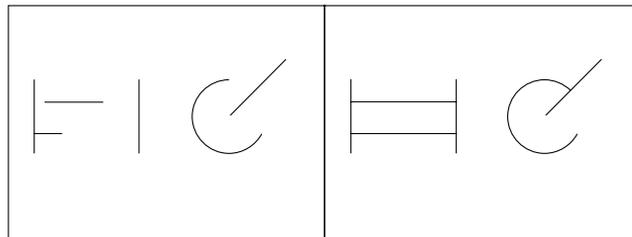
**Note:** the **Expand** command does not work with “closed” objects, such as circles. Entities to be expanded are lengthened. They do not change direction. A line remains a line; an arc remains an arc.

Start the **Expand** command, then select the entities that will act as the boundary edge. You may select one or more lines, circles, arcs, and 2D polylines. Press **Enter** to end the boundary edge selection.

```
> EXPAND
Select entities with boundary edges (line, circle, arc,
2D-polyline): P1, P2
Select entities with boundary edges (line, circle, arc,
2D-polyline): [Enter]
Select entity for expanding (line, circle, arc,
2D-polyline): P3
Select entity for expanding (line, circle, arc,
2D-polyline): [Enter]
```

Then select the entities to be expanded. This is sensitive to your pick location. For example, an arc is expanded in the direction of your pick location. The expansion of selected entities is always executed from the endpoint that lies nearest to the cursor when it selected the entity. When more than one boundary edge has been selected, the entity is expanded to the first boundary edge first. Picking a second time results in the expansion continuing to the second boundary edge.

*Note: Some objects cannot be used to expand to other entities, most notably Parts or Blocks.*



**FILLET**

The **Fillet** command (**Modify > Fillet**) rounds the intersection of two lines. These can also be lines of an object such as a rectangle. The **Fillet** command treats 2D polylines differently from lines: every vertex of the polyline is filleted. The command cannot fillet curved objects, such as arcs.

When the fillet radius is 0.9 (the default), this command precisely cleans up the intersection of two lines.

The **Fillet** command prompts you to select the two lines to be filleted.

```
> FILLET
Select first line: P1
Second line: P2
```

Then these options are shown in the options bar:

Polyline	Radius	No_Trim			
----------	--------	---------	--	--	--

**Polyline**

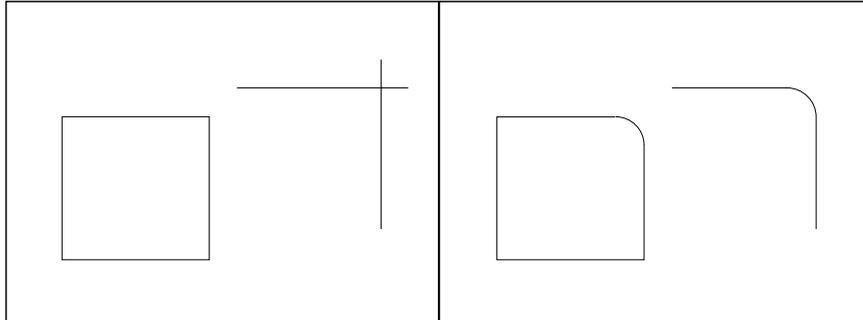
The **Polyline** option fillets a single polyline. Every vertex of the polyline is given the fillet radius.

```
> FILLET
Select first entity: Polyline
Select 2D-polyline: P1
```

## Radius

The **Radius** option allows you to define the radius for the fillet. By default, the radius is 0.0. When you change the radius, FelixCAD remembers it for the next command operation.

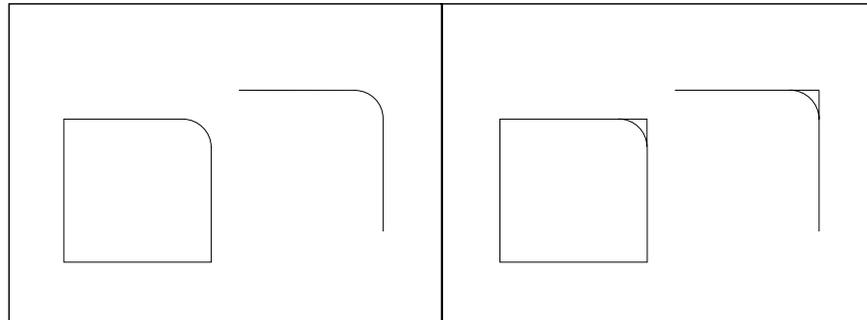
```
> Fillet
Select first line: Radius
Filleting radius <0.0000>: .75
Select first line: P1
Second line: P2
```



## No\_Trim

The **Trim** option functions as a switch: it decides whether or not the lines beyond the fillet are erased. When the options bar displays **Trim**, you switch *off* the trimming by clicking the word; it then changes to **No\_Trim**.

The following figure shows the effect of the **No\_Trim** switch.



**Trim** option switched on (left) and switched off (right).

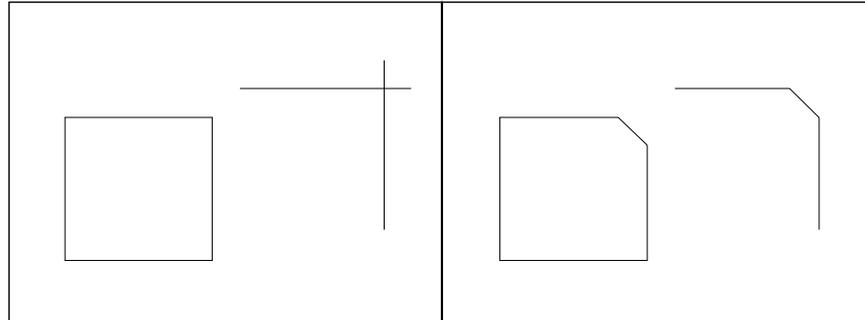
## CHAMFER



The **Chamfer** command (**Modify > Chamfer**) creates a beveled corner between two intersecting lines.

```
> CHAMFER
Select first line: P1
Second line: P2
```

Select the two entities to be chamfered by picking them with the cursor.



Polyline	Distance	No_Trim			
----------	----------	---------	--	--	--

### Distance

The **Distance** option allows you to specify the two chamfering distances.

```
> CHAMFER
Select first line: Distance
First chamfering distance <1.0000>: [Enter]
Second chamfering distance <0.5000>: 1.000
Select first line: P1
Second line: P2
```

### No\_Trim

The **No\_Trim** option functions in exactly the same manner as described with the **Fillet** command.

### Polyline

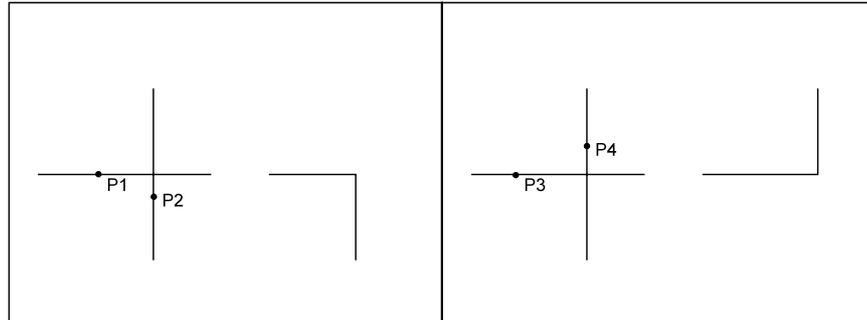
The **Polyline** option functions in exactly the same manner as described with the **Fillet** command.

**INTERSECT**

The **Intersect** command (**Modify > Intersect**) is a special case of the **Fillet** command, with the fillet radius always set to 0. This makes the command useful for trimming intersecting lines to be trimmed.

```
> INTERSECT  
Select first line: P1  
Second line: P2
```

Select the two lines to be trimmed. Be sure to make the pick on the segment of each line that is to remain.



**LENGTHEN**

The **Lengthen** command (**Modify > Lengthen**) changes the length of open entities, such as lines and arcs. Unlike the **Expand** command, **Lengthen** works without a boundary edge; it extends an entity by a specific value or ratio (percentage value) of the original size.

Although the command is called “Lengthen,” it may be used to lengthen or shorten entities. Entities are changed in length, not scaled. Entities are always lengthened at the end laying closest to the cursor pick point.

```
> lengthen
Select entity (line, arc): [pick]
Current Length <7.11>: Dynamic:
```

After selecting the line or arc to lengthen, the command reports the entity’s current length. The **Dynamic** option is the default; the option bar displays other options:

Total	Percent	Relative			Angle
-------	---------	----------	--	--	-------

**Dynamic**

The **Dynamic** option is the default option of the **Lengthen** command. You drag the line or arc to its new length. When you click with the cursor, the new length is fixed.

```
> LENGTHEN
Select entity (line, arc): P1
Current length <100.00>: Dynamic P2
Select entity (Line, Arc): [Enter]
```

The **Lengthen** command continues until you press **Enter** or **Esc**.

**Total**

The **Total** option changes the length to an absolute distance. The change is applied to 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
Select entity (Line, Arc): [Enter]
```

## Percent

The **Percent** option allows you to specify the new length by a percentage value. This percentage value relates to either line length or the circumference of an arc. For example, specifying **200** doubles the length of the line or the arc, while the value of **50** shortens the entity to half its original length.

```
> LENGTHEN
Select entity (line, arc): P1
Current length <100.00>: Percent
Percent: 60
Select entity (Line, Arc): [Enter]
```

## Relative

The **Relative** option lengthens a line or arc by a value defined by the distance between two points. This option is useful when the entity must be lengthened by reference to another object. (If the **Angle** option has been selected, you 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
Select entity (Line, Arc): [Enter]
```

## Angle

To lengthen arcs, use the option **Angle**. This option functions like a switch. Selecting it from the option bar causes all of the following options to display a prompt for the included angle of the arc.

```
> lengthen
Select entity (line, arc): [pick]
Current Length <9.32>: Dynamic: Angle
Included angle of arc <205.90>: Dynamic: [pick]
Select entity (line, arc): [Enter]
```

**DELETE PARTIAL**

The **DelPartial** command (**Modify > Delete partial**) erases a portion of a line, 2D polyline, circle, or arc. This command is useful for dividing an entity: the result is two independent entities, which can be edited or operated on separately.

Whereas the **Trim** command cuts off the end of open entities, the **DelPartial** command cuts out their middles.

2 Points	1 Point				
----------	---------	--	--	--	--

**2Point**

The **2Point** option is the default. It requires that you pick two points:

```
> DELPARTIAL
Select object (line, circle, arc, 2D-polyline): P1
First point: P2
Second point: P3
```

The part is erased between points P1 and P2. The line is divided at this point; the portion of the line between the two points is deleted.

When the entity is a circle, it becomes an arc. When a 2D polyline has a width greater than zero, it is cut at right angle to the polyline.

**1Point**

To separate or break an entity, choose the **1Point** option.

```
> DELPARTIAL
Select entity (line, circle, arc, 2D-polyline): P1
First point: 1P
Break point: int
of object: P2
```

This option is useful for breaking an entity into two pieces, for example, at its intersection with another object. To do this:

1. Select the **1Point** option (or type **1P** for short) at the “First point:” prompt.
2. Select the break point. It may be wise to use the object snap options to increase the precision of the input.

You achieve the same result when you type @ at the “Second point:” prompt. The @ symbol signifies the last point entered.

```
> DELPARTIAL  
Select entity (line, circle, arc, 2D-polyline): [pick]  
First point: int  
of object: P1  
Second point: @
```

In both cases, the entity will look the same as before -- until you select one part of the divided entity.

**Notes:** You cannot break a circle with the **1Point** option, since that would create an arc of 360 degrees.

You can use the **DelPartial** command to shorten an entity. Here's how: specify the first point on the entity where you want the new endpoint to be. Then pick the other point pass its endpoint. The entity is cut off at the point you picked on the object.

## **BREAK**

The **Break** command (**Modify > Break**) breaks objects at a point or at points you select, just like the **DelPartial** command.

```
> BREAK  
Select entity (line, circle, arc, 2D-polyline): [pick]  
Second point (or F for first point): [pick]
```

The **Break** command has been added for convenience of those familiar with other CAD programs. It is similar but not identical to the **DelPartial** command. The primary difference is that the **Break** command assumes that you might want to use the pick point when selecting the entity as the first point of the break.

You can, however, respond to the “Second point:” prompt with the **First** option (or type **F** for short). This explicitly specifies both break points independent of the entity selection:

```
> BREAK  
Select entity (line, circle, arc, 2D-polyline): [pick]  
Second point (or F for first point): F  
First point: [pick]  
Second point: [pick]
```

You have the choice to break the entity into two pieces without erasing any part of it. To do this: specify the same point for both the first and the second points. Type **@** at the request for the second point, which signifies the previous point entered. This is useful when you want to break an entity at the intersection with another entity:

```
> BREAK  
Select entity (line, circle, arc, 2D-polyline): [pick]  
Second point (or F for first point): F  
First point: int  
of object: [pick]  
Second point: @
```

**REJOIN**

The **Rejoin** command (**Modify > Rejoin**) unites separate line segments, which are aligned in the same direction. It also joins arc segments that share the same center and the radius. This function is useful for reuniting line and arc segments that have been divided by the **DelPartial** command.

Select the command, then pick the entities to be rejoined:

```
> REJOIN
Select first entity (line, arc): P1
Select second entity (line, arc): P2
```

**CHANGE**

The **Change** command (**Modify > Change**) edits the geometric character and the properties of many entities, including lines, circles, arcs, text, attributes, and part. The properties or geometry that can be changed depend on the entity.

**Note:** To change the layer, color, or linetype properties, you may find it more useful to use the **PropEdit** Command (**Edit > Modify Properties**).

```
> CHANGE
Select entity (line, circle, arc, text, attribute definition,
block insertion): P1
Select entity (line, circle, arc, text, attribute definition,
block insertion): [Enter]
Modification point: P2
```

Select the entity to be changed. You may preview the change by selecting the **Dragmode** option from the options bar.

Dragmode					
----------	--	--	--	--	--

Since the properties changed depend on the entity selected, the following sections describe the changed by entity.

## Line

The **Change** command changes one end point of the line. You can move 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 that 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.

## Circle

The **Change** command alters the diameter (or radius) of a circle. Select the circle, then determine the new diameter by picking a point or typing an X, Y coordinate.

```
> CHANGE
Select entity (line, circle, arc, text, attribute definition,
block insertion): [pick a circle]
Select entity (line, circle, arc, text, attribute definition,
block insertion): [Enter]
Modification point: P2
```

## Arc

The **Change** command alters the diameter (or radius) of an arc. Select the arc, then determine the new diameter by picking a point or typing an X, Y coordinate.

```
> CHANGE
Select entity (line, circle, arc, text, attribute definition,
block insertion): : [pick an arc]
Select entity (line, circle, arc, text, attribute definition,
block insertion): [Enter]
Modification point: P2
```

## Text

The Change command changes the following properties of a text entity:

These text attributes, or parameters, may be changed. Existing values are displayed as parameters and can be taken over by pressing **Enter**.

- **Modification point:** pick (or enter the coordinates) a new insertion point for the text.
- **Text style:** enter the name of the new text style.
- **Text height:** define the new height by typing a numeric value or by picking a new height. Dragmode is automatically activated during this option.
- **Rotation angle:** change the rotation angle by picking a point or type a number. Dragmode is automatically activated during the change of this attribute.
- **Text:** type different text, which replaced the entire selected text.

```
> CHANGE
Select entity (line, circle, arc, text, attribute definition,
block insertion): [pick text]
Select entity (line, circle, arc, text, attribute definition,
block insertion): [Enter]
Modification point: P2
Text style <STANDARD> Normal
Height <2>: 1
Rotation angle <0>: 45
Text <first floor>: 1st floor
```

## Attribute Definition

The **Change** command changes the following properties of attributes (as for text):

- Inserting point.
- Text style.
- Text height.
- Rotation angle.

As well as the these specific properties of attributes:

- **Name:** determines the name of the attribute; enter a new name.
- **Request:** enter a new line of text for the attribute prompt.
- **Parameter value:** the constant or variable value of the attribute; enter a new value (a sequence of characters or numbers).

You find more information about attributes in “Parts and Attributes, XRefs, & Groups.”

```
> CHANGE
Select entity (line, circle, arc, text, attribute definition,
block insertion): P1
Select entity (line, circle, arc, text, attribute definition,
block insertion): [Enter]
Modification point: P2
Text style <STANDARD> Normal
Height <2>: 1
Rotation angle <0>: 45
Text <first floor>: 1st floor
Name <Att003>: [Enter]
Request <Index No.>: current number
Parameter value <1.01>: 2.01
```

## Part

- The **Change** command allows you to change the insertion point and the rotation angle of a part. 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): [Enter]
Modification point: P2
Rotation angle <0>: 90
```

**POLYLINE EDITING**

FelixCAD allows you to edit polylines by the following methods:

- **Chamfer:** chamfers the polyline vertices.
- **Fillet:** fillets the polyline vertices.
- **PolyEdit:** edits the geometric properties of a polyline, such as its width.
- **VtxEdit:** edits the vertices of the polyline.



**Modify Polyline**



The **PolyEdit** command (**Modify > Edit Polyline > Edit Entire Polyline**) allows you to:

- Open and close the polyline.
- Convert lines and arcs into a polylines.
- Join the polyline with other drawing entities, such as lines, arcs, and other polylines.
- Change the width of the whole polyline.
- Undo these changes.

**Transform to Polyline**

When you select a line or arc with the **PolyEdit** command, you are asked if you wish to transform it into a polyline:

```

> polyedit
Select a 2D polyline: [pick]
Transform arc to polyline ?
  
```

Yes	No				
-----	----	--	--	--	--

Select **Yes** or **No** from the option bar. If you answer **No**, the **PolyEdit** command prompts you to select another 2D polyline.

## Open or Close

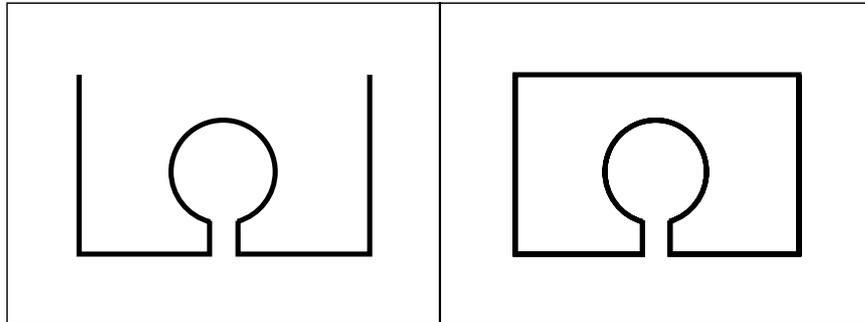
After selecting the polyline, one of two option bars are displayed. This **Open** option is displayed if the polyline is closed:

Open	Join	Width			
------	------	-------	--	--	--

The **Open** option removes the polyline segment between the start and end points of the polyline. Then, since the polyline is now open, the **Close** option is displayed on the option bar.

Close	Join	Width	Back		
-------	------	-------	------	--	--

The **Close** option joins the start and end points of the polyline with a straight polyline segment. The width of this segment is equal to the ending width of the end point of the last drawn polyline. (The **Back** option, shown above, is displayed after one operation has been executed during this command).



Open (left) and closed polyline

```
> POLYEDIT
Selection of a 2D-Polyline: P1
Select option: Close
Select option: [Enter]
```

## Join

The **Join** option joins a line, arc, or another polyline to the selected polyline. The prerequisite is that the polyline and entity to be joined must meet at the same end point; there cannot be a gap between the two.

```
> POLYEDIT
Select 2D-Polyline: P1
Select option: Join
Selection of an entity (line, arc, 2D-polyline): P2
Selection of an entity (line, arc, 2D-polyline): [Enter]
```

The width of the new polyline segment depends on the nature of the entity joined to the polyline. If a line or arc is joined to the polyline, the new polyline segment has the same width as the polyline segment's end at which the line or the arc has been joined.

If it is a polyline that was 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

The **Width** option changes the width of the entire polyline. Polylines with have different starting and ending width are made to a single, uniform width for all segments.

```
> POLYEDIT
Select a 2D-polyline: P1
Select option: Width
New width of the polyline <1.5000>: 2
Select option: [Enter]
```

## Undo

The **Undo** option undoes the last operation in this command. The **PolyEdit** command continues, and you see the "Select option:" prompt again.

```
> polyedit
Select a 2D polyline: [pick]
Select option: Close
Select option: u
Select option:
```

## Vertex Editing



The **VtxEdit** command (**Modify > Edit Polyline > Vertex Editing**) edits individual segments of a polyline. This command allows you to change the vertices of the polyline:

- Insert a vertex.
- Delete a vertex.
- Move a vertex.
- Change the width of a polyline segment.

```
> vtxedit
Select a 2D polyline: [pick]
Next:
```

Next	Previous	Insert	Move	Delete	Width
------	----------	--------	------	--------	-------

### Next and Previous

Select the polyline containing the vertex to be edited. When you select the polyline, the first vertex is highlighted with a large cyan (light blue) X marker.

You select another vertex with the **Next** and **Previous** options. After selecting the vertex, select an editing option from the options bar:

### Insert

The **Insert** option allows you to add a vertex, which also add another segment to the polyline. Select the option, then move the market to the position of the additional vertex.

```
> VTXEDIT
Select a 2D Polyline: P1
Next: [Enter]
Next: Insert
New control point: P2
Next: [Esc]
```

The width of this new segment corresponds to the value of the width of the polyline at the bordering control points before the addition of the new segment.

After inserting a segment, the **VtxEdit** command stays active. You may select another option or terminate the command by pressing **Esc**.

## Move

The **Move** option relocates a vertex of the polyline to a new position.

```
> VTXEDIT
Select a 2D Polyline: P1
Next: [Enter]
Next: Move
New Position: P2
Next: [Esc]
```

First, determine the vertex to be relocated, then define the new position of the chosen vertex. The vertex is moved to this position. Other segments of the polyline will remain unchanged.

After moving a vertex, the **VtxEdit** command stays active. You may select another option or terminate the command by pressing **Esc**.

## Delete

The **Delete** option erases a vertex from the polyline.

```
> VTXEDIT
Selection of a 2D Polyline: P1
Next: [Enter]
Next: Delete
Next: [Esc]
```

Move the marker to the vertex to be deleted, then select the **Delete** option from the options bar.

The segment that has its starting point at the selected vertex will be deleted. The previous (in order of drawing) segment is lengthened to the next vertex.

After deleting a vertex, the **VtxEdit** command stays active. You may select another option or terminate the command by pressing **Esc**.

## Width

The **Width** option changes the width of a single segment of a polyline. Move the marker to the vertex to be modified, then select the **Width** option from the options bar. Type a width or select the desired width of the polyline segment. The width of the selected segment will be changed according to the input.

```
> VTXEDIT
Select a 2D Polyline: P1
Next: [Enter]
Next: Width
Starting width <0.50>: 1
Ending width <1.00>: 2
Next: [Esc]
```

After changing the width of a segment, the **VtxEdit** command stays active. You may select another option or terminate the command by pressing **Esc**.

## Polyline to B-Spline

The **Spline** command (**Modify > Edit Polyline > Polyline to B-Spline**) fits a B-spline curve to a polyline. The curve is not forced to pass exactly through the polyline's vertices (as does the **PCurve** command, described below) but produces a smoother curve fit.

```
> SPLINE
Select polyline to alter to B-spline: P1
Number of segments for approximation (2 to 1024) <8>:
```

8					
---	--	--	--	--	--

You turn the splined polyline back to its original form using the **Spline** command with the **Decurve** option (**Modify > Edit Polyline > Decurve B-Spline**).

## 2-D Polyline to Curved Polyline

The **PCurve** command (**Modify > Edit Polyline > 2D-Polyline to Curved Polyline**) fits a curve to the chosen polyline. The curve is forced to pass exactly through the polyline vertexes. 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>:
```

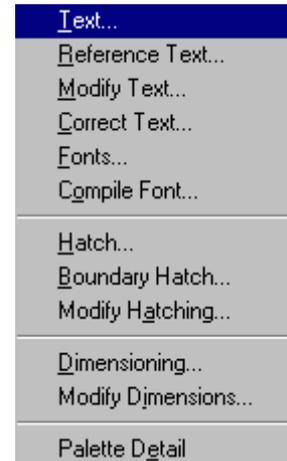
You turn the curved polyline back to its original form using the **Spline** command with the **Decurve** option (**Modify > Edit Polyline > Decurve Curved Polyline**).

Decurve					
---------	--	--	--	--	--



## Text Objects

The commands described in this chapter are used for inserting text into drawings, as well as for changing text entities. Text provides labeling, creates legends, places descriptive notes, and other written elements in a drawing. The commands relating to text are found in the **Detail** menu.



### Properties of a Text Entity

A text entity is defined by properties assigned during the creation of the text. Any of them may be changed after the creation of the original entity. The properties are:

**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.

True Type fonts always appear filled in your drawing and in your plot.

To increase the speed and performance of True Type fonts, the Windows operating system draws some True Type text directly. Due to limitations in Windows, however, FelixCAD must draw True Type text that is transformed in certain ways; for instance, text that is inclined, upside-down, backward or has a width factor not equal to 1.0.

Text that has been transformed might appear slightly more bold at the screen, especially at lower resolutions but this appearance does not affect the plotted output.

If a drawing has been opened that contains True Type fonts that are not recognized by Windows system, FelixCAD substitutes these fonts by the first found True Type font that is assigned to a font file.

- **Character string:** represents the content of a text object. All characters, numbers, and special characters may be used for this string;
- **Font** (or typeface): FelixCAD includes a set of fonts that can be used in creating text objects. Font files are marked by the extension **.shx** concerning AutoCAD's own compiled shape fonts, **.fsh** concerning FelixCAD's own compiled shape fonts; **.ttf** **True Type** fonts.
- **Text size:** height of characters.
- **Insertion point:** determines the position of the text in the drawing. The insertion point is defined by coordinate input or by selecting the point with the cursor.
- **Alignment** (or justification): determines the horizontal and vertical position of the text relative to its insertion point. The character string can run right or left from the insertion point, or the insertion point may be centered on the text. The vertical alignment of the text objects may be above, center, baseline, or below. Any text object can be rotated during insertion.

All of the parameters discussed above are set in the **New Text Font** dialog box.

## Fonts

Fonts define the shapes of the text characters that make up each character set. In FelixCAD, you can use:

FelixCAD's own compiled shape (FSH) fonts,  
AutoCAD's own compiled shape (SHX) fonts  
True Type (TTF) 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.

The **Fonts** command (**Detail > Fonts**) allows you to load fonts from font files and customize the fonts to the requirements of the drawing. Customizing a font is called a *text style*. The **Fonts** command displays the **Text Styles** dialog box, which lets you select an existing style or create a new style.



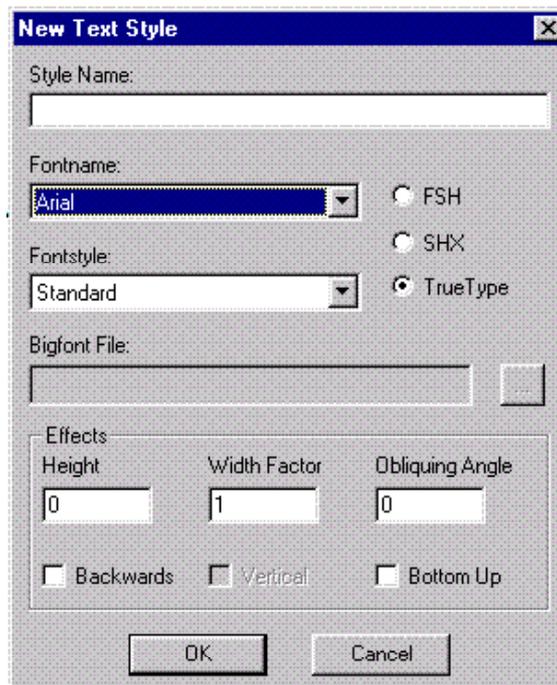
## Adding and Changing Text Styles

Selecting an existing text style, making changes to it, or adding a new style are all done in the same way and are, therefore, discussed in this one section.

### Add a New Text Style

To add a new type style to the drawing:

1. Use the **Fonts** command to open the **Text Style** dialog box.
2. Click **New**. Notice the **New Text Style** dialog box.
3. Define the parameters of the new text style and click **OK**.



SHX fonts and FSH fonts are listed by the name of the file in which the fonts are stored. All installed True Type fonts, provided with your Windows system

are listed by font family name. So you can assign a font to a text style by selecting either a True Type typeface name and its font style (Standard ,Bold or Cursive, for instance) .or a SHX respective FSH file.

If the font file is located in another folder, click the ... (find) button. The **Open Font File** dialog box opens. Select the font file from the other drive or folder.

### Changing Font Properties

After the new font has been selected, its properties can be edited. Click the **Modify** button to display the **New Text Style** dialog box

- **Height** (text size): expressed in drawing units. Entering a numeric value for **Height** gives the font a *fixed* height, which can no longer be changed when creating the text using this style. If you think that a change in the font height might be required, keep the height parameter set at 0 (the default).; you may change the font height later in the **Text** dialog box.
- **Width factor**: controls the width of the characters in relation to the height. The default of 1.0 is normal width; a smaller value makes the text thinner; a larger value spreads out the text. This ratio is sometimes called the *aspect ratio* .
- **Obliquing angle**: defines the inclination of the characters in relation to the horizontal (X-axis). The default, 0, produces upright characters. Values up to 180 (degrees) 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**: produces a mirror-image display of the text.
- **Vertical**: produces characters aligned vertically.
- **Bottom Up**: produces characters displayed upside down.

**Note:** Some fonts do not support all of these options. For instance, some fonts can be aligned vertically, but others cannot. Other fonts do not permit a change to the width factor.

It is possible to all 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 **Fonts.Tab** file in the `\cfg` directory allows you to create a font association table. During the **Open** command, FelixCAD reads this file. When FelixCAD cannot find a font file, it uses this list to substitute other fonts. The following is an example of the content of the **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. Each line in the **[FONTTAB]** section has the following syntax:

*source\_font\_filename=fcad\_font\_filename*

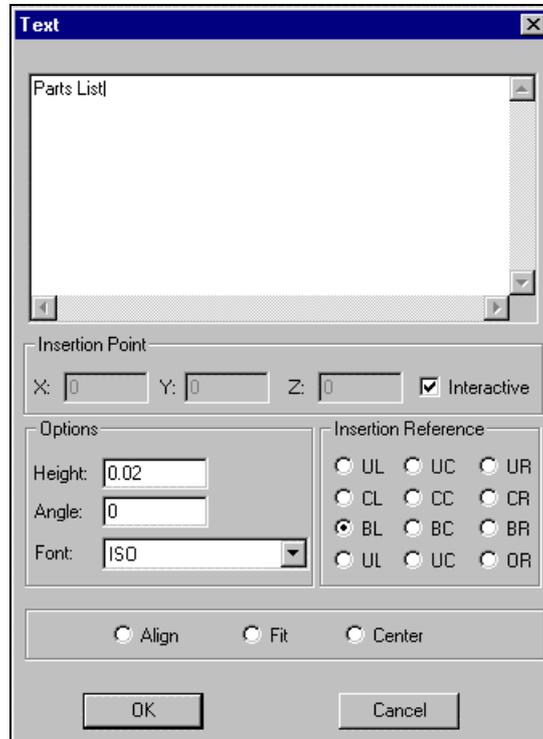
The *source\_font\_filename* must always be specified without filename extension, neither *.fsh* nor *.shx*.

The *fcad\_font\_filename* must always be specified without filename extension, but can have a path specification.

The program will search for an **SHX** or **FSH** file specified by *apply\_fcad\_font\_filename* in the application directories. If that font file is found, it will be applied automatically; otherwise the program takes the **alternative font file**, registered in the `felix4xx.ini` file.

## PLACING TEXT

**A** The **Text** command (**Detail > Text**) allows you to add text to the drawing via the **Text** dialog box. Note that executing the command **Qtext** emulates the command line versus dialog box variant of text entry (as described in the FelixCAD for AutoCAD guide):



### Text Input

Text is entered into the edit box in the upper half of the dialog box. The cursor is positioned here upon the opening of the dialog box, so you may start immediately typing text.

This dialog box offers the basic functions for editing text: the cursor keys, Page Up and Down, Home and End, and so on, work as you expect them to. Incorrectly entered text can be selected and deleted, just as with a word processor.

## Insertion point

There are two methods of defining the insertion point. The insertion point may be entered using (absolute) coordinates via edit boxes for **X**, **Y**, and **Z**.

The second option is to define the insertion point interactively on the screen. Click the **Interactive** button to define the insertion point after you click **OK**. FelixCAD will prompt you “Insertion point:”.

## Options

The **Options** section aids you in determining the height, the rotation, and the font (type style) of the text object.

### Height

Entering a numeric value for **Height** gives the font a *fixed* height, which can no longer be changed when creating the text using this style. If you think that a change in the font height might be required, keep the height parameter set at 0 (the default).; you may change the font height later in the **Text** dialog box.

### Angle

Specify the insertion angle of the text. The default value is a rotation angle of 0 degrees: the text object is inserted horizontally. Any positive or negative value between 0 and 360 is possible. Use this option to create text at right angles (**Angle** = 90) or to align the text to vertical and sloping objects.

### Font

A drop-down list allows you to select a font (text style) 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**.

- **Align:** Text is aligned between two points, with proportionate text height. The **Angle** and **Height** options are ignored. After you click **OK** to exit the **Text** dialog box, FelixCAD prompts you for the “First point:” and “Second point:”. The two points define the rotation angle and the text height.
- **Fit:** Text is fit between two points, at the specified height. The **Angle** option is ignored. After you click **OK** to exit the **Text** dialog box, FelixCAD prompts you for the “First point:” and “Second point:”. The two points define the rotation angle.
- **Center:** Text is centered on the insertion point.

## Insertion Reference

The **Insertion Reference** section specifies the position and the alignment of the text object in relation to the insertion point of the text object. During insertion of the text object, this reference point is coordinated with the insertion point. If the reference point is, for example, 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.

### Vertical Alignment

**T (top):** the insertion point is located at the top of the text.

**C (center):** the insertion point is located at the center of the text.

**B (baseline):** the insertion point is located at the baseline of the text.

**U (underneath):** the insertion point is located at the bottom of the text.

### Horizontal Alignment

**L (left):** the insertion point is located at the left of the text.

**C (center):** the insertion point is located at the center of the text.

**R (right):** the insertion point is located at the right of the text.

UL, for example, represents the *upper left* corner of the text. Select the position for the reference point by clicking a radio button.

## Special Text Characters

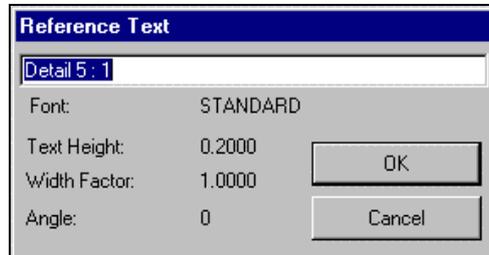
Special characters are available for use with text entities. They are written in the text dialog box one way, yet appear on the screen and printed versions in the desired format. See table below.

Special Character	Text Written in Text Dialog Box	Uses
φ	%%c	Used in place of DIAMETER or DIA.
°C °F	%%d C %%d F	Used to indicate temperature
<u>Overlined Text</u>	%%oOverlined Text%%o	-
<u>Underlined Text</u>	%%uUnderlined Text%%u	-

## REFERENCE TEXT

The **RText** command (**Detail > Reference Text**) allows you to enter new text that matches the style of existing text. Simply pick a text entity, then overwrite the string in the dialog box.

```
> RTEXT  
Select reference text: P1
```



```
Insertion point: P2
```

Finally, 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:

## MODIFY TEXT



The **TextEdit** command (**Detail > Modify Text**) allows you to modify existing text in the drawing.

Select a text entity in the drawing and FelixCAD displays the **Text Edit** dialog box. The dialog box is filled out with the parameters of the text.



When the **Interactive** check box is on, you will be prompted to relocate the text when exiting the dialog. Otherwise, you may change the insertion point coordinates in the dialog box, if desired.

## CORRECT TEXT

The **TCorrect** command (**Detail > Correct Text**) allows you to edit text and attribute definitions. You may change the text as well as the font, size, etc.

The command first prompts you to select the text or attribute definitions. Use the usual object select functions. Finish the object selection by **Enter**.



### Remain

The upper half of the dialog box lists all text properties that can be modified. The check boxes next to **Remain** indicate whether the property is to be changed globally (off) or remain unchanged (on). For example, if you click the check box below the **Height** (to turn it off), you may enter a new value for the text height. All selected text lines are changed to this text height.

### Search String

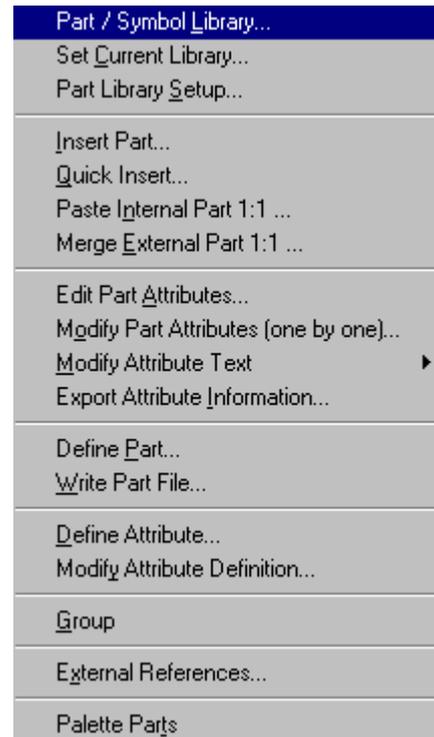
The lower half of the dialog box is a search function. Enter a string in the **Search String** edit box. FelixCAD will search for this text. Type the replacement string in **Replace by**. If this is left blank, FelixCAD erases the search string text.

# Parts and Attributes, Externally-Referenced Drawings, & Groups

This chapter describes how to work with parts, attributes, and groups. *Parts* are known as blocks or cells in other CAD programs. *Attributes* are database-like text attached to parts. *Groups* are similar to named selection sets.

While parts and groups sound similar, there are significant differences:

- A part has an insertion point; a group does not.
- A part can be inserted one or more times in the drawing; there is only one occurrence of the group.
- When a part is redefined, all of its insertions are updated; groups are not redefined.
- It is difficult to edit the entities making up a part; entities making up a group are easily edited.



## THE ADVANTAGE OF PARTS

Parts allow you to create a drawing using repetitive entities. These entities may contain a single entity, a group of entities, a partial drawing, or a complete drawing. These are inserted into a drawing as a single entity.

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 **Parts** menu.

### Properties of Parts

A part is a complex object that consists of several drawing entities:

- Basic entities, such as line, circle, and arc.
- Drawing objects, such as hatch patterns and dimensions.
- Complete drawings, whether in *.flx*, *.dwg*, or *.dxf* format.
- Drawings already defined as parts.

After a selection of entities is selected and defined as a part, it is saved by a name and treated as *a single entity* from then on.

Editing, changing, and erasing a part always affects the part as a whole. The **Explode** command (**Edit > Explode Complex Objects**), dismantles (explodes) the part into its original components.

A part is inserted in the drawing with the **Insert** command. During the command, you can specify the scale and rotation angle to fit the part to the current drawing. When the part is inserted into the drawing, all of its properties (layer, color, and linetype) are inserted with it.

The part is inserted on the layer it was defined on. If the drawing does not contain that layer name, FelixCAD automatically creates the missing layer. The exception to this rule occurs when the part was created on layer **"0"** – in this case, the part is placed on the current layer.

## Internal and External Parts

Internal and external parts are treated differently with regard to storage location and their relationship to the drawing.

*Internal parts* have been created and stored in the current drawing. They are *internal* to the drawing. These parts may be used in the current drawing, and are available for use in other drawings via the Drawing Explorer's (**Xplorer** command) **Block** option.

The **PartExp** command (**Parts > Write Part File**) exports the part to its own file on disk, in *.flx*, *.dwg*, or *.dxf* format. On their own, these parts are called *external parts*. Stored in separate drawing files, they are independent of the current drawing and can be used by other drawings and CAD programs.

We recommend that you store external part in a library so that they can be easily accessed by all users at your firm.

*Note: AutoCAD users know these External Blocks as 'Wblocks'.*

## Attribute Definitions

A part can be given *attributes*, which are used to store non-graphical data, such as part number, price, or manufacturer name.

Attributes can be displayed or be hidden from view. Attributes can be exported to a text file, then read by a database or spreadsheet for further processing, such as counting all the parts in the drawing.

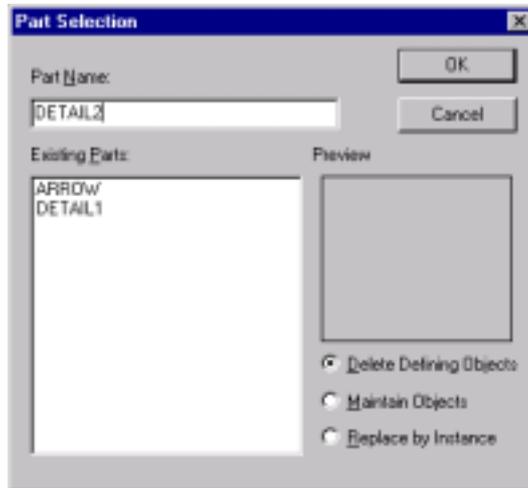
## CREATING PARTS

A part is created in three steps: (1) select a group of entities; (2) name the part; and (3) select an insertion point for the part.

### Creating an Internal Part



The **PartDef** command (**Parts > Define Part**) creates an internal part. The command displays the **Part Selection** dialog box.



#### Part Name

The name of the part can have up to 31 characters long. In addition to letters and numbers, you may use the \$, -, and \_ characters; do not use any spaces in the name!

#### Existing Parts

If the drawing already contains parts, their names are listed in the **Existing Parts** list box. Do not use the name of an existing part, unless you want to redefine a part.

If one of the parts is to be redefined, select the part name from the list. An alert box appears, warning that you are about to redefine that part. The message gives you the option of choosing a new name or redefining the part. **Note:** When you redefine a part, all insertions of that part are also redefined.

The radio buttons allows you to specify how to treat the selected entities after the part definition is completed. The three choices are:

- **Delete Defining Objects:** When the **PartDef** command completes, the entities you selected are erased from the drawing (default).
- **Maintain Objects:** The entities you selected are *not* erased from the drawing.
- **Replace by Instance:** The entities you selected are replaced by the newly defined part.

Usually, after you create a part, the defining entities are not used anymore in the drawing. That's because the purpose of the part is to be inserted in the drawing. If you want the defining objects to be removed from the drawing, select the **Delete Defining Objects** radio button (this is the default setting in the dialog box.)

**Note:** You can restore the defining objects (after they are erased) with the **Undelete** command.

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), select the **Maintain Objects** radio button.

If you wish to leave a copy of the block at the current location in the drawing, select the **Replace by Instance** radio button.

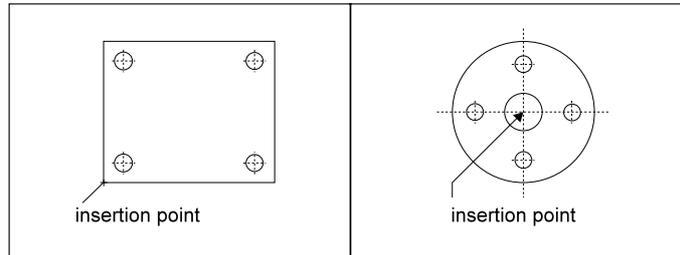
### Insertion Point

Click the **OK** button to dismiss the dialog box and the **PartDef** command continues in the text window:

```
> partdef
Displays dialog box
Insertion base point: P1
Select objects: [pick]
1 selected.
Select objects: [pick]
1 selected.
Select objects: [Enter]
*** 2 selected. ***
```

The part must have an *insertion point*. The insertion point is a reference that enables you to position the part properly during insertion. The insertion point

also serves as the basepoint for changing the scale and the rotation angle. You can pick anywhere for the insertion point – on or off the part – although users typically select the left lower corner, the center of the block, or some other discreet point of the geometry. It is a good idea to use object snap options functions to ensure precision.



### Select Objects

Finally, you select the entities 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 with the **Insert** command.

### Creating an External Part \*



The **PartExp** command (**Parts > Write Part File**) creates a part and saves it to a drawing file on disk. Defining an external part is very similar to defining an internal part.

The command displays the **Export Part** dialog box, which allows you to specify the drive, folder, filename, and file format. You can save the part in five formats, all of which can be read back into a FelixCAD drawing:

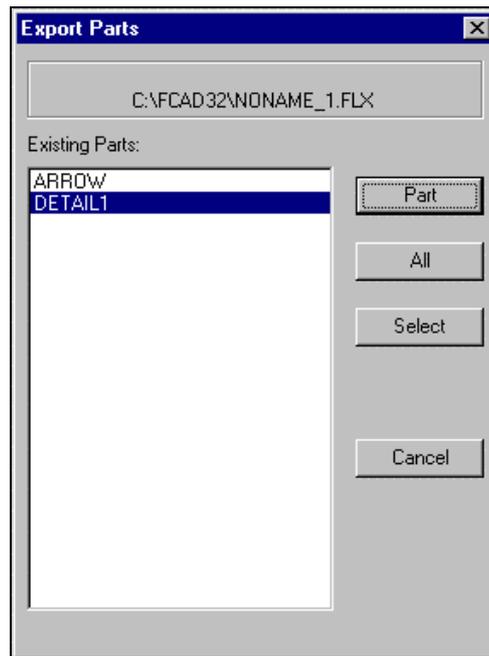
- FelixCAD drawing *.flx*
- AutoCAD R12 drawing *.dwg*
- AutoCAD R13 drawing *.dwg*
- AutoCAD R14 drawing *.dwg*
- DXF drawing *.dxf*

You would only use the AutoCAD formats if the part is to be used with compatible software, such as AutoCAD or AutoCAD LT. The DXF format is meant for use by other CAD packages. Click **Save** and FelixCAD displays the **Export Parts** dialog box. An external part can consist of:

**Select:** a single entity or a selection set of entities; select the entities using the usual object selection methods. You are prompted for the insertion point.

**Part:** an existing internal part; select the part name from the **Existing Parts** list. You are not prompted for an insertion point since the part already has its insertion point defined.

**All:** the entire current drawing. You are not prompted for an insertion point since it is assumed to be at 0,0. The entities are *not* erased from the drawing.



### Difference From Save As

The **PartExp** command can also be used to save the entire drawing. This is similar to the **SaveAs** command but with one primary difference: **PartExp** purges all unused block references and unused symbol table entries (layer, linetypes, text styles, etc.) from the drawing. This is an excellent way of ridding the drawing of unreferenced entries and frees it from unnecessary “ballast.”

## INSERTING PARTS

Once a part has been defined, you can insert it in the drawing as many times. You can think of the **Insert** command as a more efficient form of the **Copy** command. Both commands make copies, but the **Copy** command makes copies of the entities, whereas **Insert** makes a reference to the block – resulting in a smaller drawing file.

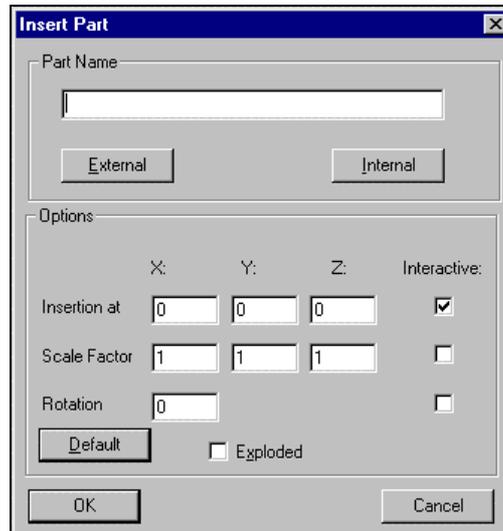
FelixCAD has four commands for inserting parts:

- **Insert** command (**Parts > Insert Part**) displays dialog box.
- **QInsert** command (**Parts > Insert Quick**) operates at the command prompt.
- **InsertQ** command (**Parts > Paste Internal Part 1:1**) quickly inserts a part by assuming scale = 1 and rotation = 0.
- **Merge** command (**Parts > Merge External Part 1:1**).

### Insert Part



The **Insert** command (**Parts > Insert Part**) uses the **Insert Part** dialog box to allow you to specify how the part is inserted.

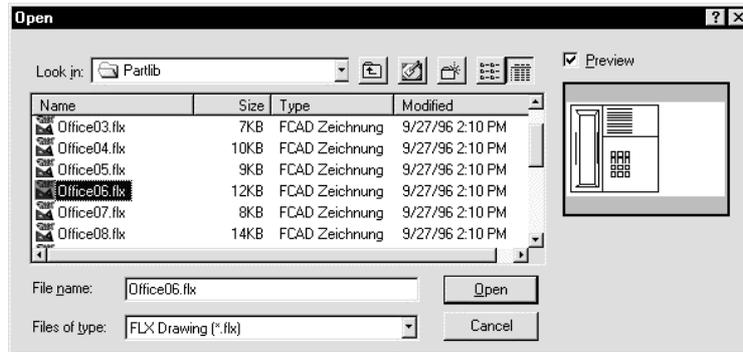


## Part Name

Type the name of the part in the text entry box. If you don't know the name (or can't be bothered typing), click either the **External** or **Internal** buttons.

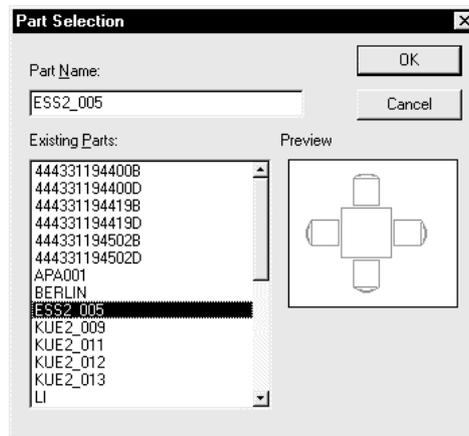
### External

The **External** button displays the **Open** file dialog box. This lets you select an external part. The preview window lets you see what the part looks like before you import it. Select any *.flx*, *.dwg*, or *.dxf* file and click **Open**.



### Internal

The **Internal** button displays the **Part Selection** dialog box. This lets you select any block stored in the drawing. The preview window lets you see what the part looks like before you insert it. Select a block name and click **OK**.



## Options

### Insertion At

Define the **Insertion Point** by typing the X, Y, and Z coordinate values. If **Interactive** is turned on (the default), you will be prompted for the insertion point after you click **OK**.

### Scale Factor

The **Scale Factor** allows you to specify the scale factor *before* the part is inserted. Remember that a scale factor greater than 1.0 increases the size of the part, and a factor less than 1.0 reduces the size of the part. If you type a negative number the part is mirrored upon insertion.

If **Interactive** is turned on (the default is off), you will be prompted for the scale factor after you click **OK**. Dragmode is active during this operation, so you will have the benefit of seeing the scale factor result before confirming the size.

### Rotation

The **Rotation** allows you to specify the rotation angle *before* the part is inserted. For counter-clockwise rotation, enter a positive value, for clockwise rotations, enter a negative value between 0 and 360 degrees.

If **Interactive** is turned on (the default is off), you will be prompted for the rotation after you click **OK**. Dragmode is active during this operation, so you will have the benefit of seeing the rotation result before confirming the angle.

### Default

The **Default** button reset all parameters back to the original values:

- **Insertion point:** Interactive
- **Scale Factor:** 1,1,1
- **Rotation:** 0

### Exploded

The **Exploded** button causes the part to be exploded into its component entities upon insertion. When exploded, the part no longer exists; instead, you get the individual entities that made up the part, which allows you to edit them.

## Quick Insert

The **QInsert** command (**Parts > Quick Insert**) is the command-line version of the **Insert** command. The command prompts you for the part name, insertion point, rotation angle, and scale factor in the text window. It does not have the explode and interactive options found in the **Insert** command's dialog box.

The **QInsert** command can be faster since it offers a reduced number of options. Also, the command is the best suited for macros and FLISP routines. There are, in fact, two versions of the **QInsert** command:

1. Select **Parts > Quick Insert** from the menu. The **Part Selection** dialog box is automatically displayed. All remaining prompts are displayed in the text window.
2. Type **QInsert** at the command prompt. All prompts are displayed in the text window. You can display the **Part Selection** dialog box by typing **?** at the "Partname:" prompt.

```
> QINSERT
Partname <part01>: [type part name or press ?]
Insertion point: 3,3
X factor <=1>/Corner/XYZ: [Enter]
Rotation angle <0>: [Enter]
```

### Partname

Type the name of the part. For external parts, include the drive and path name, as in drive:\folder1\folder2\partname.ext. If you don't know the name of the part, press **?** to display the **Part Selection** dialog box.

### Insertion point

Enter X, Y coordinates for the insertion point, or pick the point with the cursor. The part will be placed using the part's previously defined insertion point (during the **PartDef** or **PartExp** commands).

### X Factor

The "X Factor" prompt is asking you for the scale factor, which you can specify in several ways.

"X factor" asks for the scale factor in the X direction. After entering the scale factor, the command prompts you "Y Factor." Enter the value. In both cases, you can press **Enter** to keep the X factor the same as before, and make the Y factor the same as the X factor.

Remember that a scale factor greater than 1 will enlarge, less than 1 will reduce. A negative scale factor mirrors the part upon insertion. FelixCAD provides several alternatives to specifying the scale factor, as seen on the option bar:

X=Y=Z=1	Corner	XYZ			
---------	--------	-----	--	--	--

### **X=Y=Z=1**

The **X=Y=Z=1** option sets the scaling factor to 1.0 for all three coordinates. The X, Y, and Z scale factors equal 1. This inserts the part at its original size..

### **Corner**

The **Corner** option lets you pick two points in the drawing to defines the scale. Indicate the opposite corners of the rectangular window within which the part is to be scaled. **Dragmode** shows both the window and a preview of the part as you specify the two corners.

### **XYZ**

The **XYZ** option lets you specify a different scale factor for each of the three coordinates. This enables you to input different scaling factors for parts. The scaling factor for the X, Y, and Z-axis are each requested.

## **Rotation Angle**

The “Rotation angle:” prompt asks for the angle to rotate the part. The default is 0.0. Remember that a counter-clockwise rotation requires a positive value, a clockwise rotation requires a negative value between 0 and 360 degrees.

This angle can also be specified using the cursor to pick the value. **Dragmode** shows a preview of the angle as the cursor moves.

**PASTE INTERNAL PART 1:1**

The **InsertQ** command (**Parts > Paste Internal Part 1:1**) is useful for quickly inserting an internal part. The command assumes that the scale factor is 1 and the rotation angle is 0.

```
> INSERTQ
Displays Part Selection dialog box.
Insertion point: P1
```

The **Part Selection** dialog box lets you select an internal or external part.

At the “Insertion point:”, enter the X, Y coordinates or pick a point in the drawing. If you want to change the scale factor or rotation angle, select from the option bar:

Factor	Rotation				
--------	----------	--	--	--	--

See the **Quick Insert** command for a description of these options.

**MERGE EXTERNAL PART 1:1**

The **Merge** command (**Parts > Merge External Part 1:1**) is useful for quickly inserting an external part. The command assumes that the scale factor is 1 and the rotation angle is 0.

```
> MERGE
Displays Part Selection dialog box.
Insertion point: P1
```

The **Part Selection** dialog box lets you select an internal or external part.

At the “Insertion point:”, enter the X, Y coordinates or pick a point in the drawing. If you want to change the scale factor or rotation angle, select from the option bar:

Factor	Rotation				
--------	----------	--	--	--	--

See the **Quick Insert** command for a description of these options.

## ***DINSERT COMMAND***

The **DInsert** command (short for “discreet insertion”) incorporates an external block into the drawing without creating an instance of the block in the drawing. The main of this command is to:

- Preload parts for later use.
- Load table definitions defined in a template drawing (layers, linetypes, text styles, dimension styles, etc.)

This command is provided for use in macros, FLISP functions, and other programming interfaces.

```
> dinsert  
Part name: [type external part name]
```

The command only requests the filename of the external part. The ? option is not available.

**EXPLODE PARTS**

The **Xplode** command (**Edit > Explode Complex Objects**) allows you to explode complex objects, such as parts, into their constituent parts. Parts inserted in the drawing constitute a single object. It is not possible to edit any of the entities that make up the part. If you need to edit one of the entities comprising the part, then the part must be exploded. In addition to exploding parts, this command is used to explode polylines, hatching, and dimensions.

There are, in fact, two versions of the **Explode** command:

1. Select **Edit > Explode Complex Objects** from the menu. The **Explode Complex Objects** dialog box is displayed.

```
> xplode
```

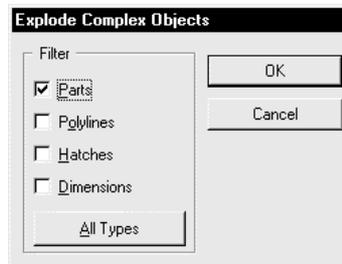
*The Explode Complex Objects dialog box appears.*

```
Select objects: P1
```

```
1 selected.
```

```
Select objects: [Enter]
```

```
*** 1 selected. ***
```



This dialog box allows you to select all objects in the drawing, then have only all parts (and/or polylines, etc) exploded.

2. Type **Xplode** at the command prompt. No dialog box is displayed.

```
> xplode
```

```
Select objects: P1
```

```
1 selected.
```

```
Select objects: [Enter]
```

```
*** 1 selected. ***
```

After selecting the object(s) to explode, press **Enter** and the parts (or polylines or hatches or dimensions) are exploded. You may now choose any of the entities for editing.

**Warning:** If attributes are attached to the part, they are lost with the explosion. The attribute data is converted to the attribute name, as a text entity.

## THE PART LIBRARY



The **Library** command (**Part > Part/Symbol Library**) lets you to manage parts and drawing files, such as project drawings, template drawings, and symbol files. A *library* is defined by a *.plb* file. The drawing files are assembled in the same folder (sub-directory) as the part library file is located. By default, this is in the *\partlib* folder but you can use any folder for storing the part library.

### Selecting a Part Library

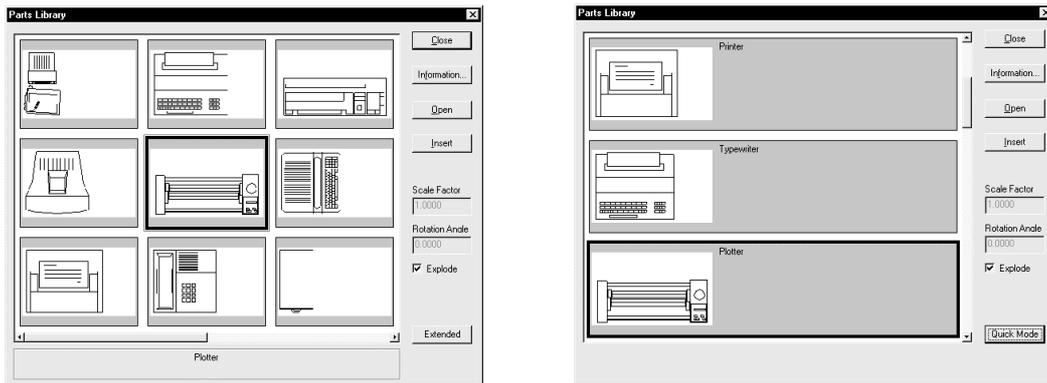
The **Library** command displays the **Set Current Part Library / Symbol Library** file dialog box. Select a part library (*.plb*) file.

Or enter the **PartLib** command, which prompts you for the name of a part library at the command prompt. This command is meant for use with macros and FLISP functions. Enter the name of a *.plb* file, such as:

```
> partlib  
File name: c:\felixcad\partlib\partlibdrawing.plb
```

Once a part library has been chosen, the **Library** command continues to use that same library. Use the **SetLib (Part > Set Current Library)** to change the part library.

The command displays the dialog box, with a preview image of each part in the library. This lets you visually select the part.



The **Part Library** dialog box in **Quick Mode** (left) and **Extended** mode (right).

The dialog box has two display modes:

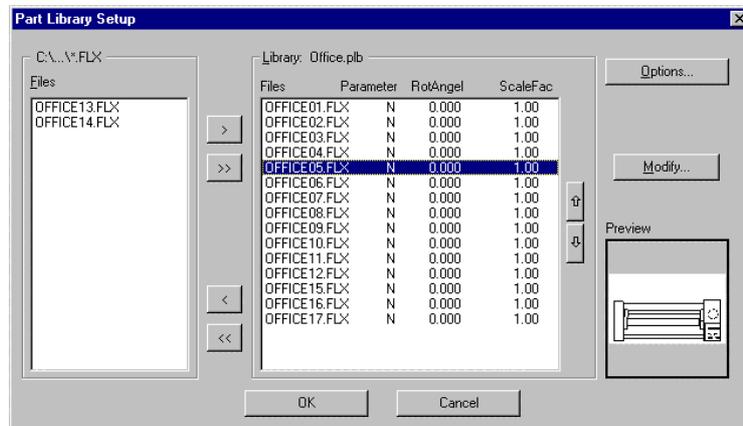
- **Quick Mode:** nine parts at a time are displayed by preview images, with the selected part's name at the bottom.
- **Extended:** three parts are displayed at a time, together with the parts' information.

### Set Up and Management

The **PLbSetup** command (**Parts > Part Library Setup**) allows you to set up a new part library, specify the user interface of the part library dialog box, and modify the content of existing part libraries. The command displays the **Part Library Setup** dialog box that:

- Selects the drawing files to be included in a library.
- Sets the preferences for part insertions.
- Sets the parameters for the layout and options of the part library dialog box.

The *.plb* files store the parameters for each part library. Note that the *.plb* file stores only the parameters that define the library; it does not store the part or drawing files. The **Part Library Setup** dialog box does all the work creating the *.plb* file; you don't need to know the structure or syntax of the part library file.



### Specifying the Drawing Files

The **Part Library Setup** dialog box allows you to select the *.flx* drawing files for the library. You are limited to selecting drawing files from a single folder (sub-directory). On the other hand, you can have more than one library file in a folder.

Note the part library setup only allows addition or removal of *.FLX* file formats. If you wish to create a part library from existing *.DWG* or *.DXF* files, convert the files to *.FLX* format using the batch processing function of FelixCAD (see *BATPROC* command).

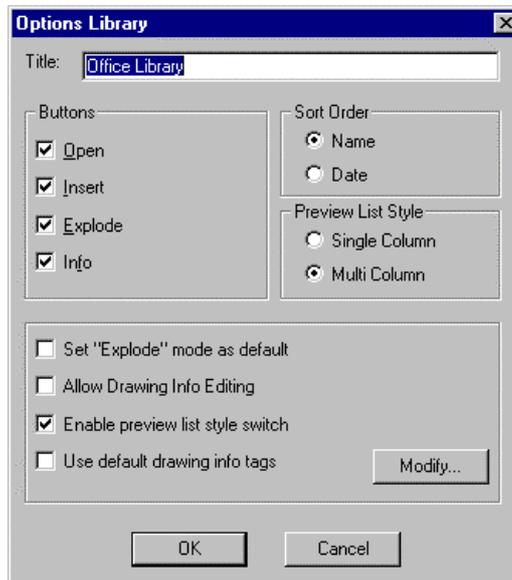
To add drawings to the library, highlight the filenames in **Files** list, then click to the **>** button to add them to the **Library** list. The **>>** buttons adds all files to the library.

The **<** button removes selected files from **Library** list, while the **<<** button removes all part files.

When a single file is selected in either list box, the preview window displays an image of the drawing.

## Options

The **Options** button displays the **Part Library: Dialog Layout Options** dialog box, which lets you select the user interface options displayed by the **Part Library** dialog box.



### Title

The **Title** edit box requires you to provide a name for the part library dialog box.

### Buttons

The **Buttons** area lets you decide which buttons you want displayed by the Part Library dialog box:

**Open:** displays the **Open** button, which allows you to open the drawing for editing.

**Insert:** displays the **Insert** button, which allows you to insert the drawing as a part.

**Explode:** displays the **Explode** checkbox, which allows you to explode the block upon insertion.

**Info:** displays the **Information** button, which allows you to view information attached to the part.

### **Sort By**

The **Sort by** area determines whether the **Part Library** dialog box sorts part files alphabetically by name (used for part libraries) or by date (used for project libraries).

### **Preview List Style**

The **Preview List Style** area determines whether the **Part Library** dialog box should start up with the single column (**Extended**) or multi-column (**Quick Mode**) format of displaying the part previews.

### **Other Options**

The Set **“Explode” mode as default** checkbox determines whether the **Explode** checkbox is turned on as the default.

The **Allow Drawing Info Editing** check box determines whether you are allowed to edit the drawing file information.

The **Enable preview list style switch** check box determines whether the **Part Library** dialog box displays the **Quick Mode** and **Extended** buttons.

The **Use default info tags** check box determines whether the default information tags (Project, Name, Drawing No., Draftsman, and Notes) or the user-defined info tags are used when drawing information is displayed (see the next item).

### Modify

The **Modify** button displays the **Part Library: Drawing Info Tags** dialog box. It allows you specify custom tags for the information fields used by the part. Here you can enter information like the manufacturer name, price, and so on.



You can change the rotation angle and scale factor of more than one part at a time, as follows:

1. Select one, more than one, or all files in the **Library** list.
2. Click the **Modify** button.
3. Click **Insert parameters activated** to turn it on (check mark shows).
4. Change the value of the rotation angle and/or scale factor.
5. Click the **Apply** button.

The select parts now have the same rotation angle and scale factor.

## **ATTRIBUTES**

You can attach *attributes* to parts. Attributes help create more meaningful drawings. They aid in materials calculations, creating bills of material, and other database-like functions.

Attributes are text objects that are joined to parts. The attribute text may be constant or variable, visible or hidden, a single or multiple attributes.

An attribute consists three components:

**Name:** identifies the attribute. When a part contains more than one attribute, the name helps you identify a specific attribute.

**Request:** the prompt text, which is displayed when the part is inserted. The request reminds the user of the type of data that should be entered.

**Value:** the default value of the attribute. The user can accept the value or type in another value – except when the attribute has been defined as *constant*. The value can be visible in the drawing, or invisible.

When an attribute is created, you must specify the following parameters (which can be edited later):

- Insertion point.
- Font options, such as font name, height, angle, and reference point.
- Visible or hidden.
- Constant or variable.

You may create an attribute and place it in the drawing all by itself. In that case, the attribute definition represents an isolated text object.

More commonly, attributes are linked to parts. In this case, you define the attribute before defining the part. In most cases, you place the attribute near or on the entities. When you select the entities that make up the part, you also select the attribute(s).

*Note that the order in which the attributes are picked will determine the order in which they appear when inserting or modifying a part. This is useful in creating drawing borders and other families of parts.*

## Define Attribute



The **AttDef** command (**Parts > Define Attribute**) creates a new attribute definition. The command displays the dialog box:

## Attribute

### Name

Provide a name for the attribute. Use as descriptive a name as possible. The name can be as long as 31 characters, and may include special characters  (underline) and \$ (dollar). In the dialog box above, the name of the attribute is “STANDARD”.

### Request

Type the text of a prompt, which is displayed when the part is inserted. The request reminds the user of the type of data that should be entered. (The request is not displayed when the attribute is defined as a *constant* attribute.) In the dialog box above, the request is “ANSI Standard:”.

### Value

Enter the default value of the attribute. The value can be up to 256 character long, including blanks and special characters. The user can accept the value or type in another value – except when the attribute has been defined as *constant*. The value is not displayed when the attribute is defined as an *invisible* attribute.

### Insertion Point

The insertion point may be defined in either of two ways. One method is to type the coordinate values in the X, Y, and Z boxes. The attribute will be inserted at that X, Y, Z coordinate.

The second method is to click the **Interactive** checkbox to turn it on. This lets you define the insertion point via the cursor, after you click the **OK** button. The advantage to the interactive definition is that **Dragmode** is active, offering you a preview of the insertion.

### Options

The text options define the height, rotation angle, and the font (text style) of the attribute text.

#### Height

The height of the text is specified by the **Height** field. Type the text height in drawing units (*not* in text points) or type **0** (zero) to specify the text height later, after you click the **OK** button.

#### Angle

The **Angle** field specifies the angle of the attribute text. For an counter-clockwise rotation enter a positive value, for a clockwise rotation enter a negative value, between 0 and 360 degrees.

#### Font

The **Font** list box lets you select a text style for the attribute text. The style must be previously defined by the **Font** command.

## Reference Point

The **Reference Point** area determines the position and alignment of the attribute text in relation to the insertion point. For example, if the reference point is TL (top left corner of the text), then the attribute text will run from the insertion point to the right, and hanging down from the insertion point.

There are twelve reference points available. The characters represent the initials of the position on the text object. BL, for example, represents Baseline Left; UR represents Upper Right. Centered positions are available, as well.

## Alignment

### Align

When the **Align** option is active, the **Insertion Point**, **Height**, **Angle**, and **Reference Point** options are grayed out. That's because the **Align** option will prompt you to pick two points on the drawing; the attribute text is placed between the two points. The first point is interpreted as the insertion point; the second point sets the angle and the scale factor, which determines the text height.

### Fit

The **Fit** option also places the attribute text between two points, like the **Align** option. The sole difference is that the **Height** field remains available. Upon insertion, the text height remains fixed, but the text string is expanded or compressed as required to fit.

### Center

The **Center** option centers the attribute text, vertically and horizontally. Text height and rotation angle may be specified as usual.

## Flags

### Invisible

When **Invisible** is on, the attribute text is not displayed in the drawing. This is useful, since attribute text tends to clutter the drawing. You can change the visibility later with the **DrawMode** command.

### Constant

When **Constant** is on, the attribute's value cannot be changed. When off, the user is prompted by the **Request**. You can change the value later with the **AttXEdit** command.

### Preselect

The **Preselect** option causes the attribute to accept its default value, automatically. When **Preselect** is active, you are not prompted for an attribute value (like with the **Constant** option), but you can still edit the value later with the **AttValue** command.

**Note:** To create more than one attribute for a part, simply repeat these steps until all attributes are inserted in the drawing near each other.

## Working with Attributes

Once the attribute is defined, you include it with a part. Then, when the part is inserted in the drawing, you need to provide values for the attributes.

### Including Attributes with Parts

To save a part with attributes, follow the instructions earlier in this chapter for creating parts. Here are the steps you need to go through:

1. Draw the entities that will go to make the part.
2. Define the attribute(s) with the **AttDef** command. Insert the attribute(s) near the entities.
3. Start the **PartDef** command and define the part.
4. When prompted, select the attributes and the entities making up the part.

The part, including the selected attributes, is now saved in the drawing.

A part with attributes can also be saved externally. Follow the same procedure with the **PartExp** command.

### Inserting Parts with Attributes

When you insert a part that includes attributes, the procedure is the same as for parts without attributes. The **Insert** command has just one added step: type the value of the attribute. (This step only occurs when the attribute does not have the *constant* definition.) I

Press **Enter** to accept the default values, or type another value. If the part contains several variable attributes, you are prompted to keep providing values for all attributes. Finally, the part is inserted in the specified position.

## Editing Attributes

There are two ways to edit attributes inserted in the drawing. You may edit:

- The values of the attributes to change the text; use the **AttXEdit** command.
- The properties of the attributes, such as their visibility or their name; use the **AttEdit** command.

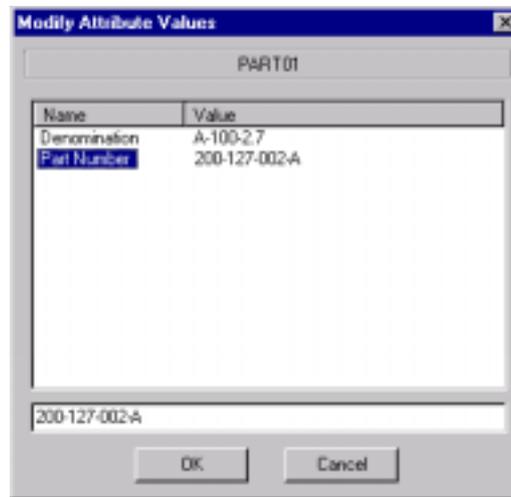
## Edit Part Attributes



If only the value(s) of the attribute are to be changed (whether constant or variable), use the **AttXEdit** command (**Part > Edit Part Attributes**).

```
> attxedit  
Select part: [pick]
```

The following dialog box appears:



The top line displays the name of the attribute.

The **Name Value** list displays the request wording under **Name**, and the current value under **Value**.

Select the attribute value to change. Edit the value in the text entry box at the bottom of the dialog box. Make your changes and click **OK**.

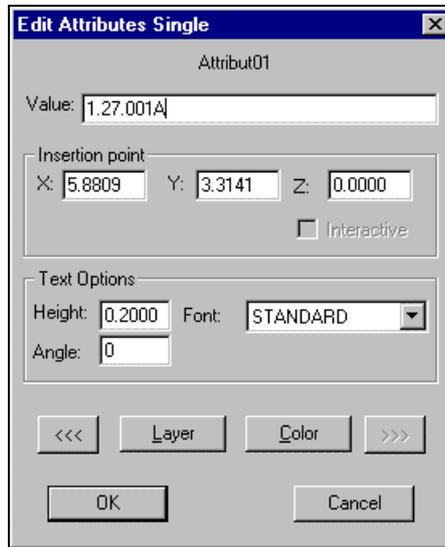
## Modify Attributes



To edit all properties of an attribute (with the exception of *constant* values), use the **AttEdit** command (**Parts > Modify Attributes**).

```
> attedit
Select part: [pick]
```

Select only one part. If the part does not contain attributes, you are prompted to select another part. After the correct part is selected, the dialog box appears:



The editing process is identical to the procedure for creating an attribute with the **AttDef** command, except that the attribute name is displayed (at the top of the dialog box) and cannot be changed. The following buttons are new to this dialog box:



When a part contains more than one attribute, the dialog box displays only one at a time. The << and >> display the previous and next attributes.

The **Layer** and **Color** buttons allow you to change the layer that the attribute resides on, and the color of the attribute text.

## Attribute Utilities

FelixCAD offers several commands for modifying the text of attributes. . These utilities are found in the **Modify Attribute Text** submenu on the **Parts** menu.

### Edit Attribute Text Value

The **AttValue** command (**Parts > Modify Attribute Text > Edit Attribute Text Value**) allows you to select a single attribute of a part and change its value at the command line. This command is useful for macros and FLISP routines.

```
> ATTVALUE  
Select attribute of part: P1  
New value for attribute <47>: 48
```

### Move Attribute Text

The **AttMove** command (**Parts > Modify Attribute Text > Move Attribute Text**) allows you to change the position of the text.

```
> ATTMOVE  
Select attribute of part: P1  
Target point: P2
```

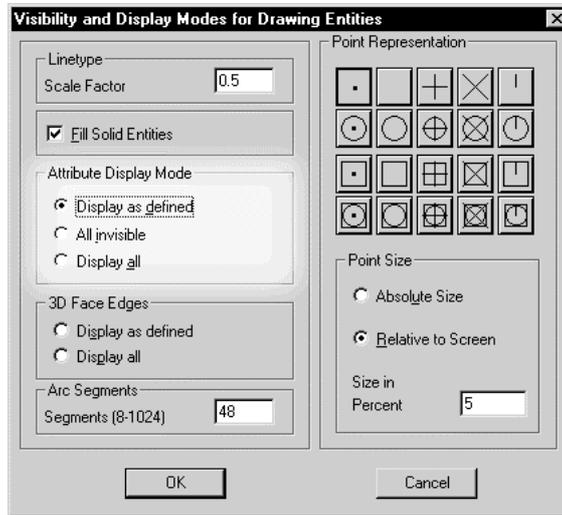
### Rotate Attribute Text

The **AttRot** command (**Parts > Modify Attribute Text > Rotate Attribute Text**) allows you to select a single attribute and change the rotation angle of the text:

```
> ATTROT  
Select attribute of part: P1  
New angle <0>: 90
```

## Attributes Display Modes

The **DrawMode** command (**Draw > Drawing Modes**) changes the display of attributes, to make them all visible (even if defined invisible) or make them all invisible. Notice the **Attribute Display Mode** section of the dialog box:



The following settings are available:

**Display as defined:** All attributes are displayed according to their definition – attributes defined as *invisible* are not displayed.

**All invisible:** all attributes are made invisible.

**Display all:** all attributes are made visible, even if defined as *invisible*.

Select the display option by clicking the radio button.

## EXTERNALLY REFERENCED DRAWINGS

FelixCAD's *externally referenced drawing* (or "xref" for short) feature lets you see other drawings in the current drawing. The xref drawing can be viewed but cannot be edited. Xrefs are useful for:

- Creating an assembly drawing from different files.
- Inserting detail drawings.
- Display a standard title block drawing.
- Temporarily reference another drawing in the current drawing.

Often, the biggest benefit of using externally referenced drawings is within a project. A workgroup creates several drawings and shares details over a network. For example, others can reference the electrical drafter's drawings.

Another advantage of xrefs is that the external drawing is *linked* to the current drawing, which is more efficient than inserting an entire drawing. In several situations, the use of externally referenced drawings is more convenient than inserting external parts into the drawing.

### Understanding Dependent Symbols

When an xref is attached to the current drawing, its symbol tables are loaded into the current drawing: layers, linetypes, text styles, dimension styles, and block definitions. However, the xref symbols are kept separate via the following naming scheme:

*Logical\_Xref\_Name* | *Dependent\_Symbol\_Name*

Notice the vertical bar separating the xref drawing name and the symbol name. Here are some examples of layer names, assuming the xref drawing's name is **Shaft.Flx**:

```
SHAFT | CONSTRUCTION
SHAFT | DETAILS
SHAFT | DIMENSION
```

The name prefix makes it easy to distinguish the dependent layers from the layers originally defined in the current drawing.

When the symbol tables derive from an xref drawing, they are called *dependent* because they depend on the existence of drawing other than the current

drawing. Other dependent symbol table entries, like linetypes and blocks, carry the same naming convention.

You cannot redefine nor rename dependent symbols. Dependent blocks cannot be inserted; a dependent layer cannot be made the current layer.

But the dependent symbol conventions allow you to control the visibility, the colors, and the linetypes of externally referenced drawings. The **Layer** dialog box displays the names of the dependent layers and you can apply the **On/Off**, **Thaw/Freeze**, **Linetype**, and **Color** options to them.

**Note:** To keep the visibility and displayed settings over subsequent sessions, make sure to set the system variable **VisRetain** to 1 (short for “retain visibility”). By default, **VisRetain** is set to a value of 0, which means that visibility and display changes made to the dependent layers are not maintained when you leave the current drawing.

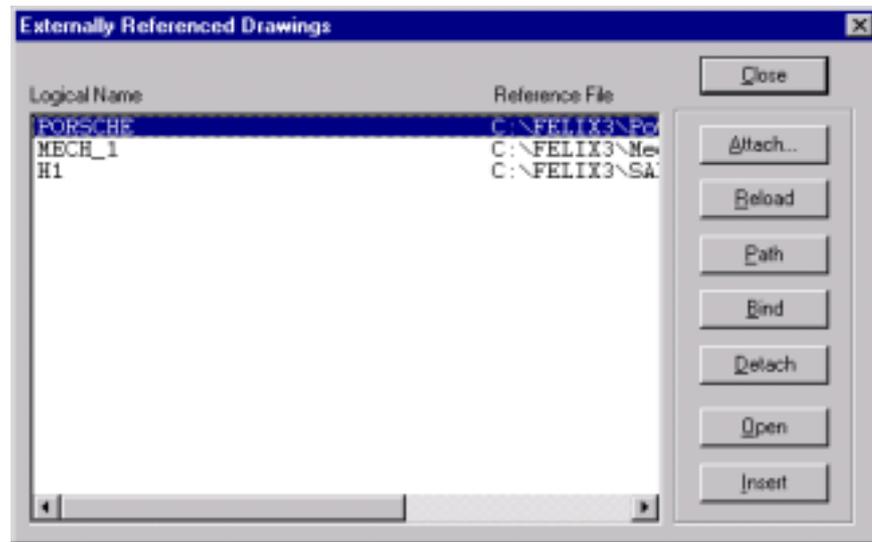
### XLink Command

The **XLink** command allows you to *link* one or more external drawings to the current drawing. A linked (or xref) drawing is treated very similar to a part (block) but there is a fundamental difference: an xref does not become a permanent part of the current drawing.

The **XLink** command (**Parts > External References**) displays the dialog box. Initially, the list box is empty. Click the Attach button to select another drawing file to attach. You may select *.flx*, *.dwg*, and *.dxf* files.

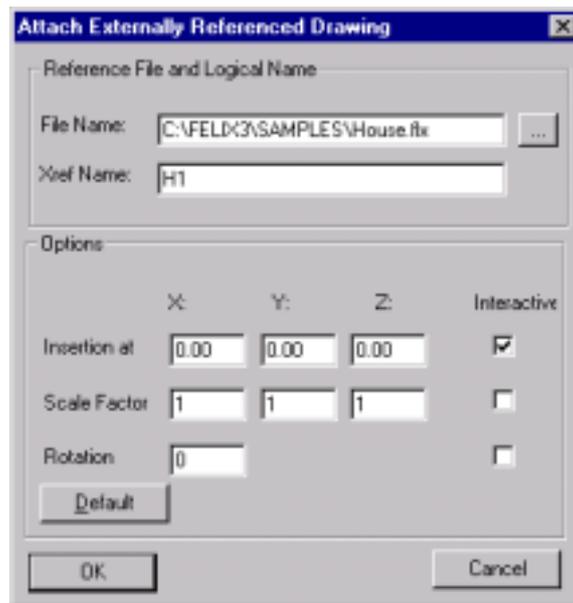
Once one or more drawings are attached, the dialog box displays the following information:

- The logical name assigned to the xref. The *logical name* is a way of identifying drawings that have a filename longer than 31 characters (Windows allows filenames up to 255 characters long).
- The drive, path, filename, and extension of the referenced drawing.



### Attach

The **Attach** option allows you to insert an external drawing as an xref to the current drawing. It displays this dialog box:



### Reference File and Logical Name

Type the name of the drawing in the **File Name** field, or click the ... button to select the drawing from a file dialog box.

You must type a logical name in the **Xref Name** field. This can be the same as the filename, if it is eight or fewer characters. The logical name must be short since it is used at the prefix for the names of dependant symbol names (layer names, linetypes, etc.).

### Options

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 earlier in this chapter.

Click **OK** and the drawing attached. If any of the **Interactive** check boxes are on, you will be prompted for the insertion point, scale factor, and/or rotation angle.

### Reload

The **Reload** button updates an externally referenced drawing during a current drawing session. The reload reflects any modifications made to the external file.

**Note:** When you attach an xref, it is displayed in its current state. When working on a project in a workgroup over a network, it may be necessary to update one or more external references from time to time. That's because another person may be editing the drawing at the same time you are using it as an xref. After this person saves their work, you need to reload the xref to ensure that you are seeing the latest version just saved.

To reload an xref, select the drawing from the list box and click the **Reload** button.

## Path

Sometimes an externally referenced drawing is moved to another folder (sub-directory) or is renamed. The **Path** button allows you to re-establish the link 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, select the external reference to be relocated from the list box and 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 some situations it may be useful to make the externally referenced drawing a permanent part of your drawing. For example, you need to send the drawing to the client, or the drawing 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.

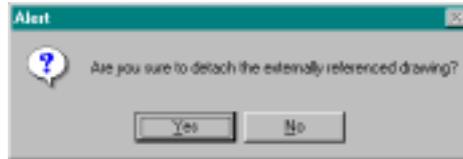
The **Bind** button inserts an externally referenced drawing into the current drawing. Once you bind an xref, the link is broken; you no longer see any modifications made in the original referenced file.

Bind also incorporates the layers, linetype definitions, text styles, and dimension styles into the current drawing. Recall that an xref used the vertical bar (|) to separate the logical name from the symbol name. With binding, the vertical bar is replaced by three characters,  $\$n\$$ , where  $n$  is a number that increments from 0. For example, the layer name HOUSE|BASEMENT changes to HOUSE\$0\$BASEMENT. The  $n$  can range from 0 to 9 as required.

To bind one or more external references to the current drawing, select an xref drawing in the list box click the **Bind** button.

## Detach

When you no longer need an xref drawing, click the **Detach** button. This also removes all dependent layer, linetype, and block references as well.



**Note:** Although you can erase an externally referenced drawing with the **Delete** command, we recommended you use the **Detach**, since this also removes the block definition from the drawing database.

## Open

FelixCAD's Multiple Document Interface allows you to display up to four drawings at a time. The **XLink**'s command's **Open** button allows you to simultaneously open an externally referenced drawing for editing. This allows you to modify the xref, for example, if you have detected discrepancies that need to be fixed.

To open a referenced drawing in another drawing window (viewport), select the external reference you want to open from the list box and click to the **Open** button.

Once you have modified an xref, save the drawing and close it. To display the revised external reference in the master drawing, use the **XLink** command with the **Reload** option.

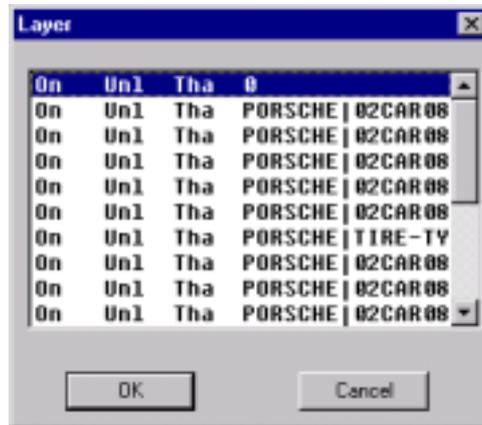
## Insert

The **Insert** button allows you to insert a copy of the xref into the current drawing like a part (block). You define the insertion point, the scale factor, and the rotation angle.

The **Insert** option can serve to reinsert an external reference that has been deleted accidentally.

## XInsert Command

The **XInsert** command combines the **XLink** command with the **Attach** option in a single step. The command is provided for convenience of allowing you to attach an external drawing directly to the current drawing.



## GROUPING ENTITIES

Drawing entities can be combined into *groups*. Groups allows you to manipulate all entities within the group in a single operation, such as with the **Move**, **Rotate**, **Mirror**, and **Scale** command. You can think of groups as being a named selection set.

Single entities of a group may still be edited individually, for example to stretch one entity. 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.

## Parts and Groups

A part (block) has its own insertion point; a group has none. A part can occur many times within the drawing; there is just one occurrence of the group. If a block is redefined, all of its instances are updated; groups cannot be treated like that.

Like parts, groups can be copied, although you should use the **Insert** command with parts. A group copied from another group is a separate collection of entities. Parts and groups each have a unique name.

**Note:** If an entity (which is member of a group) is included in a part, the entity is deleted from the drawing and removed from the group at the time that the part is defined.

## GROUP COMMAND

To group entities, or to manage and manipulate groups, use the **Group** command. The command has the following options:

Create	Explode	Rename	Add	Remove	Selectable	List
--------	---------	--------	-----	--------	------------	------

### Create

To create a group, first determine a unique name. The name is required for identifying the group later. To define a new group, follow these steps:

1. Start the **Group** and select the **Create** option.
2. Specify a unique group name to a maximum of 31 characters.
3. If you want, type a description of the group, to a maximum of 64 characters.
4. Decide whether the group should be selectable or not; the default is Yes.
5. Select the drawing objects that should belong to the group.

To create a group at the command prompt:

```
> group
Select option (Create, Explode, Rename, Add, Remove,
Selectable, List): Create
Group name: windows
Description: All windows in the house floor plan
Should the group be selectable (Yes/No) ? y
Select objects: c
First corner: P1
Other corner: P2
  18 selected.
Select objects: [Enter]
*** 18 selected. ***
```

### Exploding Groups

Group definitions are removed by the **Explode** option of the **Group** command. When a group is dissolved, the entry in the group table is deleted in the drawing database. The original drawing entities will return to their previous state within the drawing.

To explode a group, start the **Group** command with the **Explode** option. At the “Group name:” prompt, type the name of the group to be exploded.

**Note:** A single entity may belong to multiple groups, and groups may be parts of other groups. In these two cases, the **Explode** option does not allow you to pick an entity within those groups.

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 the **Group** command’s **Rename** option. This is often used when *anonymous* groups (which have names like \*A1 and \*A2) are created by the **Copy** commands exist. FelixCAD automatically gives anonymous names to groups created by commands such as **Copy**, **Mirror**, and **Array**.

To rename a group, start the **Group** command with the **Rename** option. At the “Current group name:” prompt, type the name of the group, such as \*A0. At the “New group name:” prompt, enter a new name for the group, such as **HouseWindows**.

### Add and Remove Entities

Two options of the **Group** command allow you to add and remove entities from a group. After a group name is specified, entities belonging to the group are highlighted.

- To add entities to an existing group, pick the entities that are to be added (they are *not* highlighted).
- To remove entities from an existing group, pick the highlighted entities that are to be removed

**Note:** If *all* entities are deleted from a group, the group continues to 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 editing commands.

Setting a group as *unselectable* means you are turning off the group definition temporarily so you can apply editing commands to individual entities. You switch between selectable and unselectable at any time.

This option is useful when an entity is in multiple groups and selecting the entity would make it difficult to pick the correct group to work with.

### Working with Groups

Editing commands treat all entities in a group as a single object. Commands like **Move**, **Rotate**, and **Copy** act on the entire group. When you select a single object at the “Select objects:” prompt, you select all group entities.

When you are prompted to select multiple objects, you may choose the **Group** object selection option from the option bar, and enter a group name. This selects all of the entities belonging to the group. Groups, therefore, can be treated as named selection sets.

Other modify commands (such as **Offset**, **Lengthen**, **Intersect**, and **Trim**) request a single entity. Anytime you are prompted to select a single entity, the group does not matter. These commands modify the geometry of single entity, even though the entity belongs to a group. The modified entity remains a member of the group.

The following table summarizes the editing commands and describes how entities that are members of a group will be processed when using each command.

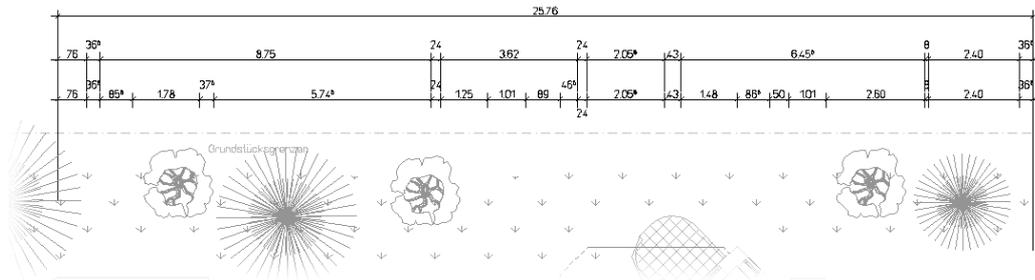
Command	Comments
DELETE	Selecting a single entity selects all entities of the group, unless you use the Single object selection mode. When a single entity of a group is deleted, the entity is removed from the group definition.
MOVE ROTATE SCALE FLIP	Selecting a single entity selects all entities of the group, unless you use the Single object selection mode.
MIRROR COPY ARRAY	Selecting a single entity selects all entities of the group, unless you use the Single object selection mode. FelixCAD gives anonymous names to copied groups, such as *A1 and *A2.
STRETCH	Entities belonging to a group selected with the Crossing or Cpolygon object selection modes are stretched.
TRIM EXPAND	The prompt to “Select cutting edges” or “Boundary edges” includes all entities of a group, unless you use the Single object selection mode.
OFFSET FILLET CHAMFER LENGTHEN INTERSECT	These commands always prompt you to select a single entity to be modified. The group is ignored. Single entities altered by these commands remain in the group. If new entities originate from these commands, they are not 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 remains in the group. When an entity is rejoined, the newly created entity is no longer member of a group, even if one or more of the pieces were part of a group.
PARTDEF HATCH HATCHEDIT	Selecting a single entity selects all entities of the group, unless you use the Single object selection mode.  This also applies to the <b>HatchEdit</b> command, 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 with its group after the <b>TextEdit</b> command modifies the text.



# Dimensioning

This chapter describes how to create dimensions with FelixCAD. *Dimensioning* is a crucial element of most technical drawings, so much so that without an exact and standardized method of dimensioning, a drawing may lose its practical value.

For this reason, FelixCAD's dimensioning commands enable fast, easy, precise, and standardized method of dimensioning.



## Associative Dimensioning

*Associative dimensioning* means that when you change the size of any object, the associated dimension automatically updates itself. Associative dimensioning causes any dimensioning to be recalculated, and the new dimension displayed, as soon as an object has been modified.

For associative dimensioning to work, it relies on the position and placement of *measurement points*.

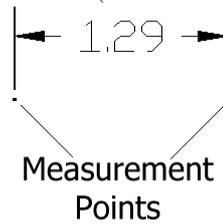
## Measurement Points

The basis of associative dimensioning in FelixCAD is the definition and insertion of *measurement points* (known as defpoints in other CAD software). To draw a dimension, FelixCAD 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.

FelixCAD automatically places *measurement points* when you place a dimension into the drawing. The small dots are placed at the end of the dimension's extension line (see figure below). Measurement points are placed

on layer “DEFPOINTS”. (FelixCAD automatically creates this layer the first time you use a dimension command.)

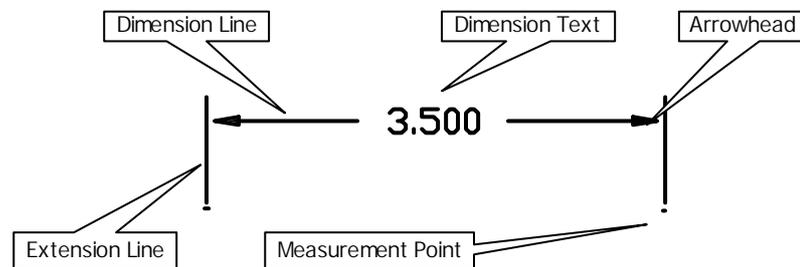
### Associative Dimension



When you change an object (such as stretch it), make sure you include the ends of the dimension.

## DIMENSION ELEMENTS

A dimension commonly consists of the following elements:



### 1. Dimensioning Text

The *dimension text* is usually (but not always) the distance being measured.

### 2. Dimension Line with Arrowheads

*Dimension lines* connect the points of measurement, or the extension lines to those points of measurement. Typically, the dimension text is placed in or near the dimension line.

Dimension lines usually end with *arrowheads*. These arrowheads can be replaced with an oblique or any other symbol acceptable to the drawing.

### 3. Extension Lines

*Extension lines* extend the point of measurement to the dimensioning lines.

For diameter or circumference dimensioning, extension lines start at the dimension value and point to the outer edge of the object.

### Alternative Measurement Units

Alternative units of measurement can be displayed with the dimensioning text. This allows the dimension to show two different units of measurement. A common example is to show both metric and imperial values.

### Tolerances and Fit

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

There are several types of dimension that you can use. Typically, the type of dimensioning is dictated by its position in the drawing. FelixCAD has accommodated nearly all positions imaginable.

#### Linear Dimensions

*Linear* dimensioning shows horizontal, vertical, and rotated distances. Horizontal and vertical dimensions are the most common in drawings; rotated dimensions are used to measure the “true” length of a sloped or angled object.

#### Coordinate Dimensions

*Coordinate* dimensions represents a distance between a specific point and its origin in the coordinate system. The measurement can be made on either the X or Y axis. It is often used in surveying and machining drawings.

#### Diameter and Radius Dimensions, and Center Mark

*Diameter* dimensions are used to measure the diameter of circular objects, such as holes and arcs. *Radius* (or *radial*) dimensions measure the radius of circular objects. The *center* dimension places several different types of center mark at the center of a circular object.

#### Angle

*Angle* dimensions measure the angle between two objects, or the angle inside a single object.

#### Baseline and Continuous Dimensions

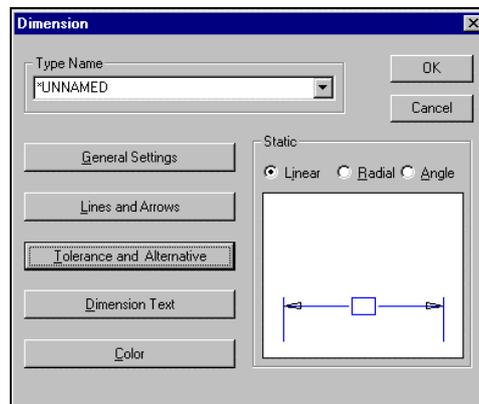
*Baseline* dimensions continues a line are dimension; additional dimensions use the same starting point as the first dimension.

*Continuous* dimensions create a chain of dimensions. All dimensions in the “chain” are in one row, the first dimension in the row dictating the direction in which the chain will be organized (horizontal, vertical, aligned or rotated.)

## DEFINING DIMENSIONS

You can start dimensioning with FelixCAD in your very first drawing. If your drafting discipline, however, has specific dimension standards, you set them with the **Dimension Type** dialog box. This dialog box lets you create a named dimension style, which stores all of your dimensioning preferences.

The **DimType** command (**Dim > DimensionType**) displays the **Dimension Type** dialog box. To create a new dimension style, simply type the name (such as ArchProj) in the **Type Name** text entry box. Then, make changes to the dimension parameters by clicking the buttons, such as **General Settings** and **Line and Arrows**.



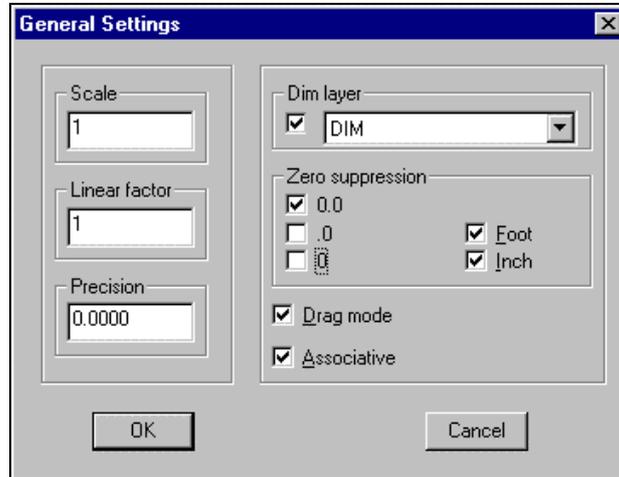
The buttons open additional dialog boxes, in which dimensioning parameters are defined.

## General Settings

The **General Settings** button displays the **General Settings** dialog box.

### Scale

The **Scale** enlarges or reduces the size of all dimensioning elements (apart from the dimension value itself). It allows the dimensions to be scaled to suit the scale of the drawing.



### Linear factor

The **Linear factor** converts the measured drawing units into a dimension figure. The default value is 1. Use this to convert metric to imperial.

### Precision

The **Precision** defines the number of decimal places to displayed. See “Precision and Units Settings”.

### Dim Layer

**Dim Layer** specifies the layer on which the dimensioning values are stored. Select the default layer, “DIM”, or enter a new layer name. A check mark indicates the layer name will be used automatically for dimensions.

## Zero Suppression

The **Zero Suppression** area specifies whether or not zeroes are displayed before or after decimal points, and whether zero feet and zero inches are displayed.

## Drag Mode

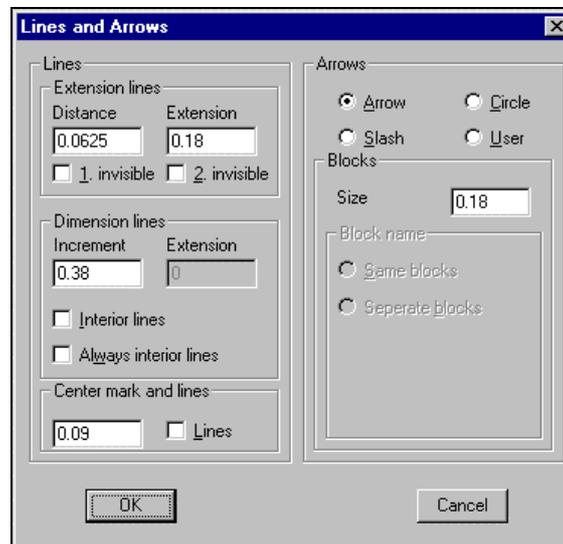
The **Drag Mode** check box determines whether the dimension is ghosted when you drag it into place.

## Associative

The **Associative** checkbox determines whether dimensions are placed as associative or non-associative dimensions.

## Lines and Arrows

The **Lines and Arrows** button displays the **Lines and Arrows** dialog box, which controls lines and the nature of the arrowheads.



## Lines

The **Lines** area determines the characteristics of the extension and dimension lines.

### Extension Lines

The **Extension Lines** section defines the nature of extension lines.

**Distance** determines space between the object and the point at which the extension lines begin. A value of 0 means the lines start directly on the object itself.

**Extension** dictates how far the extension lines should extend past the dimensioning line.

The **Invisible** check boxes are used to suppress either the first and/or the second extension line.

#### **Dimension Lines**

The **Dimension Lines** area defines the spacing and character of the dimension lines.

**Increment** sets the spacing between successive dimensioning lines that use the same extension lines.

**Extension** allows you to enter the distance that the extension lines should run past the dimensioning lines. (This value is meant for use with the **Slash** option of the arrow type.)

The **Interior Lines** and **Always Interior Lines** check boxes specify whether the dimensioning line is allowed to extend past the extension lines.

#### **Center Mark and Lines**

The **Center Mark and Lines** area determines the size of the cross mark, which indicates the center point of a curve. When zero, no center mark is drawn.

The **Lines** check box determines whether extension strokes are added to the center mark.

#### **Arrows**

The **Arrows** area specifies the type of arrowhead to be used.

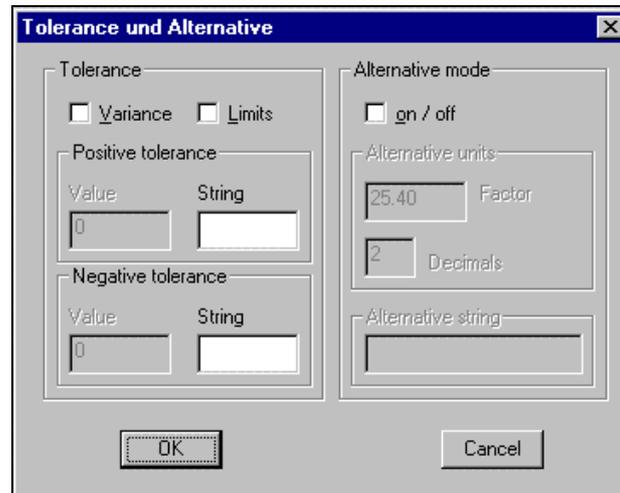
#### **Blocks**

The **Size** item determines the length of the arrow. The default length is 0.18 inches.

**Block Name** is active when the arrow style selected is **User. Separate Blocks** allows the use of different arrow blocks for each end of the dimension line.

## Tolerance and Alternative

The **Tolerance and Alternative** button displays the **Tolerance and Alternative** dialog box.



### Tolerance

The **Tolerance** section specifies the tolerance (maximum and minimum values) to be displayed with the dimension text. Values for the positive or negative tolerance are entered in **Values**. The **String** fields allow you to type in a line of text that is displayed with the tolerance.

### Alternative Mode

The **Alternative Mode** section control second, alternative dimensioning. When on, this mode allows dual dimensioning of metric and imperial values.

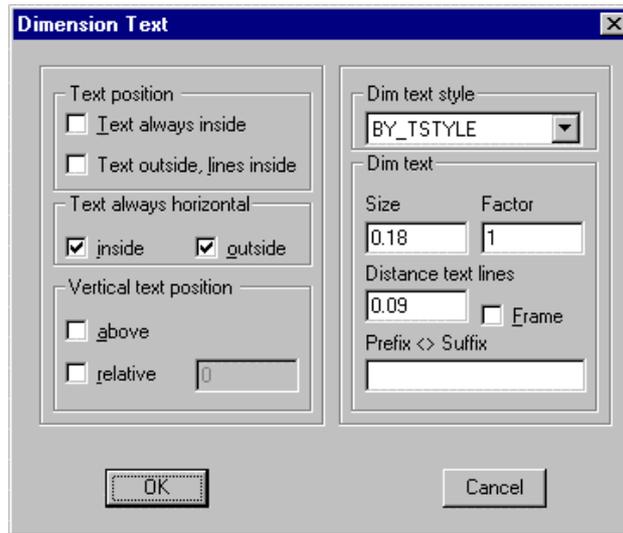
The **Factor** field contains the conversion factor between the two units of measurement, such as 25.4 for metric conversion.

The **Decimals** field specifies the number of decimal places to be displayed by the alternative dimensions.

**Alternative string** should contain a description or title of the second unit of measurement.

## DIMENSION TEXT

The **Dimension Text** button displays the **Dimension Text** dialog box. This dialog box determine the position and look (size, style, spacing) of the dimension text.



### Text Position

The **Text Position** area is used when you want to force dimension text to a specific location.

### Text Always Horizontal

The **Text Always Horizontal** options forces dimension text to always be horizontal, no matter the angle of the dimension line.

### Vertical Text Position

The **Vertical Text Position** options determine the vertical location of the dimension text with respect to the dimension line.

### Dim Text Style

The **Dim Text Style** option select a text style for the dimension text. The text style must be already defined by the **Font** command.

### Dim Text

These fields define the size of the dimension text and the distance separating text lines. The **Prefix** <> **Suffix** field enables you to type in a character string that appears before or after the dimensioning text.

**Note:** The < > metacharacter is a placeholder for the dimension text calculated by FelixCAD. For example, assuming the dimension is 12, **Approx. <> ft.** entered in this field produce the following dimension text:

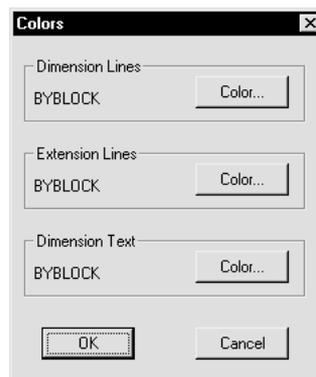
Approx. 12 ft.

### Colors

The **Colors** button displays the **Colors** dialog box. It allows you to assign colors to:

- Dimensioning lines.
- Extension lines.
- Dimension text, including the value, alternative dimensions, tolerances, limit values, and unit of measurement.

Clicking the **Color** button displays the standard **Colors** dialog box. You may choose any of the 255 colors, as well as BYLAYER and BYBLOCK.

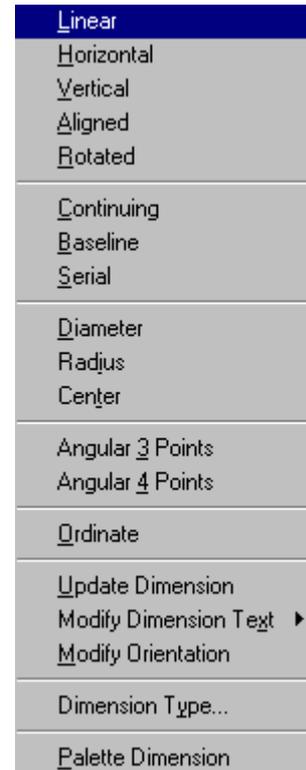


## CREATING DIMENSIONS

To create dimensions in your drawing, you can select dimension commands from:

- The **Dim** menu.
- The **Dimensioning** palette (toolbox).
- **Detail > Dimensions** and the **Create Dimensions** palette.
- Typing **Dim**-related commands, as listed below.

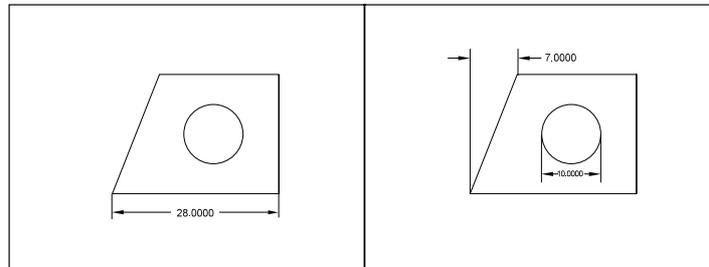
Dim Command	Meaning
DimA3p	Angle 3 point
DimA4p	Angle 4 point
DimAli	Aligned dimension
DimBas	Baseline dimensioning
DimCent	Center mark
DimCon	Continuous dimensioning
DimDia	Diameter dimension
DimHor	Horizontal dimension
DimLin	Linear dimension (Combination between DimHor and DimVer)
DimOrd	X or Y coordinate dimension
DimRad	Radius dimension
DimRot	Rotated dimension
DimSer	Serial dimensioning
DimVer	Vertical dimension



## Horizontal Dimensioning

Horizontal dimensioning is a form of linear dimensioning that measures the distance between two points parallel to the X-axis.

```
> DIMHOR
First Point: P1
Second Point: P2
Dimension line positioning: P3
Dimension text <2.50>: [Enter]
```



Once you have picked the two points of measurement, FelixCAD asks you to pick a position for the dimensioning line.

The distance between the two points is measured along the X-axis and displayed.

Press the **Enter** key to verify the calculated value and insert the dimension into the drawing.

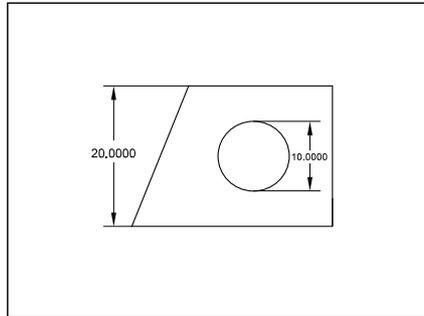
(As an alternative, you may dimension a selected object by choosing the **Select** option. You may dimension lines, polylines, circles, and arcs with the **Select** option. Pick the object with the cursor. The start and end points -- or the diameter -- of the object are dimensioned automatically.)

```
> DIMHOR
First Point: SELECT
Select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.5000>: [Enter]
```

## Vertical Dimensioning

Vertical dimensioning is a form of linear dimensioning that measures the distance between two points parallel to the Y-axis.

```
> DIMVER
1. Point: P1
2. Point: P2
Dimension line positioning: P3
Dimension text <2.50>: [Enter]
```



Once you have picked the two points of measurement, FelixCAD asks you to pick a position for the dimensioning line.

The distance between the two points is measured along the Y-axis and displayed.

Press the **Enter** key to verify the calculated value and insert the dimension into the drawing.

(As an alternative, you may dimension a selected object by choosing the **Select** option. You may dimension lines, polylines, circles, and arcs with the **Select** option. Pick the object with the cursor. The start and end points -- or the diameter -- of the object are dimensioned automatically.)

```
> DIMVER
1st Point: SELECT
select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.50>: [Enter]
```

## Linear Dimensioning

The DIMLIN command creates horizontal or vertical dimensioning based on the position of the dimension line. This command is a combination of both the horizontal and vertical orthogonal modes.

```
> DIMLIN
First Point: P1
Second Point: P2
Dimension line positioning: P3
Dimension text <2.50>: [Enter]
```

Sample: Pick two points of measurement. FelixCAD asks you to pick a position for the dimensioning line. FelixCAD displays a preview of the dimension, based on the location of the dimensioning line.

The distance between the two points is measured along the X-axis for horizontal and Y-axis for vertical dimensioning and displayed. Press the **Enter** key to verify the calculated value and insert the dimension into the drawing.

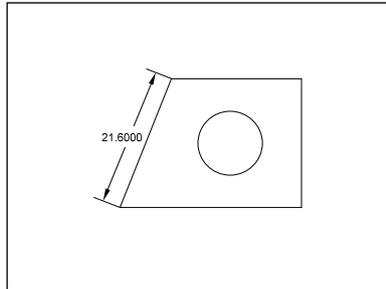
(As an alternative, you may dimension a selected object by choosing the **Select** option. You may dimension lines, polylines, circles, and arcs with the **Select** option. Pick the object with the cursor. The start and end points -- or the diameter -- of the object are dimensioned automatically.)

```
> DIMLIN
First Point: SELECT
Select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.5000>: [Enter]
```

## Aligned Dimensioning

Aligned dimensioning measures the absolute distance between two points, regardless of the relative position of the axes. This makes it possible to determine the true length of lines and edges that are not precisely horizontal or vertical.

```
> DIMALI
1st Point: P1
2nd Point: P2
Dimension line positioning: P3
Dimension text <2.50>: [Enter]
```



Once you have picked the two points of measurement, FelixCAD asks you to pick a position for the dimensioning line.

The distance between the two points is measured along the two points and displayed.

Press the **Enter** key to verify the calculated value and insert the dimension into the drawing.

(As an alternative, you may dimension a selected object by choosing the **Select** option. You may dimension lines, polylines, circles, and arcs with the **Select** option. Pick the object with the cursor. The start and end points -- or the diameter -- of the object are dimensioned automatically.)

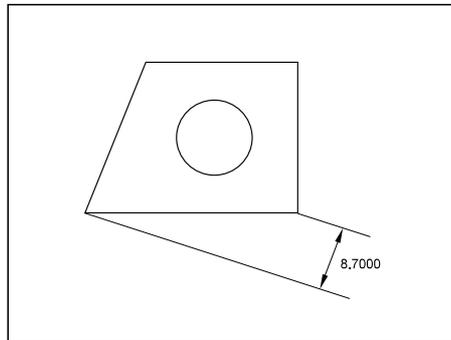
```
> DIMALI
1st Point: SELECT
select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.5000>: [Enter]
```

## Rotated Dimensioning

Rotated dimensioning is another type of linear dimensioning that measures the distance between two points at a user-defined angle. This procedure is equivalent to looking at an object from a specific angle and establishing the visible length from a different viewpoint.

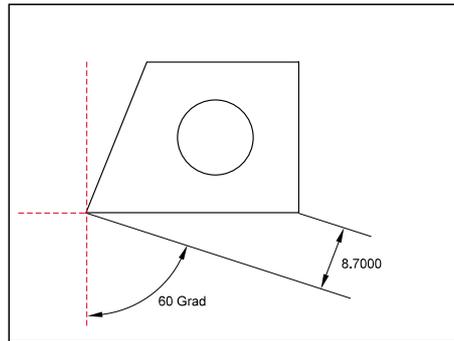
When dimensioning the distance between two points at an angle, the distance calculated for the rotated dimension will always be smaller than the absolute dimension measurement. There is one exception: this method always measures the exact diameter of a circle. Only the position of the extension lines is affected when using rotated dimensioning.

```
> DIMROT  
Angle of rotation <0>: 60  
First point: P1  
Second point: P2  
Dimension line positioning: P3  
Dimension text <2.5000>: [Enter]
```

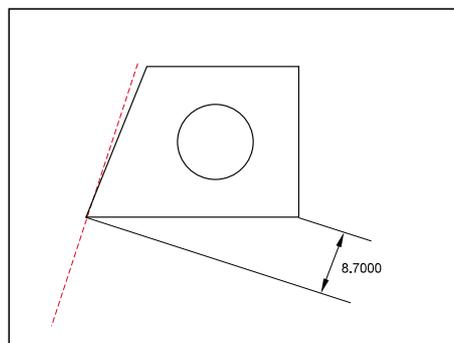


You are asked to define the angle at which the dimensioning is to appear. This angle is related to the axial alignment of the coordinate system. You can type a value or use the cursor to point.

The figure below shows 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 are prompted to define two points of measurement.

Once the points of measurement are chosen, you are prompted to define the position for the dimensioning line. The distance between the two points is displayed at the designated angle orientation. Press the **Enter** key to accept the calculated value.

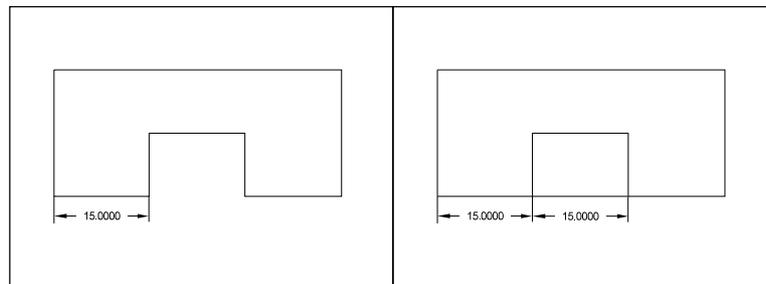
(You may also select a specific dimensioning object function from the option bar by choosing the **Select** option. The start and end points of a line, polyline, or arc will be interpreted automatically and used for the measurement calculation.)

```
> DIMROT
Angle of rotation <0>: 60
1st Point: SELECT
Select line, polyline, circle or arc: P1
Dimension line positioning: P2
Dimension text <2.5000>: [Enter]
```

## Continued Dimensioning

Continued dimensioning is a variation of linear dimensioning that continues an existing dimensioning: a second dimension is linked to an existing dimension, creating 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. For this reason, you can only connect dimensioning of the same type. For example, a new horizontal dimension can only be added to an existing horizontal dimension.



```
> DIMCON
Select a dimension: P1
2nd Point: P2
Dimension text <2.5000>: [Enter]
```

The command prompts you to select an existing dimension. When selecting the dimension, remember to make your selection as close as possible to the point at which the dimension is to be continued.

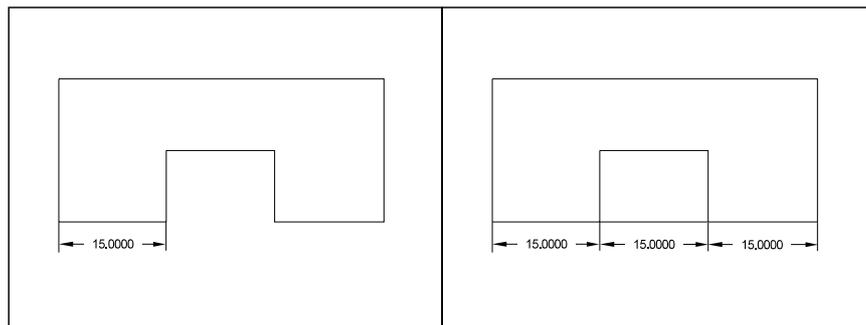
The continued dimensioning is based on just one point of measurement. The starting point for the continued dimension is the end of the dimension closest to your pick point. Choose the second point by entering coordinates or by selecting with the cursor. We recommend you use object snaps.

The command displays the calculated dimension. This value can either be altered, or confirmed with **Enter**. You are not asked to position the dimensioning line, since this is done automatically by FelixCAD.

### Serial Dimensioning

Serial dimensioning is similar to continued dimensioning. Whereas only one continued dimension can be added to an existing dimension, serial dimensioning lets you 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.

```
> DIMSER
Select a dimension: P1
Series <continuous> [Enter]
2nd Point: P2
Dimension text <2.5000>: [Enter]
Series <continuous>: [Enter]
2nd Point: P2
Dimension text <2.5000>: [Enter]
Series <continuous>: [Esc]
```

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 the **Select** and **End** options, which are the same as for continued dimension, two other options may be made from the **Series Dimensioning** option 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.

**Baseline** allows you to create baseline dimensioning parallel to the existing linear dimensioning. (This is described in the next section.)

Select the second point by typing in the coordinates or selecting with the cursor. Use object snap whenever possible.

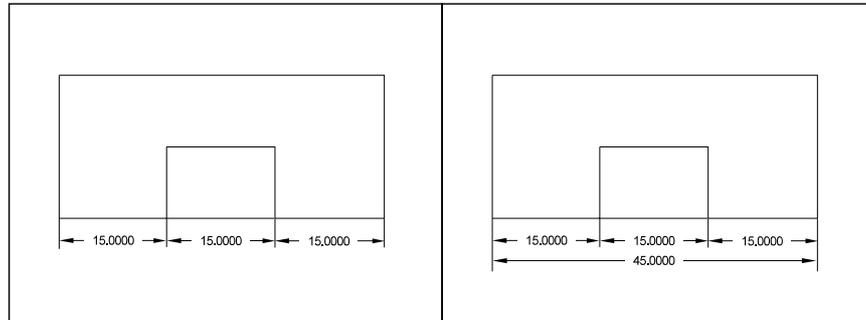
The program displays the calculated dimension. You may change this value or confirm it by pressing **Enter**. You are not asked to position the dimensioning line, since it is placed automatically in line with the existing dimensioning.

The command repeats itself. Terminate the command by pressing the **Esc** key.

## Baseline Dimensioning

Baseline dimensioning adds a second dimension parallel to an existing dimensioning line. This allows you to read the cumulative dimension between two end points covered by a number of individual dimensions within a chain.

Baseline Dimensioning assumes that another linear dimension already exists. The baseline dimension assumes the same properties as the existing dimension.



### DIMBAS

Select a dimensioning line **P1**

2nd Point: **P2**

Dimensioning text <20.50>: [**Enter**]

Select the starting point of the dimensioning chain, taking care to use the same direction as placed during the creation of the original dimension.

Select the second point to be used for the baseline dimension. Enter the coordinates at the keyboard or pick a point with the cursor. Whenever possible, use object snap for accuracy.

The command displays the calculated dimension. This value can either be altered, or confirmed with **Enter**. You are not asked to position the dimensioning line since the second dimension is drawn at a preset distance from existing dimensioning lines.

## Ordinate Dimensioning

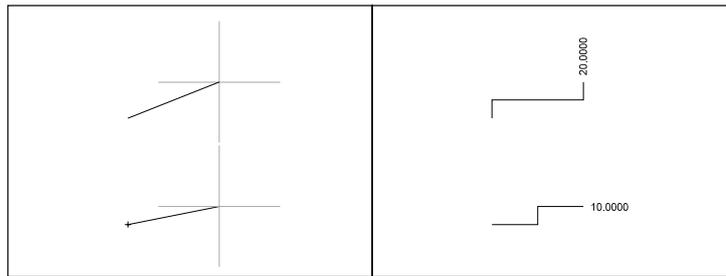
Ordinate dimensioning is used to determine the X and Y coordinates of a point in the drawing.

```
> DIMORD
Coordinate point: X
Coordinate point: P1
Extension length: P2
Dimensioning text <20.00>: [Enter]
```

The **X** and **Y** options are displayed on the option bar. Selecting one option to specify whether you want the X or Y coordinate value placed. The X-value is the default and will be used if the Y option is not selected.

You are prompted to select the point to be measured. Type in the coordinate values, or select with the cursor. If the point is a geometric point on an entity, use object snap.

When the command prompts for “Extension length:”, indicate the extension line position. This can be positioned at any alignment and distance from the dimensioning line. To help, the command draws a line from the selected coordinates to the current cursor position. This illustrates the distance and alignment between the two positions. Select the 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 is drawn in orthographic mode, and not as an angled line. An example of this is shown in the above figure.

The program displays 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 labeled as required.

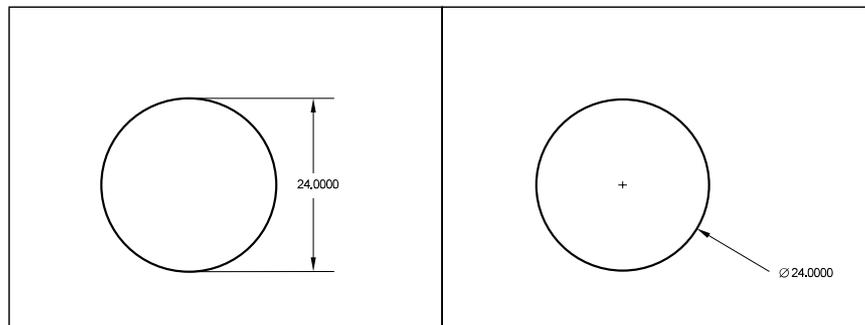
If a 0,0 point is referenced on different views of the same part, care should be taken to ensure the same 0,0 point is referenced. This is not so much for the software (FelixCAD will allow ordinate dimensioning from different reference points) as it is for producing accurate part drawings for manufacture.

## Diameter Dimensioning

Diameter dimensioning is used to indicate the diameter of circles, arcs, and other curves. Unlike the horizontal or vertical dimensioning, there is no dimension line. Instead, a single extension line shows the diameter of the curved object. The diameter symbol is  $\varnothing$ .

```
> DIMDIA
Select circle or arc: P1
Dimensioning text <24.00>: [Enter]
Extension length: P2
```

The first prompt asks you to select a circle or an arc. You may also select a polyline that contains arcs. Select carefully with the cursor: the point you pick is where the extension line connects to the object.



We recommend that you use the QUAdrant object snap to precisely select the 90-degree points on the circle or arc. The CENter object snap selects the precise center of the curve.

The command display the diameter is measured. Confirm by pressing the **Enter** key or edit the text, as required.

The final prompt asks you to position of the extension line. Select a point at a reasonable distance from the object. The alignment of the extension line is fixed by the two earlier pick points and cannot be altered. The extension line is

drawn straight along an imaginary line that extends from the center of the object to the pick point.

A cross mark (+) is placed at the center of the circle of arc, unless you turn off that feature with the **DimType** command.

## Radius Dimensioning

Radius dimensioning is the same as diameter dimensioning, except that the radius is calculated.

```
> DIMRAD  
Select circle or arc: P1  
Dimensioning text <24.00>: [Enter]  
Extension length: P2
```

## Center Mark

The center mark is used with curves. It places a mark (+) at the center of a diameter or radius.

```
> DIMCENT  
Select circle or arc: P1
```

The command prompts you to select the circle or arc. You may also select a polyline arc.

If a circle or arc is correctly selected, the center mark appears. If not correctly selected, you are asked to select the object again.

The **DimCen** system variable allows you to change the size of the center mark.

## Angle Dimensioning

FelixCAD contains commands for fast and accurate angular dimensioning. They are called **DimA3P** (short for angular 3-point) and **DimA4P** (angular 4-point).

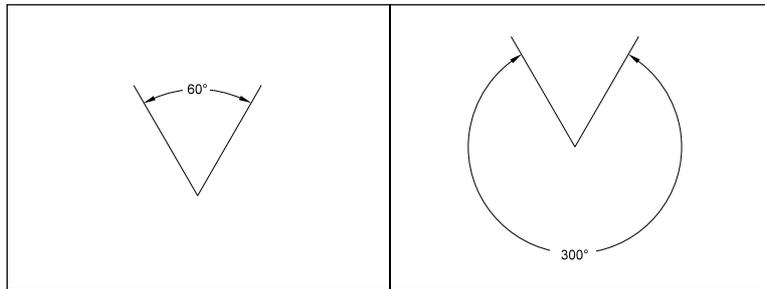
### Angular 3P

The **DimA3P** command uses three pick points: one at the apex, and one each on the legs of the angle. This command finds both the interior angle and the exterior angle.

```
> DIMA3P
Apex: end
of: P1
1st Point: P2
2nd Point: P3
Position measurement curve: P4
Dimensioning text <60°>: [Enter]
```

When prompted “Apex:” pick the vertex position. We recommend that you use **INT**ersection object snap whenever possible.

Pick a point on each leg of the angle. Specify the location of the dimension arc. The dynamic preview assists you in locating the best position. **Note:** The position you select determines whether the interior or exterior angle is dimensioned.



After confirmation, the program will display the measured value. Confirm with the **Enter** key, or change as required.

### Angular 4P

The **DimA4P** command is very similar to the **DimA3P** command, except 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. Either the interior or the exterior angle can be dimensioned.

The first two prompts 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°>: [Enter]
```

## MODIFYING DIMENSIONS

FelixCAD contains commands that allow you to edit dimensions. The commands are found in the **Modify** section of the **Dim** menu, as well as under **Detail > Modify Dimensions**.

### Modify Dimensions

The **Modify Dimension** palette allows you to move the text on the dimension line in a number of ways, each illustrated by an icon. Another icon allows you to reach the **Dimension** detail dialog box, while another icon displays a detail editing palette from the main palette.

Dim Command	FUNCTION
DimUpd	Update dimension.
DimTRot	Rotate dimensioning text.
DimTNew	Change dimension text.
DimTMove	Move dimension text.
DimTPos	Undo repositioning.
DimObl	Modify dimension orientation.

## UPDATE DIMENSION

After changing the dimensioning style, all new dimensions are created with the new criteria. The **Update Dimension** command updates all of the existing dimensions so that they comply with the new dimensioning style settings.

```
> DIMUPD
Select Dimension(s): Fence
First Fence point: P1
Endpoint of the line: P2
Endpoint of the line: P3
[Enter]
Select Dimension(s): All
2 Dimensions found

or
DIMUPD
Select Dimensions(s): All
Select Dimensions(s) [Enter]
3 Dimensions found!
```

You may update individual, a selection set, or all dimensions contained in the drawing at a time. Select the dimensions to be updated, using one of the object selection functions found in the option bar. Confirm the selection process by pressing **Enter**.

The selected dimensions are updated to conform to the new dimension style. The command displays the “x dimensions found” notice, telling you how many dimensions have been updated.

## CREATING A DIMENSIONING STYLE

At the beginning of this chapter, we described how to create customized dimensions with the **DimType** command. The **Dimension** dialog box lets you define the size, text font and height, alignment and positioning of the dimension text, the look of the arrows and extension lines, as well as other parameters. These parameters are called a *dimension style* or “dimstyle” for short.

The **DimSave** command saves the current dimension settings as a named dimensioning style. This lets you to store and recall a number of different styles. Architectural dimensioning, tolerance dimensioning, or country- and language-specific styles are reasons for being able to create, save, and load dimstyles.

**Note:** dimensioning styles are only stored with the current drawing. To share dimstyles with other drawings, you should create the dimstyle in a template drawing. Templates are described in Chapter 1 “Getting Started: The Basics of Drawing.”

### Saving a DimStyle

Select **Detail > Modify Dimension** from the menu. From the palette, select the icon representing the **Dimension** dialog box. 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 alphanumeric input except special symbols. No differentiation is made between upper and lower case characters.

Confirm the selected name to store the dimensioning style.

You can list all of the names of all dimension styles, as follows:

```
> DIMSAVE
Name of the dimensioning style: List
List of dimensioning styles <*>: ar*
ARCHITECT01
ARCHITECT02
```

Listing the existing dimensioning styles completes the save dimensioning style command. Press **Enter** to restart the command and save the current dimensioning settings.

```
> DIMSAVE
Name of the dimensioning style: ARCHITECTURE03
```

## Restoring a Dimstyle

The **DimRest** command allows you to restore a dimstyle.

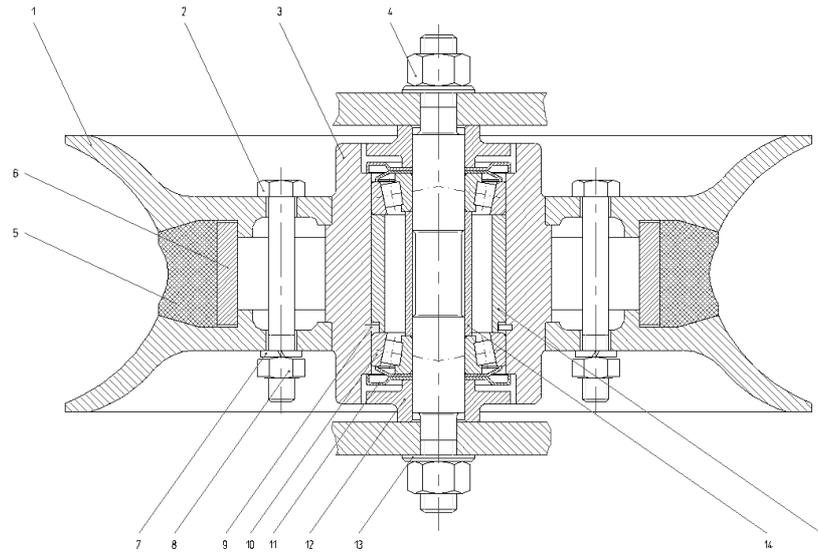
The command displays the names of the current dimensioning style. You are prompted to choose a dimensioning in the drawing, whose style will be set as the current dimstyle.

```
> dimrest
Set dimension type by reference dimension.
Select object: [pick]
```

Any dimensions were created prior to the definition of the new style can be updated with the **DimUpd** command so that they adhere to the new style settings.

# Hatching

This chapter discusses the process of hatching an area, a section of a drawing, and the cross sectioning of a part. Applying hatching or area fills using distinctive patterns increases the clarity and legibility of a drawing. Hatching (or cross-hatching) an object adds meaning to the drawing, and helps differentiate between materials, areas, or other unique attributes. Many disciplines specify norms and specifications that demand the correct use of specific hatch markings within a drawing.



The **Hatch** and **BHatch** commands (**Detail > Hatch** and **Detail > Boundary Hatch**) enable you to use pre-defined hatch patterns to fill selected areas of the drawing. With the aid of the **Cross Hatching** dialog box, you select the hatch pattern from a list, and view a sample 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.

To use the associative character, the hatch pattern must be attached to objects in the drawing.

## Hatch Pattern Definitions

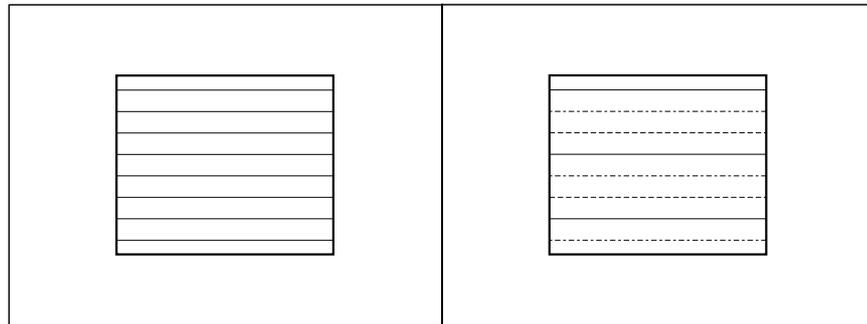
A hatch pattern is comprised of a pattern of line segments. FelixCAD includes a number of typical hatch patterns used by various disciplines. You may choose to use other hatch patterns for specialized circumstances.

Hatch patterns are stored in files with the *.pat* file extension in the *\applic* folder.

## Hatch Pattern Elements

### Hatch lines

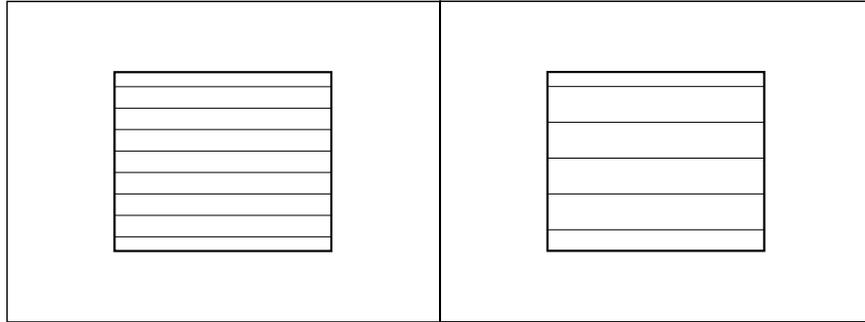
A hatch pattern can contain a number of diverse line types, such as solid, dashed, and dotted:



A variety of lines are used to 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 (1.0) drawing unit. Changing the line spacing is possible to adjust a pre-defined hatch pattern to suit the proportions of any given drawing element.

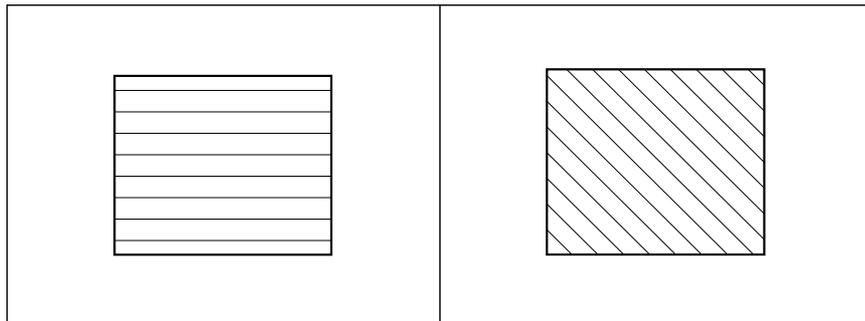


Hatch patterns with different line spacing.

**Hatching Angle**

The lines of a hatch pattern can be drawn at any angle. Standard hatch patterns are often drawn horizontal, vertical, or at 45 degrees, but may be drawn at any angle.

The hatch angle of  $0^\circ$  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.



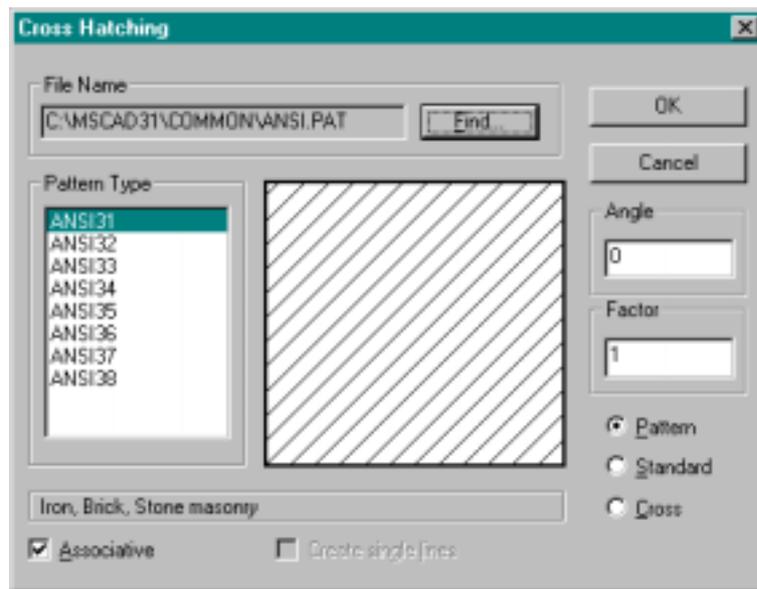
Hatch pattern drawn at a  $0^\circ$  angle (left) and one at a  $-45^\circ$  angle (right)

**HATCH COMMAND: ADDING HATCH MARKS TO AN OBJECT**

Select **Hatch** from the **Detail** menu, or type the **Hatch** command at the prompt.

The command prompts you to select an object. The Option Bar contains a list of object selection functions. Select the object(s), then terminate object selection process by pressing **Enter**.

The **Hatch** command opens the **Cross Hatching** dialog box, which allows you to select a pre-defined hatch pattern, change its parameters, or to create your own pattern. In the center of the dialog box, the preview function lets you see the effect of your choices.



Select the pattern and confirm your selection with **OK**.

If you do not see the hatch pattern you need to use, click the **Find** button to select another hatch *.pat* file.

To change the parameters of an existing hatch pattern, alter the **Angle** and scale **Factor** values, as required. **Factor** affects the spacing of the hatch lines, measured in drawing units.

To create a simple, user-defined hatch pattern, click the **Standard** or **Cross** buttons. This option displays a preview of a standard pattern consisting of continuous horizontal lines spaced at a distance of one drawing unit.

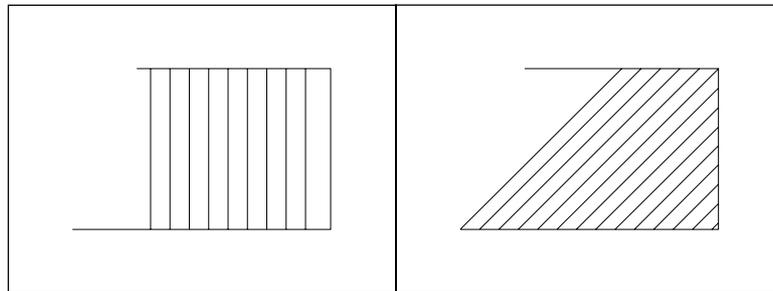
You can change this pattern via the **Angle** and **Factor** parameters. The **Cross** button adds lines at right angles to the existing lines.

The **Associative** button allows the hatch pattern to update automatically when the boundary of the hatch is changed.

To apply the designed hatch pattern to your drawing, exit the dialog box with **OK**.

#### **Note: Hatching Open Areas**

You can hatch open areas (not fully closed) . The hatch pattern is only applied to the extent of the selected contour line area. The following diagram illustrates this procedure.



### **MODIFY HATCHING**

You can change associative hatch patterns with the **HpEdit** command (**Detail > Modify Hatching**).

```
> HPEDIT
Select an associative hatch: [pick]
Select additional elements? <N>: [Enter]
```

The command then displays the **Cross Hatching** dialog box. You can make changes to the hatching parameters.

# Inquiry and Information Programs

The chapter describes commands that provide you with information about the properties of drawing objects, and about the current drawing parameters. FelixCAD provides commands for finding the area, circumference, coordinate identification, distance, and angles. These information commands are found in the **Edit** menu. The commands covered by this chapter are:

Command	Meaning
'Id	Identify X, Y, Z coordinates.
'Dist	Determine the distance and angle between two points.
Area	Report the area and circumference.
Einfo	Display entity information.
Tables	Summarize database table data.

' Indicates a transparent command.

## IDENTIFY COORDINATE



The **Id** command (**Edit > Identify Coordinate**) determines the X, Y, and Z coordinates of a point identified on the screen. (**Id** is short for “Identify.”) Since it is a transparent command, **'Id** can be used in the middle of another command.

```
> `ID
Point: P1
X: 1.27 Y: 4.45 Z: 0.0
```

When prompted by the “Point:” prompt, pick the point with the cursor. We recommend you use an object snap if the point is geometrically significant, such as a corner point, the center of a circle, or the intersection of two lines. The X, Y, Z coordinates are displayed in the command line area, as shown in the example above.

***DISTANCE / ANGLE***

The **Dist** command (**Edit > Distance/Angle**) calculates the distance and angle between two points. (**Dist** is short for “Distance.”) Since it is a transparent command, **'Dist** can be used in the middle of another command.

The two points you pick may be arbitrary points, or they can be part of a drawing object. We recommend you use an object snap if the point is geometrically significant, such as a corner point, the center of a circle, or the intersection of two lines.

```
> `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
```

The command displays the following information:

- Absolute distance;
- Distance along the X, Y, and Z-axis.
- Angle in the XY plane.
- Angle from the XY-plane.

The **Distance** is the absolute distance (shortest-line distance) between the two points.

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.

The **Delta X**, **Delta Y** and **Delta Z** report 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.

**AREA / CIRCUMFERENCE**

The **Area** command (**Edit > Area / Circumference**) finds the area and the circumference of one or more drawing objects. For objects that do not have an area, the length is displayed instead of the circumference. The command also allows you to add and subtract areas from the total area.

To determine the circumference and the area of one or more objects proceed as follows:

```
> AREA
First point: P1
Next point: P2
Next point: P3
Next point: P4
Next point: [Enter]
Area: 0.75 | Length: 3.66
First point: [Enter]
```

1. Determining the circumference (or length) and the area of an object is done by defining points that describe the geometry of the object. The preferred method is to use object snap options to select objects and their boundaries, but you can also input the coordinates of the points.
2. Press **Enter** at the “Next Point:” prompt to finish specifying the area. FelixCAD reports the area and the length.
3. You can compute another area by selecting more points at the next “First point:” prompt, or you can finish the command by pressing **Enter**.

Object	AddArea	SubArea		
--------	---------	---------	--	--

**Object Option**

Regular geometric areas (such as circles, arcs, polylines, and rectangles) may be selected via the **Object** option.

Select the object and confirm the selection by picking **OK**. The values for the circumference and the area appear in the text window.

```
> AREA
First point: Object
Selection of an entity (circle, arc, polyline): P1
Area: 0.87 | Circumference: 3.3
First point: Enter
```

You may continue computing the circumference and area of objects, or terminate the command by pressing **Enter**.

### Add and Subtract Areas

The **Area** command allows you to add or subtract areas from the total area. Select the first area or pick the points that describe the area geometry. The calculated values will be displayed in the text window.

Select the **AddArea** option to add an area, or **SubArea** to subtract an area from the total area.

```
> 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 values (circumference or length and area) for the total area are displayed in the text window.

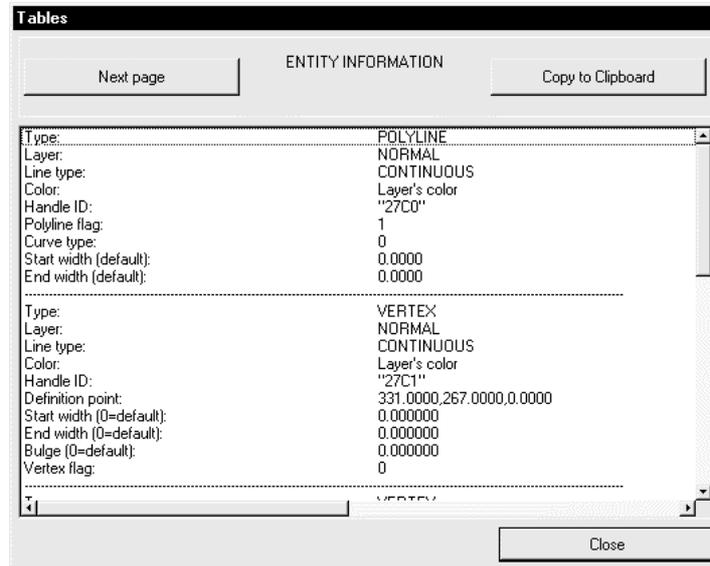
The options bar displays only the options that are not active at the moment. For example, when the **AddArea** option is active, the options bar displays

Object	SubArea			
--------	---------	--	--	--

The value calculated for the area is stored in the **Area** system variable and can be retrieved via the FLISP expression (getvar "AREA") at the command line.

**ENTITY INFO**

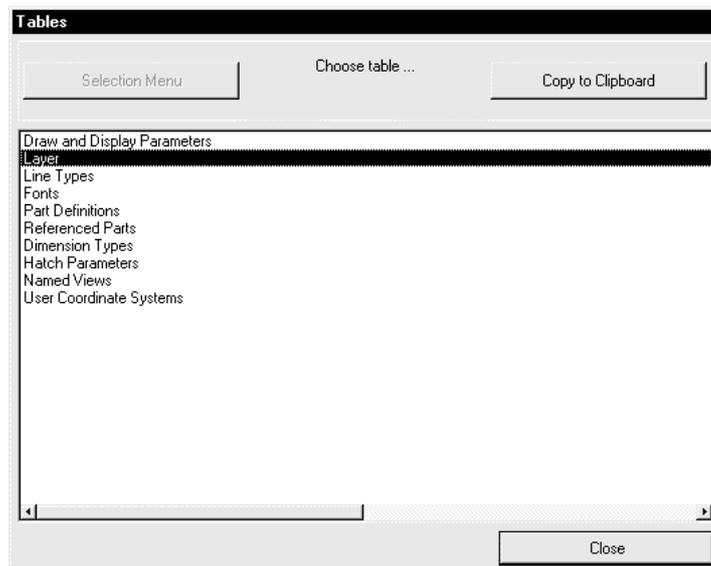
The **EInfo** command (**Edit > Entity Info**) lists information about the data stored for the selected objects. The **Clipboard** button copies the data to the Clipboard.



**DRAWING DATABASE TABLES**

The **Tables** command (**Edit > Drawing Database Tables**) displays information about the tables in the *.flx* file. *Tables* store information about:

- Draw and display parameters.
- Layer.
- Line types.
- Fonts (text styles) .
- Part (block) definitions.
- Referenced parts (blocks).
- Dimensioning types.
- Hatching parameters.
- Named views.
- User coordinate systems.



Select a table name, then click the **Selection Menu** button. This displays a second dialog box, which lists details about the table you selected.

Table Name	Code	Layer
0	CO: 7	LT: CONTINUOUS
TEXT	CO: 4	LT: AUSGEZOGEN
MIT-LANG	CO: 1	LT: MIT-LANG
SCHRAFF	CO: 1	LT: AUSGEZOGEN
BEM	CO: 1	LT: AUSGEZOGEN
FEIN	CO: 1	LT: AUSGEZOGEN
DUENN	CO: 2	LT: AUSGEZOGEN
NORMAL	CO: 7	LT: AUSGEZOGEN
NORMAL_NT	CO: 7	LT: AUSGEZOGEN
SCHRAFF_NT	CO: 1	LT: AUSGEZOGEN
DUENN_NT	CO: 2	LT: AUSGEZOGEN
MIT-KURZ_NT	CO: 1	LT: STRICHPUNKT
USER2	CO: 7	LT: AUSGEZOGEN
USER1	CO: 7	LT: AUSGEZOGEN
POSI	CO: 7	LT: AUSGEZOGEN
INFO	CO: -2	LT: AUSGEZOGEN
VDAPS-GEO	CO: 7	LT: AUSGEZOGEN
PHAN	CO: 5	LT: AUSGEZOGEN
SYM	CO: 1	LT: AUSGEZOGEN
ISO	CO: 1	LT: AUSGEZOGEN
HILF	CO: 3	LT: AUSGEZOGEN
VERECKT	CO: 2	LT: VERECKT

The figure above shows sample output from the Layer table. The **Selection Menu** button takes you back to the first dialog box.

The **Copy to Clipboard** button copies the displayed information to the Windows clipboard.



## Export and Import

One of the principle features of FelixCAD is its ability to interchange information with other CAD and Windows programs.

FelixCAD allows you to export files in a variety of formats. This allows you to edit the drawings using other programs.

Similarly, FelixCAD can import other drawing and graphical formats. This allows you to edit drawings created by other software packages.

It is of prime importance that data be imported and exported without losing any attributes, text information, or any other part of the data, whether graphic or non-graphic.

Import and export command are provided for:

- Internal and external parts.
- Attributes.
- Text entities.

Each of these is explained in this chapter.

*Note: Importing and Exporting a large number of drawings at one time is covered under the BATPROC command.*

## SAVE AS: DXF & DWG \*

There are several official and *de facto* standards for exchange drawings between CAD packages. Two of these file formats are known as .DXF (drawing interchange format) and .DWG (the AutoCAD native file format).

If a FelixCAD drawing needs to be exported as a file with the .*dwg* or .*dxf* extension, it should be saved using the **SaveAs** command (**File > Save As**).

After entering the command, the common Windows file dialog box opens and allows you to specify the drive, path, sub-directory (or folder) and filename.

Select the file format from the **Save as type** list:

- AutoCAD R12 File (\*.dwg)
- AutoCAD R13 File (\*.dwg)
- AutoCAD R14 (supports 14.01) File (\*.dwg)
- DXF File (\*.dxf)

Match the version number (R12, R13 or R14) with the version of AutoCAD receiving the file. For AutoCAD LT, use **R12** for LT R1 and R2; use **R13** for LT 95; use **R14** for LT 97 and 98.

For other CAD programs, it is best to experiment. First, try the **R14** option. If there are too many errors in the translated drawing, try **R13** or **R12** options.

If the software program does not read .*dwg* files, then use the DXF option.

## OPEN: DXF & DWG

If a drawing is to be imported in **DWG** or **DXF** format, use the **Open** command (**File > Open**). Then follow these steps:

1. Select the import format from the **Files of type** list.
2. Type the name of the file to be imported, or select the file from the list, in the **File name** field. (If necessary, enter the exact path of the import file in **Look In**. Remember, if no path is entered, the current folder is used.)
3. Confirm the selection clicking **Open**. FelixCAD may prompt you for the location of font files it cannot find on its own.

## **WMF EXPORT \***

A drawing can be exported in the **WMF** (Windows metafile) format, which is a vector format used mainly to exchange graphics among Windows applications. WMF is preferable to BMP format.

The **WmfOut** command (**Edit > Copy Region to Windows Metafile**) exports the drawing to a file or to the Windows Clipboard. The command presents the following options:

### **Window**

Define a window by picking two points in the drawing.

### **View**

The entire current view is exported as WMF.

### **Extents**

The extents (entire) drawing is exported.

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 copy something else to the Clipboard. In FelixCAD, this is done with the **CopySelect**, **BmpOut**, **WmfOut**, and other commands, as well as the **Cut** and **Copy** commands in other Windows applications.

Exporting a drawing or a part of it as *.bmp* or *.wmf* lets you use the drawing in presentation programs that create slide shows, such as Microsoft PowerPoint, Corel Presentations, or Lotus Freelance Graphics. Such software packages allow you to add logos, textual descriptions, special effects, or other artwork to create professional slide shows.

**BITMAP EXPORT \***

A drawing can be exported in the **BMP** (Windows bitmap) format, which is a raster format used mainly to exchange images among Windows applications.

The **BmpOut** command (**Edit > Copy Region to Bitmap**) exports the drawing to a file or to the Windows Clipboard. The command presents the following options:

**Window**

Define a window by picking two points in the drawing.

**View**

The entire current view is exported as BMP.

**Extents**

The extents (entire) drawing is exported.

**Size**

You are asked to specify the width and height of the bitmap in pixels. Unless you have a good reason for changing this value, always accept the default. The range is from 10 to 2,000 pixels.

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 copy something else to the Clipboard. In FelixCAD, this is done with the **CopySelect**, **BmpOut**, **WmfOut**, and other commands, as well as the **Cut** and **Copy** commands in other Windows applications.

Exporting a drawing or a part of it as *.bmp* or *.wmf* lets you use the drawing in presentation programs that create slide shows, such as Microsoft PowerPoint, Corel Presentations, or Lotus Freelance Graphics. Such software packages allow you to add logos, textual descriptions, special effects, or other artwork to create professional slide shows.

## CREATING AND VIEWING SLIDES

The **SldOut** (or **MSlide**) command (**Edit > Make Snapshot**) lets you save a view of the current drawing as a *slide*. Creating a slide file is like capturing the screen of a drawing. You can view the slide later with the **VSlide** command.

The extension of slide files is *.sld*.

Slide files can be used as an alternative to WMF and BMP files (see the **WmfOut** and **BmpOut** commands earlier in this chapter). You use slide files in icon menus, which are called by the **IconMenu** command.

You can collect slide files in *image libraries*. Image libraries have an extension *.ilb*. Images from image libraries (WMF, BMP, or slides) can be utilized in dialog boxes, which are called by an FLISP routine or another application. Detailed information on how to use icon menus and display images in dialog boxes is found in the *Programmer's Guide*.

### Creating a Slide \*

To create a slide file, follow these steps:

1. Display the view of the drawing that you want to save as a slide file. You may need to use the **Zoom** and **Pan** commands to precisely position the view.
2. Type the **SldOut** command.
3. In the **Create File** dialog box, specify the location and the name for the slide file, and click **OK**.

### Viewing a Slide

To view a slide file, follow these steps:

1. Type the **VSlide** command.
2. In the **View Slide** dialog box, choose the slide you want to view.
3. Click **Open**. The slide is then displayed on the screen.
4. Type **Redraw** to return to your drawing.

**Note:** You cannot draw or edit a slide.

## **COPY TO CLIPBOARD**

The **CopySelect** command (**Edit > Copy Selection to Clipboard**) copies selected objects from the drawing and places them on the Windows Clipboard in FelixCAD formats.

**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 this and other FelixCAD drawings.

Once in the Clipboard, you use the **Paste** command to paste the objects from the Clipboard into another FelixCAD drawing. You *cannot* paste the contents into another Windows application.

To copy objects from one FelixCAD drawing to another:

1. Start the **CopySelect** command.
2. You are prompted for an insertion base point. Select a point or type X,Y coordinates. This point is used later during the paste process. We recommend that you use 0,0 as the insertion base point. The same coordinates should also be entered when you paste the information into the destination drawing.

```
> COPYSELECT  
Insertion base point: P1
```

3. Select the objects to be copied to the clipboard. Finish the object selection by pressing **Enter**.

```
Select objects: [pick]  
Select objects: [Enter]
```

4. Switch to the other FelixCAD drawing.
5. Use the **Paste** command to paste the object(s) into the drawing. The prompts are the same as for the **Insert** command.

## PASTE FROM CLIPBOARD

The **Paste** command (**Edit > Paste Selection from Clipboard**) copies objects from the Clipboard and inserts them in the current drawing. This command only pastes objects in FelixCAD format; it does not work when the Clipboard contains other data, such as text or a raster image.

You copy objects from a drawing to the Clipboard with the **CopySelect** command (as described earlier in this chapter), then paste copies into other open drawings. (To copy objects within the current drawing, use the **Copy** command found in the **Modify** menu.)

To paste objects from the clipboard into the drawing:

1. Start the **Paste** command.
2. Specify the insertion point. The objects appear in the drawing, highlighted and attached to the cursor. Drag the highlighted objects to position them in the drawing or type X,Y coordinates. We recommend 0,0.  

```
> PASTE  
Insertion point: P1
```
3. Specify the scale factor for insertion, or press **Enter** to accept the default.  
Scale factor <1>: [**Enter**]
4. Specify the rotation angle for insertion, or press **Enter** to accept the default.  
Rotation angle <0>: [**Enter**]

When inserting the objects from the Clipboard you can also specify the scale factor and rotation angle *before* specifying the insertion point. Choose the corresponding option from the option bar when prompted for the insertion point:

```
> PASTE  
Insertion point: Factor  
Scale factor <1>: .5  
Insertion point: Rotation  
Rotation angle <0>: 90  
Insertion point: [Enter]
```

## PART FILES

Parts (or blocks) are usually created and stored in a drawing. You can, however, export the parts to *.flx* files on disk. This lets you create a parts library that others in your firm can use. Commands related to parts are found on the **Parts** menu.

### Write Part File \*

The **PartExp** command (**Parts > Write Part File**) works identically to the **PartDef** command (discussed in “Parts and Attributes / Groups”) but is used to export parts.

Both commands define entities of a drawing or to define the whole drawing as a part. The part names, the insertion point of the part, and the drawing entities are entered or selected.

### Import Part

To import a part, select one of these options from the **Part** menu:

**Insert Quick**  
**Merge External Part**  
**Insert Part**

You find detailed information about the inserting of internal and external parts in Chapter 8 “Parts and Attributes / Groups.”

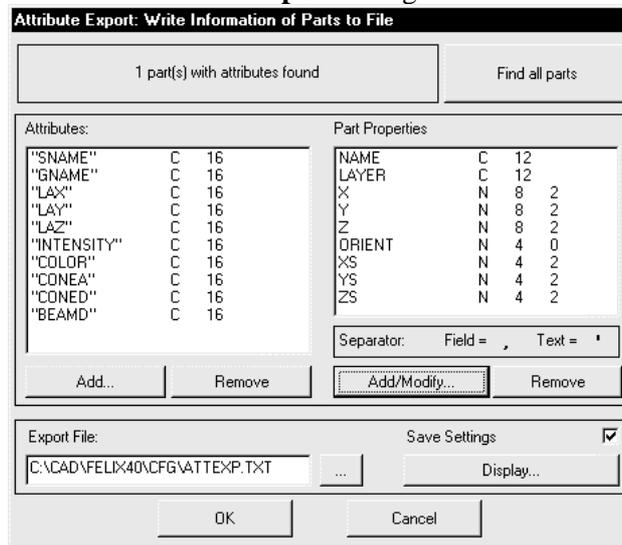
### Export Attributes



The **AttExp** command (**Parts > Export Attributes Information**) allows you to extract attributes for parts and entities, and save these in a text file. The file is formatted with the “comma delimited format” (CDF), which is easily read by spreadsheet and database programs for further processing.

To export attributes, follow these steps:

1. Start the **AttExp** command.
2. Select the objects whose attributes are to be exported. The usual object select functions in the Options Bar are available.
3. Complete the object selection by pressing **Enter** or the right mouse button. Notice the **Attribute Export** dialog box.



4. Those parts that contain attribute values are filtered from the entities selected by you. The number of the parts is displayed in the upper area. You can search the whole drawing for part by clicking the **Find all parts** button.
5. Add the attributes to be exported by clicking the **Add** button. The **Attributes** list shows the attributes which have been selected for export:
  - **N**: numeric value.
  - **C**: character string.
  - **16** (or other number): length of the field input for the attribute. For numeric values, the number of the decimal digits is also displayed.

To remove attributes from the list, select one or more lines in the list, and click **Delete**.

6. Next, determine the field type in the **Part Properties** list. Click the **Add/Modify** button to specify 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.

7. In the **Part Properties / Definitions** dialog box, you can change the field and text delimiter. In a CDF file, commas usually separate the fields, and text is usually surrounded by quote marks.
8. Determine the name of the output file. Click the ... button to display the file dialog box.
9. The **Display** button lets you preview the output. Here you can export the attribute data the Windows Clipboard.
10. The **Save Settings** check box saves these settings for the next time you export attributes.
11. The **OK** button exports the attributes to the CDF file. In some cases, you may get an error message. These errors are usually due to incorrect formatting.

## Printing and Plotting Drawings & Using Layout Mode

This chapter describes how to output a drawing to a printer or plotter.

FelixCAD uses the Windows drivers to print or plot. Before FelixCAD can print or plot, you must ensure the correct printer or plotter drivers are installed. Under Windows, this is usually done automatically when you attach a new piece of hardware to the computer.

The commands described in this chapter are:

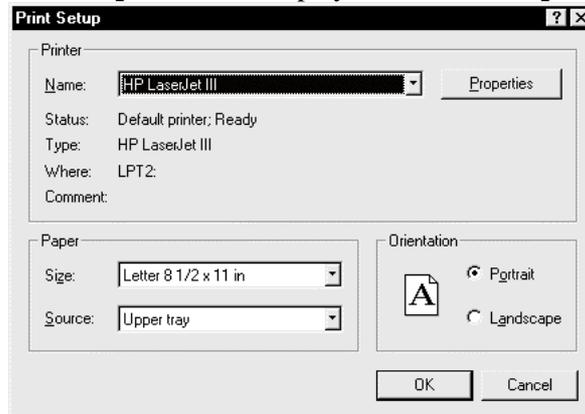
Command	Meaning
Psetup	Printer/plotter setup.
Print	Print/plot the drawing.
TLMode	Toggle between model and paper space.
Viewport	Create and control viewports.

So that circles and other curves are plotted at the highest quality, FelixCAD temporarily sets the circle resolution to **256** when the current value is lower. When higher than 256, FelixCAD uses the value as set in the **CircleRes** system variable.

*Note: For batch file printing and plotting, see the BATPROC command.*

**PRINTER** Setup

The **PSetup** command (**File > Printer Setup**) allows you to select the printer or plotter in the same that output devices are selected in other Windows applications. The **PSetup** command displays the **Print Setup** dialog box.



You find detailed instructions regarding the options specific to the printer/plotter (as well as installing and connecting the printer or plotter) in the Windows *User Manual* or in the output device's manual. Only the steps related to FelixCAD's use of the printer/plotter are in this manual.

**Printer**

The **Printer** area of the **Print Setup** dialog box specifies the printer (or plotter) to be used. Output devices, which are available on your computer system, are displayed in the **Name** list. To see the complete list, click the arrow at the right edge. To change the printer, select a different printer name.

**Properties**

The **Properties** button opens another dialog box, which contain settings specific to the selected printer. You will find more information regarding options and installation possibilities in the device's user manual.

**Paper Size**

The size of the paper, as well as the source of the paper, is controlled from the **Paper** area. Select the proper size and feed.

**Orientation**

Select either **Portrait** or **Landscape** for the desired paper orientation.

## PRINT OR PLOT A DRAWING

The **Print** command (**File > Print**) sends the drawing to the printer or plotter. First, though, select the extent of the drawing to be printed:

View	Extent	Window	NView	
------	--------	--------	-------	--

These options allow you to determine the size and position of the area of the drawing to be printed.

### View

The **View** option prints the currently visible view in the current window of the drawing. Before using this option, you should (1) select the window you want plotted; and (2) use the **Zoom** and **Pan** commands to position the view.

### Extent

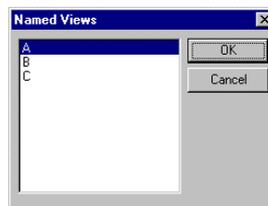
The **Extent** option prints all objects in the drawing are covered. Before using this option, you should regenerate the drawing with the **Regen** command.

### Window

The **Window** option 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 select a previously saved view from the **Named View** dialog box. Before using this option, you must create named views with the **View** command. Named views are useful, since they are saved with the drawing for subsequent print output.

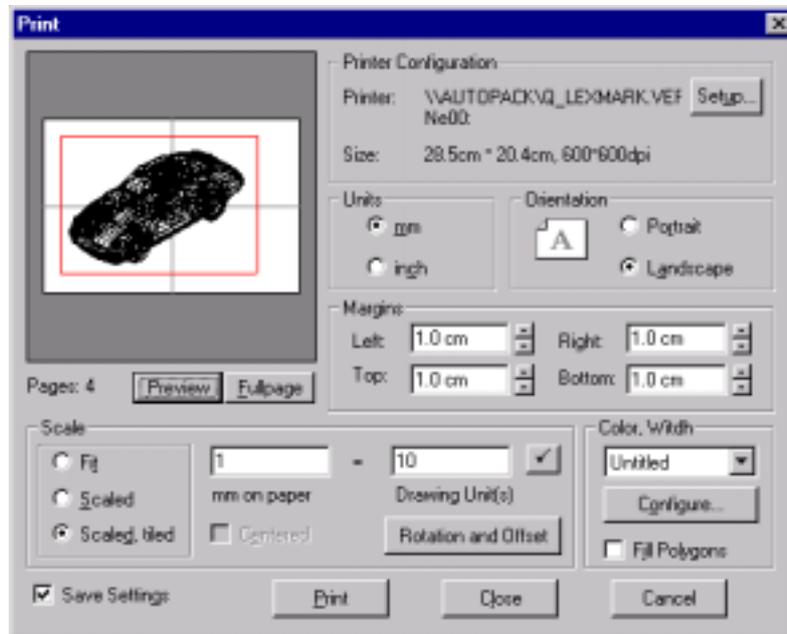


### Print Dialog Box

After defining the size of the print, the **Print** dialog box appears. This dialog box allows you to specify parameters specific to FelixCAD, such as

configuration, units, orientation, margins, scale, and color. It also contains a preview window.

The *preview window* assists you in seeing the effect of changing settings in layout and scaling.



### Printer Configuration

The **Printer Configuration** area lists the name of the printer, the paper size, and print resolution (in dpi). Click the **Setup** button to display the **Print Setup** dialog box (as seen earlier with the **PSetup** command), which lets you control the paper source, paper size, orientation, resolution, and the number of the copies.

The **Print Setup** dialog box contains only the settings that your output device supports.

## Margins

The **Margins** area specifies the edges of the paper that cannot be printed. Enter the values for the **Left**, **Right**, **Top**, and **Bottom** margins.

Note: All plotters/printers have what are termed as 'hard limits' or margins. These limits will not allow printing within certain distances of the paper edges, regardless of where the software margins are set. Refer to your printing device user guide for details.

## Scale

### Fit

The **Fit** option fits the drawing to the margins. This results in a full picture of the object in the print area, but is usually not to any standard scale.

When the **Fit** option is active, it is not possible to set the options for scaling.

### Scaled

The **Scaled** option allows you to define the exact scale of the drawing. Note that scaling is only available when the **Fit** option is off. The setup of the plot scale is done in two steps:

1. Define the size of a drawing unit. Select either **mm** or **inch** in the **Units** area.
2. In the scale fields, enter the scale for **mm on paper** and **Drawing Units**.

For example, you are preparing a surveying drawing where:

- The plan units are mm.
- The drawing units are meters.
- The plot scale is 1:250.

You enter 1 = 0.25 or 1 mm = 0.25 m.

Another example: you are working with an architectural drawing where:

- The units are imperial.
- The plan scale is 1 inch = 20 feet.

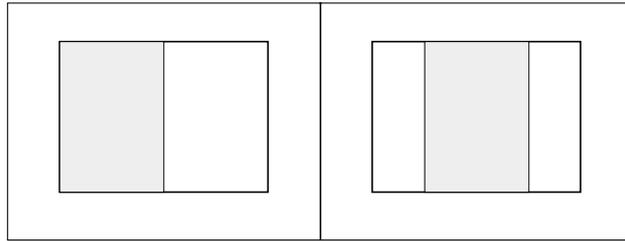
You enter 1 = 20.

The print dialog box shown on a previous page shows this example.

You see the effect of setting the scale in the preview window, after you select the check mark button to the right of the drawing units.

### Centered

The **Centered** option is related to the **Fit** option. Because the **Fit** option does not change the aspect ratio of the drawing to the paper, a full picture may only be output either to the length *or* width of the picture but not 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.

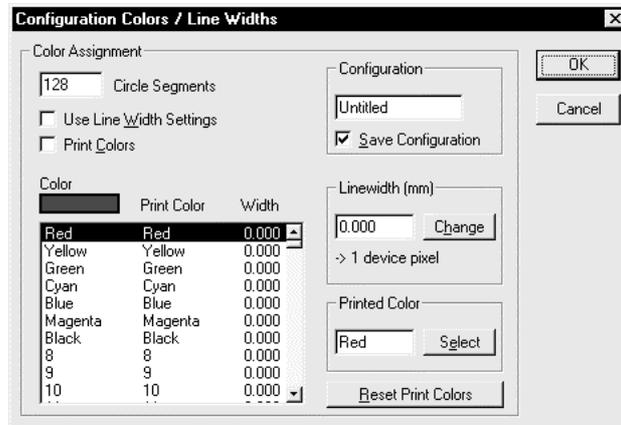
**Note:** The **Centered** option is only available when using the **Fit to Margins** option.

### Poster (Tiled)

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).

## Color, Width

The **Color, Width** area is for defining the colors and line widths output to the printer. By assigning line widths, you improve the legibility of the drawing. To set up colors and line widths, click the **Configure** button, which displays the **Configuration Colors / Line Widths** dialog box.



To set up line widths, you associate the line widths with color numbers. When you created the drawing, every object was assigned a color, or, more accurately, a color number (also called the color *index*). For detailed instructions on the definition of colors for layers or for objects, see “Layer and Object Properties”. For example, red is color #1.

Any color can be given a line width in mm or inches. For example, you can assign a line width of 0.36mm to color red. This means that all objects colored red in the drawing will be plotted 0.36mm wide. The setting of line widths can be saved under a configuration name, and then reused by future printing operations.

### Use Line Width Settings

The **Use Thickness Settings** check box determines whether line widths will be used during plotting.

The **Print Colors** check box switches the color printing on or off. Of course, the output device must have the capability to print in color. Another consideration is how the printer/plotter draws colors. Some devices have palettes that override the program settings. In these cases, see if there is an option for control by 'Software' which will allow FelixCAD to assign the correct colors to each pen.

The **Color / Print Color / Width** list box allows you to coordinate colors with width, as follows:

1. To assign a print color to a drawing color, click the **Select** button in the **Printed Color** area. In the Colors dialog box, select color sample or type its color index number. Click **OK**.
2. To assign a line width to a drawing color, type a linewidth in the text entry box in the **Linewidth** area. Click the **Change** button and the width is updated.

### **Configuration**

You can save the line widths and color configuration by entering a name in the **Configuration** area, then ensuring the **Save Configuration** check box is on.

### **Fill Polygons**

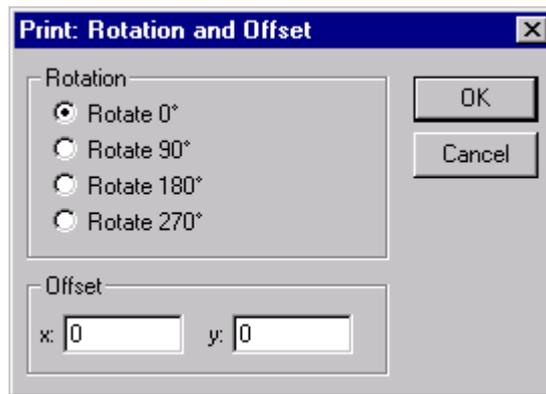
The **Fill Polygons** check box forces FelixCAD to plot filled polygonal areas. If the box is not checked (the default), only the outlines of filled areas are output, which can lead to a faster plot.

## Print Rotation and Offset

Click the **Rotation and Offset** button to display the **Print: Rotation and Offset** dialog box.

The **Rotation** area allows you to select a plot rotation in increments of 90 degrees. The drawing is rotated on the sheet by 0, 90, 180, or 270 degrees. Rotation is applied clockwise.

The **Offset** option moves the plot origin from the lower left corner (0,0) of the paper (the normal position) to another location on the paper. This lets you plot the drawing in a specific location on the page. Specify the **x** and **y** values in paper units (either mm or inch, as determined in the prior dialog box).



The settings are reflected in the preview of the dialog box in the full-page preview

## LAYOUT MODE

One of the primary advantages of working with CAD software is that any drawing plan can be created in real world units. Most drawings, however, eventually need to be printed on paper. Usually this means that the drawing needs to be plotted in a (usually) smaller scale.

For some drawings, you may need to print a drawing that incorporates different scales for different parts of the drawing. Examples include:

- Drawing title blocks.
- Display different views and regions.
- Create details of the model.
- Lay out part lists, legends, 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*. These modes are sometimes known as model space and paper space, or print space.

The objects are drawn in model space. In paper space, you reproduce the model in different views and scales, and annotate the drawing with text. How you use layout mode is determined by user and company CAD standards. Generally there are two methods applied to their use:

- 1) Keep the drawing borders at 1:1 scale and scale down the views to fit or
- 2) Make all views in layout mode at 1:1 scale and scale up the drawing boarder, text and dimensions to suit.

It is recommended you choose one method only.

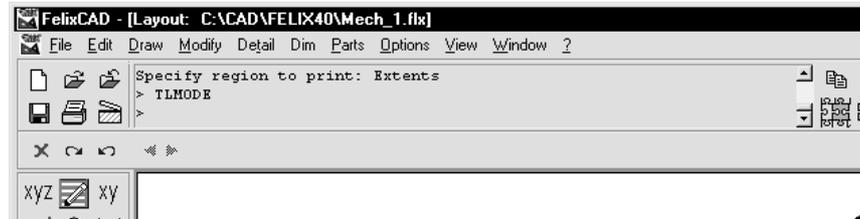
**Note:** Layout mode is not always the easiest way of finishing a drawing. If you create a pure 2D drawing, often it is easier to insert (on separate layers) title blocks, part lists, and details in model space, which are scaled to a larger factor to fit to a certain paper size. Some drawings are created easier at true scale.

When using layout mode, remember that this feature is provided to arrange and lay out 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 the **TIMode** command (**File > Layout Mode / Model Space**).

When you switch from model to paper mode the first time, the layout window is empty, which represents a blank drawing sheet. A single window replaces the tiled viewports of the current drawing (displaying the model in different views). The title bar of this window shows the caption [**Layout:** and the drawing name.



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 performs two operations:

1. Determines the sheet size for printing, particularly by insertion of a title block.
2. Allows the creation of viewports that display individual views of the model.

You can draw in paper space just as you do in model space. All the drawing and editing commands are available. Since paper space is designated to lay out drawing output, a number of 3D commands are disabled, including: **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.

## Viewport \*

The **Viewport** command creates rectangular areas, called *viewports*, which display the model in print space. The command controls the view direction and display portion of the model in those viewports. The options of this command are:

New	ON	OFF	Layer	View		
-----	----	-----	-------	------	--	--

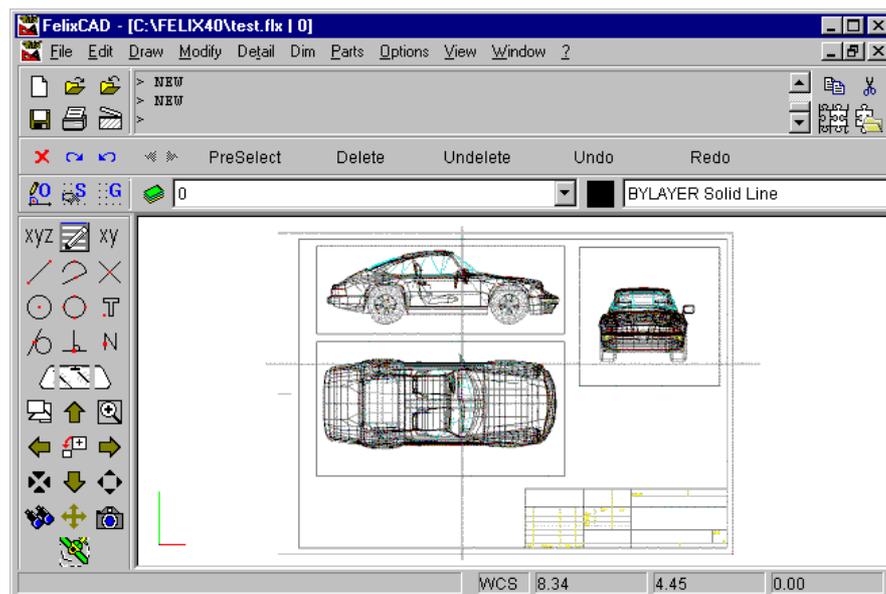
### New

After switching to layout mode for the first time, the first step is to create one or more viewports to display one or more views of your model.

The **New** option creates these viewports. After specifying two corners of the viewport, the entire model is displayed in its current extents and in plan view.

```
> VIEWPORT
First corner: 10,10
Opposite corner: @45,25
```

The second step is to determine the correct zoom ratio to allow the contents of this viewport to be displayed at the correct scale when printed on paper. This scale is directly related to the size of the title block and the plot scale used to plot this drawing. This topic is discussed in greater detail later in this chapter.



## Modifying the Viewports

A *viewport* is more than just another window. It is a drawing entity in its own right. It is represented by a rectangle and acts as a container of a certain view of the construction. Theoretically, any number of viewports may be created in model space.

You can use editing commands on viewport entities because they are treated as drawing objects. You can move a viewport to another position with the **Move** command. You can change the size of a viewport with the **Stretch** command (the view in the viewport is not scaled with the viewport). When you change the size of a viewport with the **Scale** command, the view inside the viewport is scaled by the same factor. You can erase viewports from the drawing with the **Delete** command. Viewports may be copied with the **Copy** command. The **Rotate**, **Array**, **Flip**, and **Mirror** commands also modify the viewport with the limitation that the edges of the viewport's rectangle remain parallel to the X- and Y-axis of the paper space. Also, these commands do not rotate or mirror the view in the viewport.

You can use object snap modes and coordinate filters for editing viewports and when drawing in paper space. You can only manipulate viewports while in paper space.

## Turn Viewports Off and On

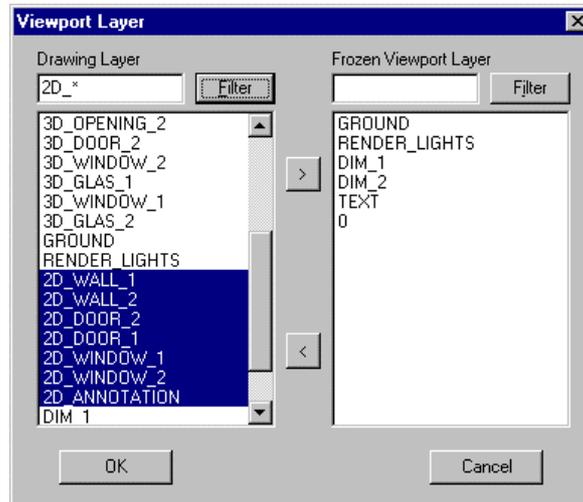
The **Viewport** command's **On** and **Off** options control the visibility of the contents displayed in the viewport. The **Off** option turns off the visibility of the viewport objects: select the viewport frame or pick to an object inside the frame. To redisplay the model within a deactivated viewport, choose the **On** option and pick to an edge of the viewport's rectangle.

The visibility of the viewport frames themselves is controlled through layers. Like any other entity, viewports are located on a layer. We recommended that you place viewports on one or two specific layers, such as "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's **Layer** option freezes and thaws the layers of the contents of viewports. For example, you could use this option to display 3D views in certain viewports. Only those objects are visible that are on layers with the prefix "3D\_\*". Other viewports only display layers containing detailed 2D objects, which have been placed on layers with the prefix "2D\_\*", "DETAIL\_\*", and "DIM\_\*".

After selecting the **Layer** option, the **Viewport Layer** dialog box is displayed.



The **Drawing Layer** list box contains the names of all layers defined in the current drawing. The **Frozen Viewport Layer** list box contains those layers not to be displayed in the currently selected viewport.

You change the layer visibility by selecting one or more layer names in one list, then click the **>** or **<** button.

To freeze or thaw an entire group of layers, you can specify strings with wild cards, such as **2D\_\***, and then click the **Filter** button. This highlights all layers that start with **2D\_\***, as shown in the figure above.

## View Control in Viewports

The **Viewport** command's **View** option enlarges a viewport to fit the entire window. It also allows you to change the zoom level, 3D viewpoint, and so on.

```
> VIEWPORT
First corner: View
Select viewport: [pick]
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	Ratio	NView	3DView	Viewpoint
------	-----	-------	-------	--------	-----------

### Zoom

The **Zoom** option displays the prompt, “Zoom Scale factor: ”. Once a scaled relationship between model 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.

During the **Zoom** option, the option bar contains other zoom options: **Window**, **Extents**, **0.9**, **0.1**, and **Previous**.

### Pan

The **Pan** option moves the visible portion within a viewport. Two points specify the pan direction. The scale of the model is not changed.

You can also use the pan functions from the control panel: Pan Left, Pan Right, Pan Up, Pan Down.

```
Viewport control: 'PANLEFT
```

## Ratio

The **Ratio** option is related to the **Zoom** command. It allows you to specify the scale ration between paper and model space, much like during the **Print** command:

```
Viewport control: Ratio  
Info: Height of Viewport: 4.28; Height of model  
representation: 484.95  
Paper Space Units <1>: 1  
Model Space Units <113.33>: 2
```

## NView

The **NView** option display a view that had been saved earlier as a named view with the **View** command in model space. The **View** option opens a dialog box similar to the one displayed by the **View** command in model space (see Chapter 3).

The **Show** button is useful for save and retrieving named views. This option previews the selected view in the drawing before confirmation.

You can improve your effectiveness of working in layout mode by establishing named views in model mode, then use them in the viewports of the paper space. This is especially useful when a scaled relationship must be maintained between model and print space. (See the *Techniques and Tips for Layout Mode* section later in this chapter for more hints and recommendations.)

### Note:

When you use the **View** command in layout mode (not the **View** option of the **Viewport** command), you can create named views within paper space.

## 3DView

Naturally, you can display the model from different viewpoints in space. The **3DView** option allows you to set individual 3D views for each viewport. You can also select a predefined view, such as plan view, front view, or an isometric view.

The **3DView** option displays the same dialog box as invoked by the **3DView** command in model space (see Chapter 3).

**Note:**

Using the **3DView** option zooms the content of the viewport to the model space drawing extents. To achieve a scaled relation between paper space and model space for isometric or other individual 3D views of the model, we recommended that you use named views prepared in model space.

### Transparent Commands

As mentioned earlier, you can use the transparent commands **'PanLeft**, **'PanRight**, **'PanUp**, and **'PanDown** from the control panel. In fact, you can use all transparent commands in layout mode.

A useful example is the transparent command **'SetViewDir** to set a standard isometric 3D view within a viewport.

The transparent command **'Dist** allows you to measure the distance between two points.

### Tips and Techniques for Layout Mode \*

Create your construction (in model space) in real world units (at scale 1:1).

Use separate layers for dimensions, hatching, text, etc.

When managing layers, keep in mind that you might want to have different logical views of your model for use later when presenting or plotting your drawing with the aid of layout mode.

At a certain stage of the drawing process, you will find the need to print out the drawing to present it to your client or to forward it to the manufacturer. In this situation, layout mode can help in many cases:

1. Switch from model space to paper space with the **TIMode** command. Notice that the title bar now reads **Layout:** followed by the drawing name.
2. Set the limits to equal to the paper size used for printing, such A-size or ISO A0.

3. Create a layer for the title block (unless you won't be needing a title block). Make the layer current and insert the title block in paper space at the origin (0,0). As an alternative, you can draw a rectangle with the paper size your printer uses when printing.
4. Create a layer for the viewports. Make this layer current before you create new viewports with the **Viewport** command. Make sure to use this layer for viewports only. We recommended you activate the snap mode with applicable units (inch, millimeter, and centimeter) when creating viewports.
5. Once the viewport has been established, set up the scale between model and its representation on the drawing sheet. This step sometimes causes headaches. Set up the scale for displaying the model by:
  - a. Position a rectangle on a temporary layer in model space that matches the viewport window in paper space at a specific scale.
  - b. Create named views in model space (with the **View** command), which can be retrieved within the paper space viewports in with the **Viewport** command's **View** option.
6. With the scale set up, be sure to avoid uncontrolled zooms in the viewport. Confine yourself to the **Pan** commands and to the **ZoomFac** command (specifying whole numbers).
7. To modify the height or length of a viewport, we recommend that you use the **Stretch** command. The model displayed in the viewport is not 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.

## Visualization: Hidden Lines and Rendering

Three-dimensional models are created in FelixCAD as a wireframe drawing. With the **3DView** command, you can view the 3D geometry in predefined isometric views, as well as from any 3D view direction. When opening additional windows of the current drawing, you can specify independent 3D view directions for the new viewport.

With most view commands, FelixCAD displays all lines that make up the wireframe model. To display a more realistic image of your 3D model, you can have FelixCAD remove the lines that are hidden from your view (called “hidden line removal”), and to creating full-color renderings the 3D model.

This chapter describes the visualization commands, which are found in the **View** menu:

Visualization Command	Meaning
Hide	Create a new hidden-line drawing in a separate window.
Fhide	Perform a fast hidden-line removal.
Render	Render the 3D drawing.
Rconfig	Specify parameters for renderings.
Light	Insert a light to illuminate the rendering.
LightEdit	Edit the light parameters.
Arender	Create an animated rendering.

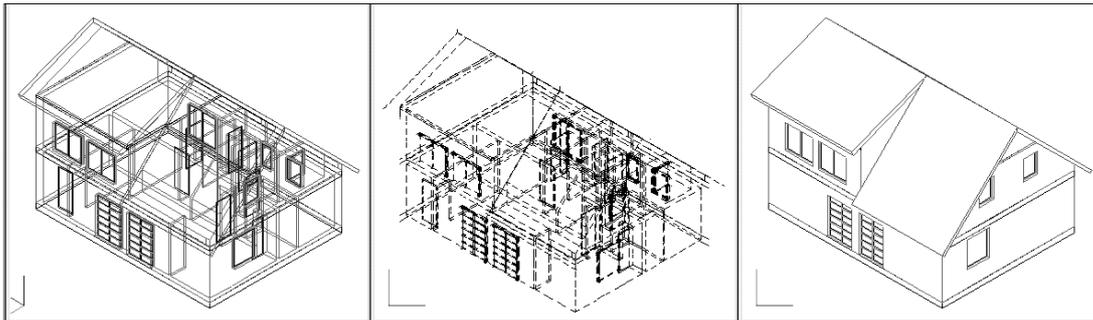
The **FHide** and **Render** commands are based on the OpenGL graphics system from Microsoft, which provides high-performance hidden-line removal and rendering.

**HIDDEN LINE DRAWING \***

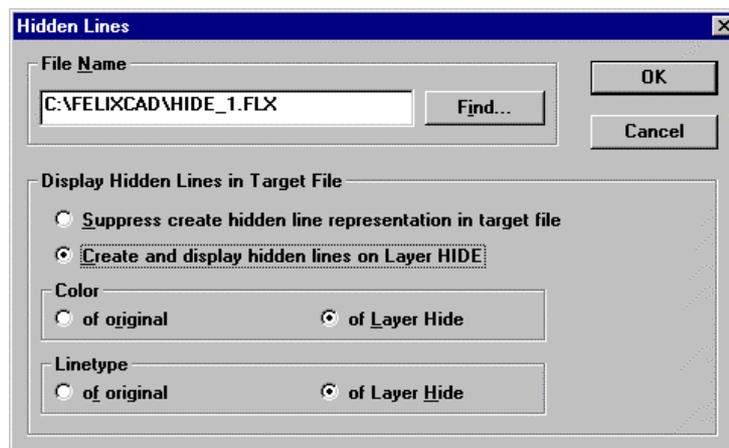
The **Hide** command (**View > Hidden Lines > Hidden Line Drawing**) creates a separate *.flx* drawing file containing the current view with the hidden lines removed.

As an option, lines and edges that are computed as hidden are placed on a system-generated layer named “HideLayer” (see figure below). You can turn on or off this layer, assign them a dashed linetype or another color. In addition, you can remove lines from that special layer and retain those you want to keep for demonstration purposes.

Note that the “3D” hidden-line view is actually a flattened 2D drawing.



3D drawing, showing all lines.

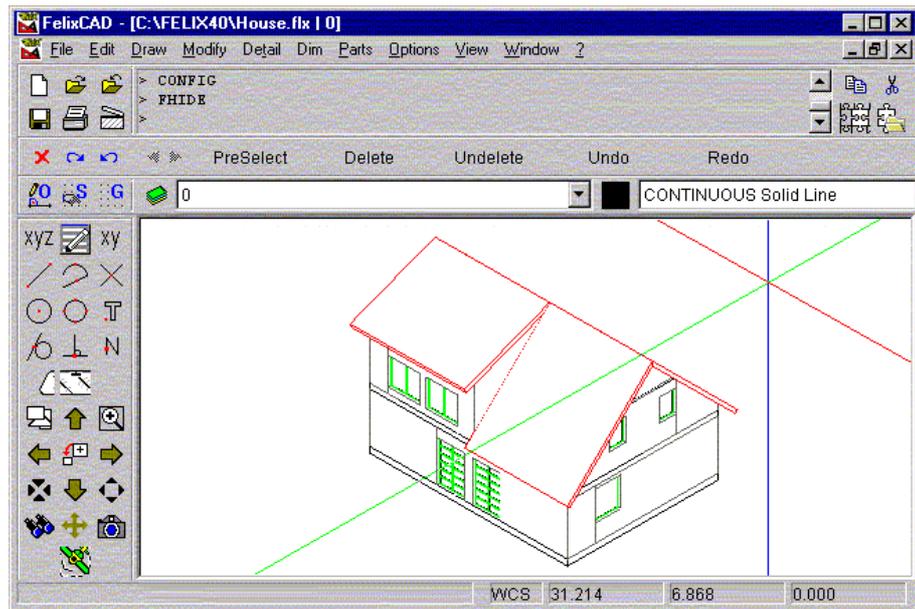
Hidden lines on layer **HideLayer**.Hidden lines removed by **Hide**.

## FAST HIDE

The **FHide** command (**View > Hidden Lines > Fast Hide**) creates a hidden line view in the current drawing. (This is in contrast to the **Hide** command, which outputs the hidden lines to a new drawing.) This **FHide** command provides you a visual impression of the 3D model from the current viewpoint.

You can continue to draw in hidden-line display mode, even using object snap functions. The hidden-line representation is maintained until a **Redraw**, **Regen**, **Zoom**, or **Pan** command refreshes the drawing window.

The **FHide** command is based on the OpenGL system from Microsoft, which allows high performance hidden-line removal.



**RENDER \***

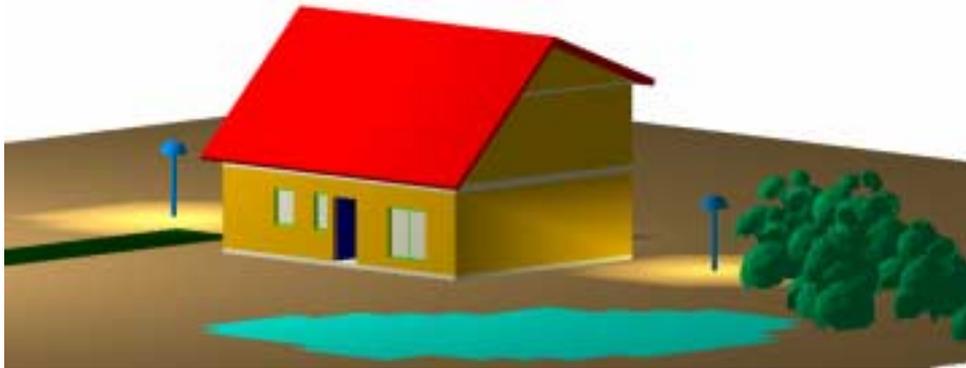
The **Render** command (**View > Render > Render**) creates an image of your 3D drawing, called a rendering. The *rendering* is a realistic illustration of the model. To increase the realism, the rendering can be made to perform smooth shading and evaluate the lights set into the drawing.

You use rendering to visualize and present your 3D geometry, as it would appear as if it has been constructed or built. Note that FelixCAD also includes an **animated rendering** function as detailed below this render description..

To render the current viewport of a drawing, start the **Render** command. The command prompts you to select where the rendered view should be output to.

Screen	File	Clipboard			
--------	------	-----------	--	--	--

- **Screen:** the rendering is output to the current viewport.
- **File:** the rendering is saved to a *.bmp* (Windows bitmap) file on disk.

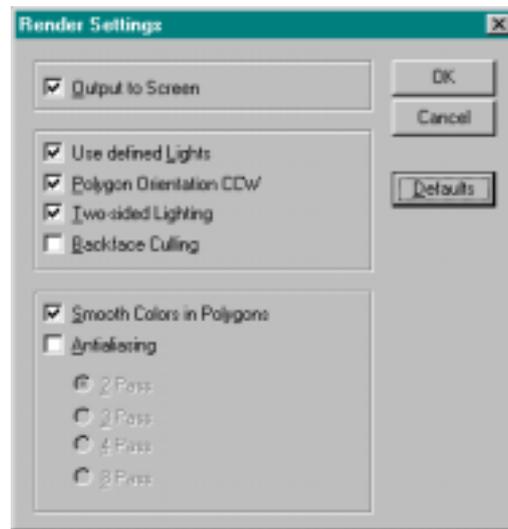


- **Clipboard:** the rendering is copied to the Windows Clipboard in bitmap format; it can be pasted into another Windows application with the **Ctrl+V** (**Edit > Paste**) command.

The rendering is based on the preferences set with the **RConfig** command. The **Light** command allows you to place lights in the drawing and define their type and settings. The **LightEdit** command is used to modify the settings of light sources.

## RENDER PARAMETERS

The **RConfig** command (**View > Render > Render Parameters**) displays the **Render Settings** dialog box, which lets you control the quality of rendering.



The **Defaults** button resets the preferences to default values (as shown in the illustration above). This is convenient, because a number of settings can influence the performance of render computations dramatically.

The **Output to Screen** option 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 option is not checked, the rendered image is not displayed before the entire rendered image is computed.

The **Use defined Lights** option allows you to activate or deactivate the use of light sources. When disabled, the program generates a simple shading of the 3D model; otherwise, the inserted lights are used for the rendering.

The three other options allow expert users who are familiar with 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

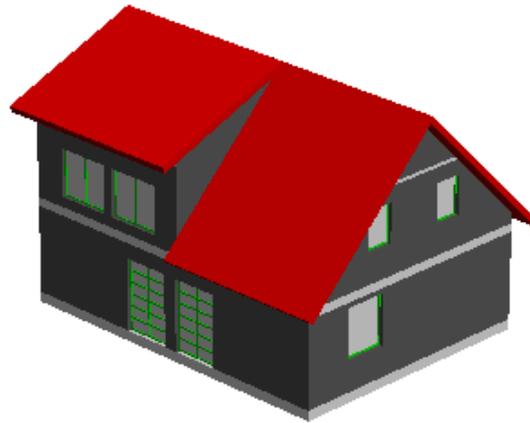
It is recommended to use the default values if you are not familiar with these terms.

The **Smooth Colors in Polygons** option **specifies** whether smoothing colors are computed upon individual polygon surfaces. Applying an adequate light environment allows smooth coloring on individual surfaces.

The **Anti-Aliasing** option allows you to control the process of smoothing jagged edges caused by aliasing. *Aliasing* describes certain display effects, like the staircase effect seen on diagonal lines displayed in a drawing. Anti-aliasing reduces those artifacts. Turn on anti-aliasing 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 dramatically increase the rendering time, anti-aliasing should be applied only for high-quality image output (usually when rendering to a file).

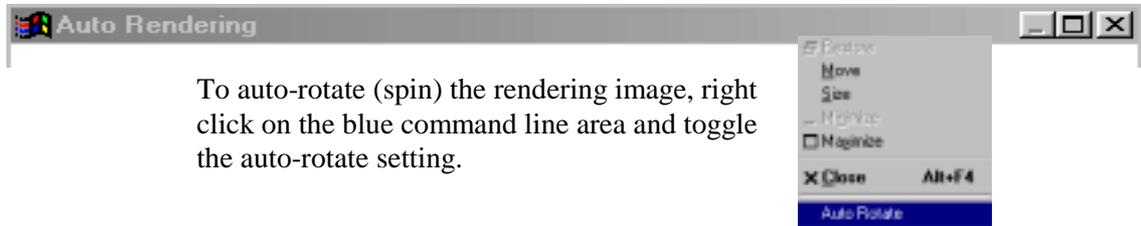
### ***ANIMATED RENDERING***



The **Arender** command (**View > Animated Rendering**) displays a rendering of the current drawing in a separate pop-up window.

You can rotate the rendering by situating your pointer inside the Arender window, and holding down the left mouse button. While the mouse button is depressed, the icon will change to a small hand, allowing you to rotate the image about the screen.

Zooming and panning for Arender is accomplished by holding down the right mouse button. While depressed, you can slide the pointer up or down to zoom and pan the image.



To auto-rotate (spin) the rendering image, right click on the blue command line area and toggle the auto-rotate setting.

## LIGHTS \*

The **Light** command (**View > Render > Insert Light**) allows you to insert lights in the drawing. The lighting environment in a 3D model is set up with one or more light sources. When no lights are explicitly present, FelixCAD assume ambient light for the light source.

### Light Types

The render program recognizes four types of light sources:

- Ambient light.
- Linear light.
- Point light.
- Spot light.

An *ambient light* is not really a light source. It is an overall light source that occurs everywhere equally in the scene. 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, 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 an incandescent lamp. It has a position, intensity, and a color.

A *spotlight* 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 available in all renderings, but the other sources must be placed explicitly in the 3D model. The **Light** command insert light sources into the drawing.

Although you can insert as many light sources as you want, you can have only eight light sources turned on.

The **Light** command displays the **Light Sources** dialog box, which allows you to define parameters for the light, among them the color and the intensity of the light source.

Lights sources are represented in the drawing as block objects. A light insertion is always placed on a layer with the name "RENDER\_LIGHTS."

## Editing Light Parameters

You modify the settings of an inserted light source later with the **LightEdit** command (**View > Render > Edit Light Parameters**). This command requests you to select a light, represented by a block insertion. Then the **Light Sources** dialog box opens, in which you can alter the current preferences for the chosen light.

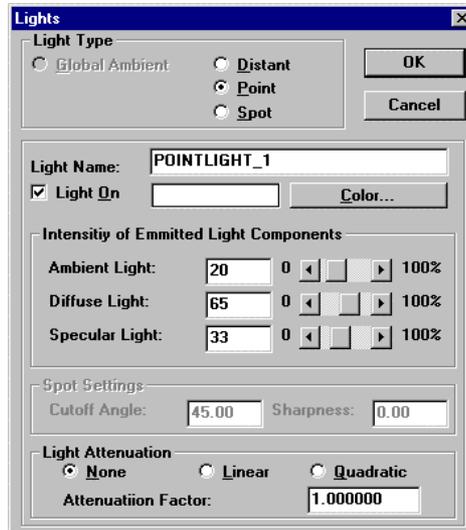
## Relocating and Removing Light Sources

To change the location of a light or the lights direction, use the standard modify commands **Move** and **Rotate**.

To remove a light source from the drawing, use the **Delete** command.

## Light Sources in Detail

The **Light Preferences** dialog box is used for all four types of light sources. Depending on the light type, some areas of the dialog box may be grayed out because those parameters do not apply to the specified light type. The following figure shows the properties and intensity values of a point light.



## Ambient Light

Ambient light is not really a light but defines an overall value of 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 the problem of having the entire model rendered too darkly.

Ambient light has no direction, which means that all surface polygons in the drawing are illuminated uniformly by this type of light (independent of 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 the other light types. For that reason it is not covered by the **LIGHT** command. To modify the value for the intensity of ambient light use the **LightEdit** command with the **Ambient** option (**View > Render > Ambient Light**) 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. Increasing the intensity of ambient light tends to saturate the image and give it a washed-out appearance.

The default color of ambient light is white.

Ambient light may also be turned off to simulate absolute darkness. This corresponds to an intensity of 0%. Only other light sources in the drawing will then make drawing objects visible. Turning ambient light off can be utilized to generate special effects when rendering the model.



Ambient light turned off.

### 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, intensity, and a color. The intensity of the light, however, 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, intensity, and a color. The intensity of a point light diminishes with distance.

You set the manner by which the light attenuates with distance (none, linear, or square) and the factor for distance falloff in the **Light Attenuation** section of the dialog box in.

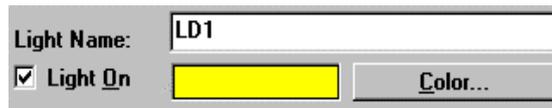
## Spot Light

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, intensity, and a color. In addition, the spotlight has a direction and a cone angle.

You determine the light attenuation (falloff with distance) for spotlights. 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.

## Common Light Settings

Light source insertions have some settings in common, and some setting specific to specific light types. This section describes the in-common settings.



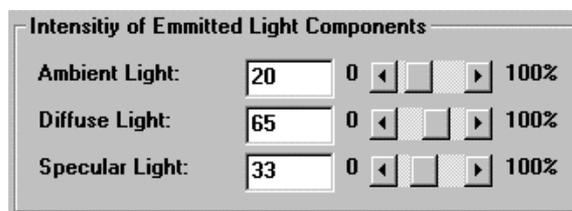
In the **Light Name** text edit box, you specify a name of a new light source or alter the name of an existing named light insertion. Give each light a unique name. You refer to the light name with the **LightEdit** command.

The setting of **Light On** determines if the light is evaluated during rendering. Turning off a light means that the render program does not use the light to illuminate the model. The light settings are retained when turning the light status back on.

Clicking the **Color** button display the standard Windows **Color** dialog box, allowing you to define a color for the current light source. The default color 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 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. Once it hits a surface, however, 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 specularly as shininess.

## Light Attenuation

The **Light Attenuation** section lets you specify distance falloff for point lights and spotlights.



First, you determine the manner the light attenuates with distance:

**None:** no attenuation is applied for the point light or spot light.

**Linear:** linear falloff is applied, which is commonly used for rendering. The brightness of the light decreases in inverse proportion to the distance from the source.

**Quadratic:** square falloff 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.

Second, if you have chosen linear or square falloff, you can specify an **Attenuation Factor** in the edit box. The default value is 1. Light attenuation at a certain location in the model will be:

- Linear falloff: the inverse of the product of distance and attenuation factor;
- Square falloff: the inverse of the product of square of the distance and attenuation factor.

### Specific Spotlight Settings

The **Spot Settings** parameters are set only 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:

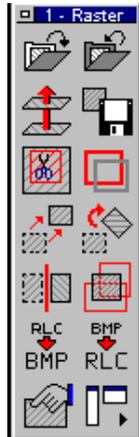
- **Minimum value, 0:** a sharp accentuated light cone.
- **Any other value:** the brightness is highest at the target line of the spotlight and decreases to the edge of the cone (by the exponent specified).

**Maximum value, 128:** the smoothest integration of a spotlight into the environment.



# Raster Images: Using and Editing

## RASTER PALETTE



The **View > Raster** menu item opens the **Raster** palette (toolbox) with most of the commands available on it.

All raster editing and manipulation commands are available for RLC (large scanner) format. Only some of the editing commands are available for TIF and BMP files as noted in the following pages.

There are conversion programs on the market that may assist you in changing from another format to one of these supported formats. FelixCAD 4.01 includes a converter for RLC $\leftrightarrow$ BMP (monochrome only).

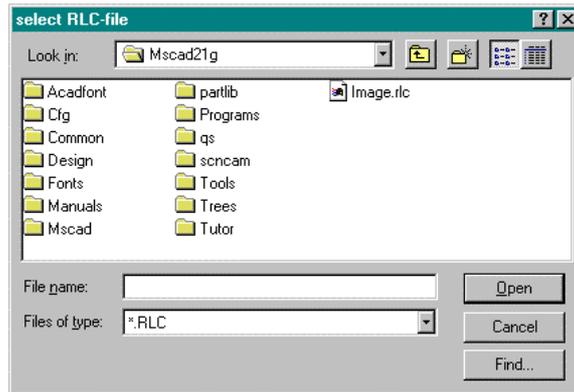
*Note: it is necessary to ensure your windows system is setup to view ALL file types otherwise you will not be able to view/locate it*

**Note: Because inserted raster images do not become integrated in the drawing file, it is important to not delete or move the original raster image file so the link between the two remains intact.**

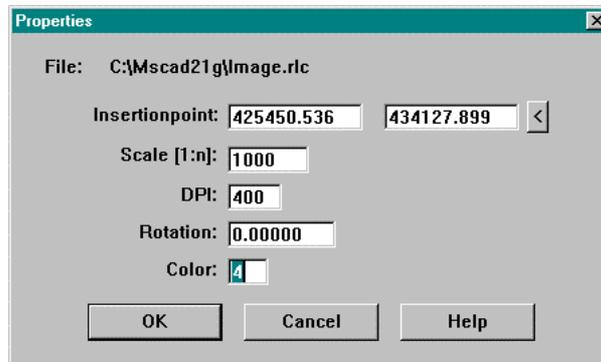
**LOAD RASTER FILE (RLOAD)**

This palette button inserts a scanned or raster image from a file into the current drawing. New for FelixCAD 4.01 and higher is the addition of *color.tif* and *.bmp* file formats. These two formats have been added to the existing *.rlc* format from previous releases.

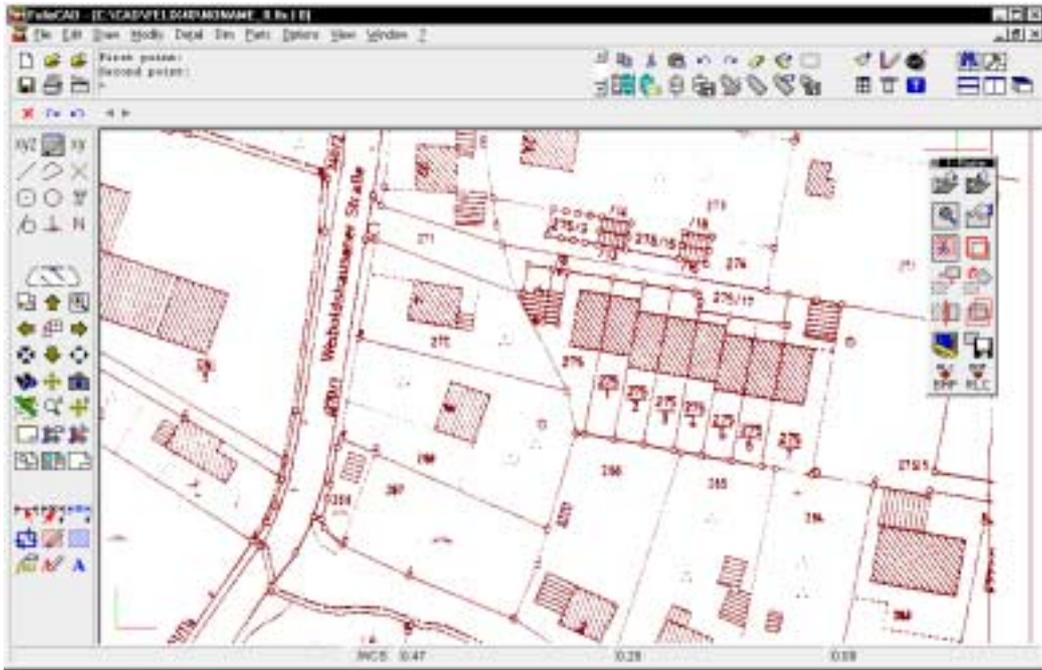
The Load Raster File button displays the **Select File** dialog box to allow you to select the raster image you wish to insert into the drawing. Pick the raster file type (RLC, TIF, or BMP) from the **Files of Type** pick list. FelixCAD is currently limited to two raster images at one time.



Once you select an image, the following dialog box asks you where to insert the image, the scale factor, the color of the image, the DPI (dots per inch), and the rotation factor.



Once you have entered the information, the raster image appears on the screen. You may need to zoom to the extents of the image using the zoom tools on the control panel to see the complete image.



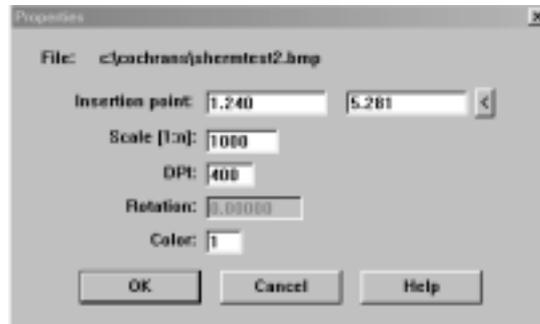
### **REMOVE RASTER (RUNLOAD)**



This **Remove Raster** button removes the selected image from the drawing. If 2 images are currently loaded, the system will prompt for which image to delete.

**PROPERTY SETTINGS OF RASTER FILE (RPROP)**

The **Property Settings of Raster File** button brings the following dialog box and allows you to change any of the fields. The changes are only done to the current image and do not affect the original stored on your computer.



**Insertion point:** location of the image's lower-left corner.

**Scale [1:n]:** Sets the scale factor to make the raster image larger or smaller.

**DPI:** dots per inch, affects the size of the image.

**Rotation:** orients the image in reference to the drawing. Positive rotation is counterclockwise, negative rotation is clockwise.

**Color:** Sets the color the entire image takes on:

Number	Color
1	Dark red
2	Green
3	Gold
4	Dark blue
5	Purple
6	Turquoise
7	Light gray
8	Light green
9	Light blue
10	Light yellow

**SAVE RASTER (RSAVE)**

The **Save Raster** button allows you to save all of the changes to the image, done by the **Property Settings Raster File** button, back to your hard drive.

These changes are permanent, so be careful and make sure you have a back up of the original raster image before saving the changes.

**TOGGLE RASTER FRAME ON/OFF (RBORDER)**

The **Toggle Raster Frame On/Off** button toggles the rectangular frame border around the image. The frame is turned off or on. The frames of all raster images 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.

**MOVE RASTER (RMOVE)**

The **Move Raster** button moves an image in the drawing, as the name suggests. If you have more than one raster image in the drawing, you are asked to select the one to be moved:

```
select raster /<point>/?/Last/#n [#0]:
```

The options have the following meaning:

- **<point>**: you select the raster image with the cursor.
- **Last**: select the last image selected.
- **#0**: Type the id number of the image; the first image is #0.
- **?**: displays all numbers to choose from.

You are then asked for a **Base point** and a **Destination** point, just like the standard **Move** command.

**ROTATE RASTER (RROTATE)**

The **Rotate Raster** button lets you rotate RLC and TIF raster images (does not support BMP images) to match your vector drawing. The prompt is as follows:

```
select raster /<point>/?/Last/#n [#0]:
```

This allows you to select the correct raster image to rotate, using the same options as for the **Move Raster** command. Next, you are prompted:

```
Rotation angle:
```

This is the amount of rotation: clockwise is a negative rotation and counter-clockwise is a positive rotation.

**MIRROR RASTER**

The **Mirror Raster** button (RLC only) allows you to mirror image RLC files only. You first select the image on screen:

```
select raster /<point>/?/Last/#n [#0]:
```

After selecting the image, you are asked;

```
mirror axis:
```

Two options appear on the options bar: **Horizontal** and **Vertical**. Pick the one you wish to use. After a few seconds, the raster image is mirrored and FelixCAD displays the **Properties** dialog box. This lets you specify the properties of the mirrored copy.

The original image is not removed. The mirrored image is displayed according to the image properties.

**RMONT**

The **RMont** command (RLC only) lets you copy a portion of, or the whole, raster image. You are prompted to place a window around the portion to be copied. This is similar to the **Zoom Window** command. You are prompted for the **First Corner** and then the **Second Corner**.

After picking the area to copy, you are asked to confirm the properties of the new image. The changes are only for this drawing session, unless you save the property changes.

**RCUT**

The **RCut** command (RLC only) lets you remove a portion of the raster image. 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. 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 [#0]:
```

Then you are asked to pick the polyline to use for the clipping. Finally, the command asks:

```
save properties for 'C:\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;

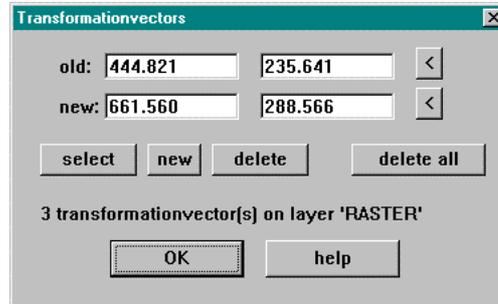
```
properties saved for 'C:\Image.rlc'.
```

and then the image is cut or clipped inside of the polyline and the properties are saved for the original raster image.

## RVECTORS



The **RVectors** command (RLC only) edits 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." You can actually draw the vectors before starting this command and it will detect them for you.



### Old

**Old** is the current starting location of the currently selected transformation vector. You can type a new location if you wish or use the < button to visually pick a location on the screen.

### New

**New** is 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 visually pick a location on the screen.

### Select

The **Select** button will let you select which vector you are currently editing. You pick the vector on screen.

### New

The **New** button create a new vector from the coordinates displayed in the **Old** and **New** fields. It duplicates 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 < buttons to the right of the **New** and **Old** coordinates.

**Delete**

The **Delete** button deletes the currently selected vector, both from the routine and the screen.

**Delete All**

The **Delete All** button deletes all 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.*

**4-POINT TRANSFORMATION (RTRANS)**

This command performs a simple transformation that stretches the raster image to match your vector line work. You are required to enter four 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 four transformation vectors, since 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**

For printers and plotters that can output raster images (such as inkjet, laser, electrostatic, and dot matrix), you can control whether you want the vectors printed on top of the raster image -- or the other way round. This way you can see the vectors on top of the image, if desired. The prompt of the PrintOpt command 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. The raster will now print/plot as you have indicated.



## The BATPROC Command (Batch Processing) \*

The **BATPROC** command is a tool for Batch Processing of FelixCAD FLX, DXF and DWG files. It has three main functions:

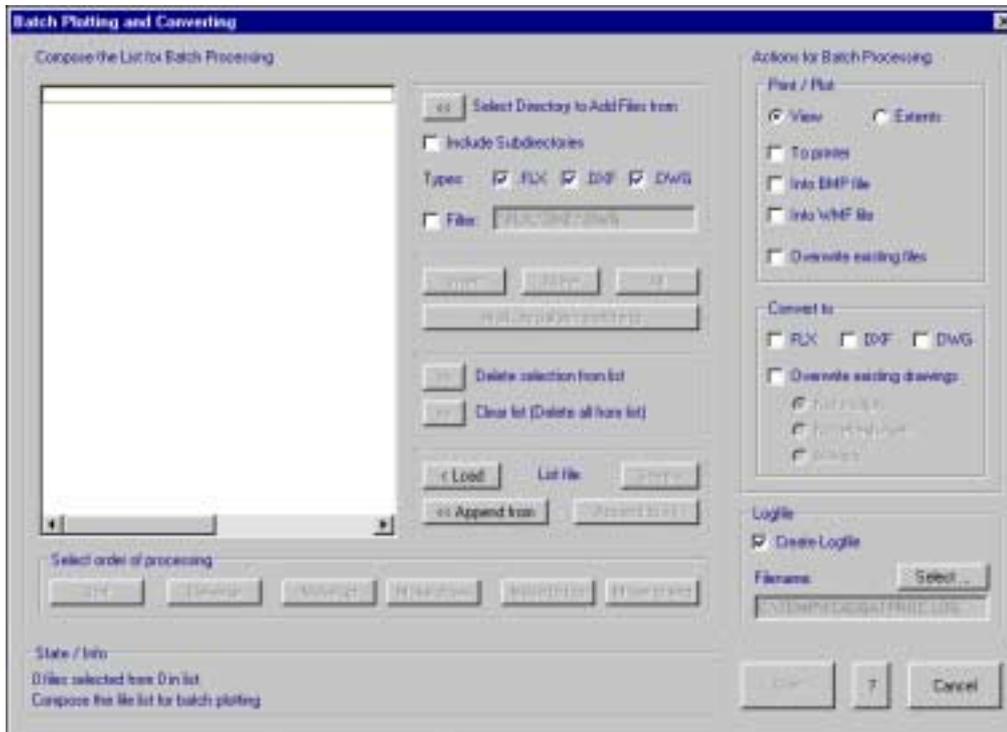
- Converting files (FLX, DXF and DWG) to Raster Format (WMF or BMP)
- Converting files (FLX, DXF and DWG) to Vector Format (FLX, DXF and DWG)
- Batch plotting of files (FLX, DXF and DWG)

These functions can be used alone or in combination (example. batch plotting and file conversion).

All Batch Processing functions are started by adding files for processing to a **'List'** located on the left side of the Batch Plotting and Converting Dialog box. The **'Actions'** to apply to the 'List' are located at the right side of the Dialog box.

## START BATCH PROCESSING

You can start the command by simply typing "BATPROC" or by selecting the menu item 'Batch Processing' from of FILE menu. Once this is done the following dialog box appears at the screen:



## CREATING AND MODIFYING FILE LIST

The first step in Batch Processing is creating a 'List' of drawing files to process. Located on the left side of the BATPROC dialog box, there are several file selection tools that enable you to:

- Add files from any directory
- Save a List of files
- Append a file list to the current list or load a saved file list.
- Remove files from the current list.

## Options for Adding Files from a Directory

The first step is setting the selection options. The two options for file selection are displayed as check boxes:

**Include Subdirectories** (click on box to include all subdirectories of the chosen directory)

**Filter:**  (edit filter list & click on box to select only these file types)

## Adding Files from a Directory

The next step is to select a directory and add all files inside that directory to the list. The button to do this is labeled:  Once you click on this button, browse through your computer directories (networked or local) and choose the directory containing the files you want to process. To add files from a different directory, simply repeat the command or use the  button (see List Files below).

*Note: If you have previously selected a directory and saved that list of files, you can load a list file instead (see List Files below).*

## List Box

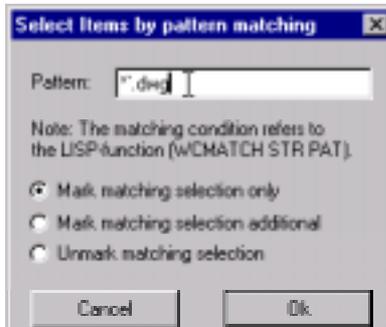
Once the previous two steps have been completed, you will see a list of files from which to select individual ones for processing. Select some or all of the files by marking them inside of the box. To do this, use the mouse (left click) in combination with the <SHIFT> or <CONTROL> Key to get a multiple selection. The selected file(s) will be highlighted. This in fact operates in the same fashion as all Windows programs for file selection. *Note: that if you start the processing, all files inside of the list, not only the highlighted files, will be used.*

### Selection Options

After selecting or deselecting items of the list box, you can change the selections using any of six buttons:

- . Clicking this button will select all items from the list.
- . Clicking this button will deselect all selected items of the list. In result no item in the list will be selected.

- **Invert**. Clicking this button will invert the selection; this means all selected items will be de-selected, while all others will be selected.
- **Mark By Pattern Matching**. Clicking this button will open a new dialog box; there you can select or deselect items by applying one or more patterns to the list of drawing files. The following picture shows the dialog box:



Patterns Matching Dialog Box

The meaning of the pattern matching is not the same as DOS/ WIN - file names, but the same as the 'wcmatch' function. See the "Customization Manual" for more information.

Here is a short summary of special selection characters:

- \* = any sequence of characters; ? = any single character;
- @ = any single letter; # = any single number;
- . = any non-alphabetical character; , = separates patterns;
- ~ = not the following match (works only at the first position)
- [..] = any character inside                      .- = character in the range
- [~] = any character, which is not inside of the brackets
- ` = suppresses the special meaning of the following character

- **>>** **Delete selected from the list.** Use this button to remove the selected items from the list.
- **>>** **Clear list** .Use this button to fully clear the list. *Note: This function does not delete any file from your file system.*

### List Files

List Files are stored lists of files that can be recalled for later use. There are four buttons options for this function:

- **Save >**. Once a list of files is composed from the previous steps, this list can be saved for future using the button.
- **< Load**. After pressing the button, you retrieve a previously saved 'List File' which contains a list of drawing files. Repeating the command will exchange the current file list with the new one selected.
- **<< Append from**. Activating the button will add the contents of a 'List File' to the current list.

- **Append to>>**. Activating the button will add the contents of the current list of files to a selected 'List File'.

*Note: You can generate a drawing file list using the old DOS-Command dir:*  
> dir \*.flx /S /B > C:/TEMP/draw.lst

### List Order

There are five buttons to help you define the order of processing:

- **Sort**. The Sort button will sort the list in alphabetical order. If you apply this function, duplicate items will be removed too.
- **Reverse**. The Reverse button turns around the order of the items inside of the list box.
- **Move up**. The Move Up button will move all selected items by one item.
- **Move down**. The Move Down button moves all selected items down by one step.
- **Move to top**. The Move to Top button moves all selected items up to the top of the list.
- **Move to end**. The Move to End button moves all selected items to the end of the list.

*Note that all of these functions for changing the list order, leave the selection unchanged.*

### INFO/STATE

This area of the BATPROC dialog box, located in the lower left corner, informs you about the process and the results of all list actions. On the first line, you can see the current state - i.e. how many files are in the list and how many are selected. The second line shows you the progress of the current action, and if ready, the result of this action.

### DEFINING THE ACTIONS FOR BATCH PROCESSING

After composing the list, the actions for all items of the list have to be specified. The related fields are placed on the right side of the dialog box.

## Printing/ Plotting

The area of the BATPROC dialog box labeled 'Print / Plot' is to 'output' the selected files. This output is transferred to either a device (such as a plotter or printer) or a raster file (BMP or WMF format). There are six options for this function which include the following radio buttons and check boxes:

- **View.** Activating this radio button process a file to the current view at the time the drawing was last saved.
- **Limits.** Activating this radio button means file(s) will be processed at each drawing limit(s) or extents.
- **To printer.** Activating this check box will send selected files to the printer or plotter.  
*Note: Before you start the batch printing, it's necessary to select the printer and the options for printing. It's highly recommended to open one of the drawing files which are be plotted and print this using the set options. These options are defaulted for each drawing of the list.*
- **Into BMP-file.** Activating this check box will 'plot' the selected files to a BMP file (Bitmap).
- **Into WMF-file.** Activating this check box will 'plot' the selected files to a WMF file (Windows Metafile)
- **Overwrite Existing files.** Activating this check box will determine if existing BMP and WMF files, of the same name, will be overwritten or not.

## Converting Files to Other Vector Formats

In addition to batch plotting & raster conversion features, the **Convert to** section of the BATPROC dialog box allows you to convert selected files to one or more other vector formats (FLX, DXF or DWG). The AutoCAD DWG formats will be written for Release 12. *Note: For multiple file type conversions it's highly recommended to switch off the **Overwrite Option**, because enabling it can destroy the original drawings.*

### Conversion Overwrite Options

It's possible to overwrite existing files in converting files. To do this, activate the  **Overwrite existing drawings** check box. Once this option is activated,

there are three possible modes of operation, which are chosen by means of radio buttons:

- **Not multiple.** This radio button prevents overwriting the file, if it has already written or if it is inside of the drawing file list, which will follow. As an example assume you have the composed a list, which contains both a file conv.flx and a file conv.dwg. These files would be converted into both the DXF and the DWG format. If the unlimited overwriting was set, the first file conv.flx would be open and saved as conv.dxf and as conv.dwg. It would overwrite the existing file conv.dwg too. Because this file is also in the list, it would be opened again. To prevent such occurrences, it's highly recommended that you use the **Not Multiple** option to prevent overwriting of drawing files. The technical background of this option is, that for each file that is to be overwritten, another one is checked. If this file has already been written by the Batch Process, or if it is part of the remaining list, the file won't be overwritten.
- **Not refresh itself.** This radio button prevents the file from overwriting itself.
- **Always.** This radio button allows overwrite each selected file, including itself.

### LogFile Options

The Logfile section of the BATPROC dialog box is located in the lower right corner. It has two variables which activate the logfile option and create a name for that logfile.

**Create Logfile.** Activating this check box allows you to define a logfile, which will be written during the batch processing.

button. You can select a Logfile filename, by using this button. If a chosen filename already exists, it will be overwritten. It's useful to leave this option activated.



## Customization: Palette and Resource Manager

This chapter discusses FelixCAD functions that allow you to customize the user interface. The following table contains a summary of the available commands found in the **File** menu:

Command	Function
PalMan	Palette Manager
Palette	Palette
Menu	Menu
Tablet	Load tablet menu
TabSection	Load tablet section
Macro	Execute macro

This chapter gives you an overview of some of the customization possibilities with FelixCAD. For complete details, refer to the *Customization and Programming* manual.

### Palette

A palette (or toolbox) is a collection of buttons that represent a common group of commands. You can open up to ten palettes simultaneously, and place them anywhere on the screen.

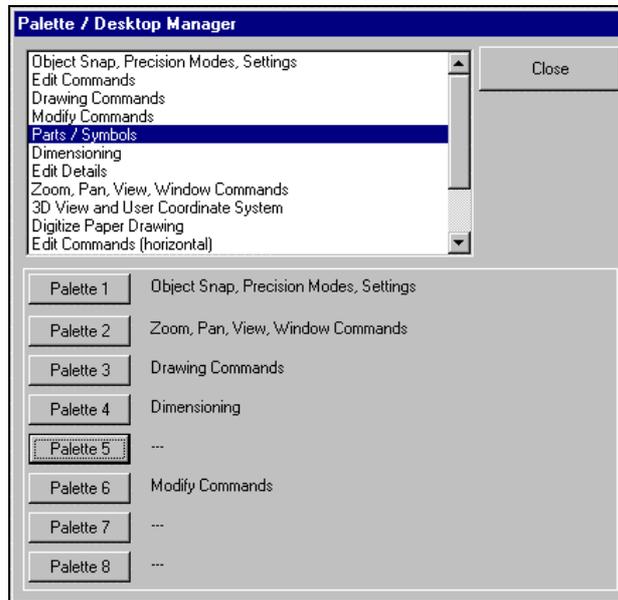
Palettes allow you to quickly access commands; to execute these commands, you need only select the button with the cursor.

In general, all of the commands found in a given pull-down menu are contained in a single palette. FelixCAD, however, includes some palettes in which the commands from several pull-down menus are combined for practical reasons.

Use the **Dialog and Menu Editor** to create your own palettes, or to customize existing palettes for particular tasks. Detailed instruction on this customization is contained in the *Programmer's Guide* manual.

## PALETTE MANAGER

You control the display of palettes with the elect **Palette / Desktop Manger**. Type the **PalMan** command or select **Palette Manager** from the **File** menu. The following dialog box opens:



The upper half of the dialog box lists all palettes currently at your disposal. This list displays the names of all palette files with the extension *.mnp* in the directory defined during program configuration the menu folder (or sub-directory). By default, this is the *\applic* folder. Instructions for this configuration are found in the *Programmer's Guide* manual.

To open one or more palettes:

1. From the list, select the palette you want displayed.
2. Notice the ten buttons, designated **Palette 1** through **Palette 8**. Click one of the ten buttons. This assign an identification number (for example, Palette 1) to the palette. If there is another palette with the same ID number already open, that palette is closed and is replaced by the newly selected palette.
3. Click **Close**. FelixCAD opens the palette.

To add customized palettes to the **Palette Manager**, edit the **Fcad.Pal** file, which is located in the *\applic* folder. The structure of the file is:

```
Palette description 1  
palett1.mnp
```

```
Palette description 2  
palette2.mnp
```

```
Palette description 3  
palette3.mnp
```

and so forth.

The palette description is found in the list shown in the **Palette Manager** dialog box. The line following always names the palette file. You can change or delete the entries as you wish.

## COMMANDS FOR PALETTE CONTROL

### Palette

The **Palette** command (**File > Resource Manager > Palette**) opens and places a palette on the FelixCAD desktop. The **Palette** command differs from the **Palette Manager** in that **Palette** lets you chose the palette from any folder (sub-directory). It is not necessary that the palette files be situated in a particular directory.

**Palette** can only be executed when *less than ten palettes* are open. When the maximum of ten palettes is open, the message “A maximum of ten palettes is available!” is displayed and the command stops.

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

The **PalClose** command removes a palette from the desktop. Specify the Palette ID number, such as P1, P2, etc. (you find the number displayed on the title bar of the palette) or type the name of the palette, such as Dimension. The corresponding palette is closed.

## **MENU**

FelixCAD is designed to work with a variety of pull-down menus. The definitions of the pull-down menus are located in files with the extension of *.mnu*.

You can create your own pull-down menus using the **Dialog and Menu Editor** program. You will find detailed instructions on this procedure in the *Programmer's Guide* manual.

To load a pull-down menu follow this procedure:

1. Select **File > Resource Manager > Pulldown Menu** or type the **Menu** or **Pulldown** commands.
2. Select a menu *.mnu* file from the dialog box.
3. Click **OK**.

## **DIALOG AND MENU EDITOR \***

With the **Dialog and Menu Editor** (DME) FelixCAD provides a powerful tool for customization. You open the DME with the **DlgEdit** command (**File > Resource Manager > Run Dialog and Menu Editor**).

When the file dialog box appears, use the **Files of type** drop list to specify the type of the resource file to be edited:

<i>.dlg</i>	Dialog file
<i>.mnp</i>	Palette file
<i>.mnu</i>	Menu file

Detailed information on customizing these items and using the DME is provided in the *Customization and Programming* manual.

## **MACROS \***

Using macros automates the execution of a sequence of command. A *macro* minimizes repetitive keystrokes. You create a macro by saving the command sequence in a *.mcr* file, which can be executed as many times as you require.

The procedure for this is as follows:

1. To execute a macro, select **File > Resource Manager > Run Macro** from the menu, or type **Macro** at the command prompt.
2. Select a macro file (extension *.mcr*) from the file dialog box.
3. Confirm the selection by clicking **OK**. Notice that the macro immediately begins to run.

There are further instructions for creating macros in the *Programmer's Guide* manual.

## **Load LISP File \***

The **LoadLisp** command (**File > Resource Manager > Load LISP File**) allows you to load an FLISP file via a file dialog box. Detailed information on the built-in FLISP interpreter is found in the *Programmer's Guide*.

## **Editor**

The **Editor** command (**File > Resource Manager > Editor**) calls the Windows Notepad editor, which allows you to edit FLISP files or other resource files (such as linetype and 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 **ListFile** command (**File > Resource Manager > List File**) displays a specified file in a list box. This allows you to read the resource file without editing it.

## ***DIGITIZING TABLET*** \*

FelixCAD supports the use of a digitizing tablet as an input device and for cursor control. The tablet lets you enter coordinates for drawing points, as well as select commands via icons. Before you can use the tablet, you must load an *.mnt* tablet menu file. This is performed automatically when FelixCAD starts, if the tablet configuration had been selected during program configuration.

The **Menu** command enables you to change the tablet menu file in the same way as described previously for the pull-down menu file. To load a tablet menu follow this procedure:

1. Select **File > Resource Manager > Menu** from the menu. You can also type the **Tablet** command.
2. Select a tablet menu file from the dialog box. If necessary, change the file type to *.mnt*.
3. Confirm the selection by clicking **OK**.

A tablet can also be loaded by selecting the symbol for program customization. In this case you must select a *.mnt* file. The remainder of the selection is made according to the previously described procedure.

## **Replace Tablet Menu Section** \*

The **Replace Tablet Menu Section** option 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 **TabSection** command. 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 Windows WinTab standard to interface with digitizing tablets. Contact the tablet manufacturer for the necessary Windows device driver. When the tablet is working as a standard Windows pointing device, then it can be configured for FelixCAD.

FelixCAD supports the tablet in two modes: digitizing and command entry. When set to *digitizing* mode, you can trace over paper 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 optionally 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 eight menu areas.

Tablet configuration is done using the **TabConfig** command. This command loads a dialog box 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 by 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 **DigiConfig** command. The program asks for the value of two coordinates and then asks you to pick the corresponding location on the plan.

You turn on and off the digitizing function with the **DigiOn** and **DigiOff** commands. The program has a digitizer palette called the “Digitize Paper Drawings” that can make the process easier.

## Editing Tablet Menus \*

The tablet menu commands are stored in an ASCII file that can be edited using any standard text editor, such as the Windows Notepad. (If you edit the file with a word processor, such as WordPerfect, make sure 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 two primary areas of the file: (1) the **SETTINGS** area, which defines the row and column layout of the menu areas; and (2) the **FIELD** area, which defines the commands that are 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 eight 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 the **Esc** key (twice in a row) 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 edit from it there.



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