FastCAD v7.5x

FastCAD v7 software designed and written by

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Chapter 1

Introduction

Welcome to FastCAD v7 and FastCAD v7 LE!

FastCAD v7 is a general purpose **2d/3d CAD** platforms for your **Windows 95/98/NT/ 2000 and XP** operating systems, based on the award-winning software that introduced thousands of users to the world of **CAD (Computer Aided Design) in 1986**.

FastCAD v7 was designed with three things in mind: ***power***, ***speed***, and ***ease of use.*** Using the product of this unique design philosophy, you'll soon be busy producing professional level **CAD** drawings.

Like our software, this reference manual is a tool—designed to guide you through your design and drafting projects. The user manual is simple, yet thorough enough for both the novice and the experienced **FastCAD** user.

This manual is divided into two main sections:

The ***Introductory Chapters*** (**Chapters 1 through 5**) provide preparatory information about interfacing with **FastCAD v7**, and instructional information about general **CAD** concepts. These chapters are especially helpful if you are new to **CAD**.

The ***FastCAD v7 Reference*** (**Chapters 5 through 24**) documents all the commands available to you through the ***Menu bar***. Refer to these chapters as questions or situations arise.

The Internet

The Internet offers us unprecedented convenience and flexibility in communicating with users like yourself. We strongly encourage you to regularly visit our World Wide Web site for updates, add-ons, news, etc.

www.fastcad.com

Manual Conventions

Read these Sidebars

Occasionally you'll see sidebar text such as this. Typically, you'll find tips or background to help you use the associated topics more effectively.

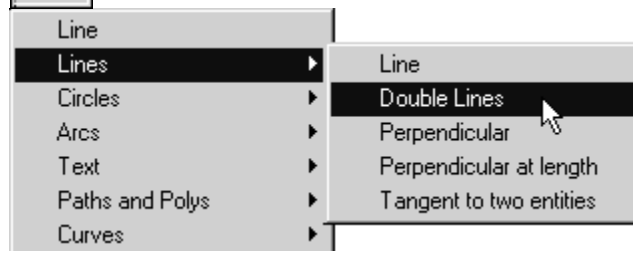
Through the course of this manual, we have attempted to adhere to written conventions to make your interpretation of instructions and descriptions clear.

CAPS are used for **COMMANDS**, such as **LINE** or **REDRAW**.

[Menu commands] are referred to in ***brackets*** **[Draw > Lines > Double Lines].**

An example of a ***nested*** menu command is:

Draw > Lines > Double Lines



Filenames and named components of **FastCAD**'s drawing screen, such as ***named menus, named icon bars,*** **or** ***named dialog boxes***, are shown in **italics**. Terms of ***general emphasis*** are also **italicized**.

Boldface is used for ***dialog box*** **items**, **options**, and **button labels**.

Where appropriate, "**command line prompts**" appear in quotes. "**Items you are instructed to type**" also appear in quotes.

KEYS on your keyboard, such as **SHIFT** or **SPACEBAR**, are typed in **caps**.

The instructive word "**click**" implies pressing the ***left*** mouse button. This action is sometimes explicitly termed a "**left-click**". The term "**right-click**" means press the ***right*** mouse button. The term "**middle-click**" means press the ***middle*** mouse button which is a ***scroll wheel switch*** (with the understanding that standard mouses may not support the ***scroll wheel switch***).

The instructive word "**select**" implies pointing the screen cursor to an item or object and ***left-clicking*** the mouse.

The instructive word "**pick**" implies pointing the screen's ***pick box*** cursor to a drawing entity and ***left-clicking*** the mouse.

The instructive word "**choose**" implies the selection and subsequent clicking of a button on a dialog box.

The term "**entity**" is reserved for anything that appears on the actual drawing screen. Anything you can put on the drawing screen (such as a ***line***, ***symbol***, ***dimension***, ***text***, etc.) is an entity.

System Requirements:

- A Pentium II or newer CPU
- **MicroSoft Windows 95**, **98**, **ME**, **2000, NT, XP Home** and **XP PRO** operating systems.
- At least **128MB** random-access memory (**RAM**), **256MB** recommended for better **Windows** performance. The more the better!
- **24** or **32 bit** display mode capable VGA card.
- **CD-ROM** or **DVD** drive

  (Software is available on 3.5" floppy disk also).
- 4 MB available hard-disk space.

  (Future add-ons and enhancements may require additional space.)

Installing FastCAD v7

Before you get started, it's a good idea to return your postage-paid registration card. Registered owners receive some exclusive services, including telephone technical support.

Insert the **FastCAD v7 CD** into your **CD ROM** drive and the installation window will automatically appear.

If it does not appera after a few seconds double-click the "**My Computer**" icon on your Windoes Desktop, double-click the **CD ROM** icon and then the **"Autorun.exe"** file displayed in the window that appears. Click on the desired option to install **FastCAD v7,** the **Symbol Catalogs** and other options.

During the installation of **FastCAD v7** you will be prompted for your ***name***, ***organization*** and ***serial number***. If you haven't received your ***serial number*** press the **Next button** to continue the installation. You will receive by email your unique ***name***, ***serial number*** and ***authorization code***.

Until you receive this information your **FastCAD v7** will display a dialog box on startup that says "Y**ou have 14 of 14 working days of full function remaining**" with a button labeled "**Enter Authorization Code**" . This allows you 14 days of working with **FastCAD v7** to receive your unique ***name***, ***serial number*** and ***authorization code*** from **Evolution Computing Inc.**.After you receive this information **CLICK** the "**Enter Authorization Code**" button and enter them in the appropriate boxes. When **FastCAD v7** is properly authorized the "**You have 14 or 14 working days of full function remaining**" screen will not appear on startup.

If you have to reinstall **FastCAD v7** again you will need to re-enter this information so write your ***name***, ***serial number*** and ***authorization code*** somewhere so you won't loose it. Beter yet print several copies of the email you received with this information.

Support

Registered users are eligible for free telephone support (local and long-distance phone charges may apply).

In the United States and Canada, call our **Technical Support line between 9:00 AM and 5:00 PM Mountain Standard Time** (**Arizona does not observe Daylight Savings Time!**). We prefer that you be close to your computer, if possible. Please have your serial number at hand, ***since support is available to registered users only***.

If you purchased your copy of **FastCAD v7** outside of the U.S. or Canada, please consult your local dealer. He or she is your best source of convenient local support.

In addition, be prepared to provide the following information:

- Your version of **FastCAD v7**, always available from **[Help > About FastCAD…**] from the menu, or by typing "**VER**" at the command prompt.
- Computer name and model, as well as the name and model of any relevant peripheral components.
- Whether you are using **Windows 95/98/98ME/2000/NT/XP**, and what version if applicable.
- Specific instructions to reproduce your problem.

Technical Support Line: (480) 967-6967
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On the Internet, visit our website to find the appropriate e-mail addresses to direct correspondence/attachments.

www.fastcad.com

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Key Features

If you've used the **DOS** version of **FastCAD**, you'll notice that this version is designed to integrate itself fully within the **Windows** environment. **FastCAD v7** uses the same video card drivers, printer/plotter drivers,and **TrueTupe** fonts as the rest of your **Windows** applications. Otherwise, **DOS** users will see virtually all of the features and functionality of previous **Evolution Computing Inc.** products and more.

New features include:

- **True 32-bit double precision programming with full Windows 95/98/2000/ NT, XP Home** and **XP PRO support**, including better file selection support of long file names, and print dialogs that support multiple printer selection.
- **256 Color Support,** with large color selection and definition dialog, custom palettes, etc.
- **Much superior linear dimension insertion and editing**.
- **New Path/Poly DYNAMIC EDIT** command is much more powerful.
- **Dynamic Text Cursor** lets ***you see actual text before you place it***. You may dynamically ***resize*** and ***rotate*** the text before placing it.
- **New entity selection is much faster**. You are automatically in **each** or window pick mode. A right-click opens a popup menu with further selections such as by ***layer***, ***color***, ***entity type*** etc.
- **AutoCAD 2000 import/export with support for AutoCAD .SHX fonts.**.
- **Multiple UNDO/REDO/FLIP.** up to 10 levels**.**
- Ability to **specify Line Widths to geometric sizes** as well as **fixed width Pen Thicknesses** for printing solid filled lines up to 2.54mm / .1” lines..
- **Define & Extract attributes defined in Symbols**.
- **File Notes**. Embed notes about each drawing that do not display on the drawing.
- **More powerful Convert function**. Convert entire folders of drawings from one format to another (**FCD, ECW & FCW to FC7 and FS7, DXF/DWG to FC7 etc.)**.
- **Template Files**. Create a **Template file .FT7** with all of your customt layer named, text properties, pen thicknesses, line styles etc. that you want when you start a new drawing.
- **Recently Used Files List.** Pick one of the last four files opened.
- **Parts files can be any valid import forma**t (**DWG**, **DXF**, **FCD**, **FCW**, **FC7**)
- **Change Multiple Text entities to read the same.**
- **Single or double pick INTERSECTION** modifier.
- **FRONT**, **ABOVE**, **BELOW** and **BACK** Commands.
- **Replace Symbols with other Symbols**. Pick the **Symbol Definition** you want to change the **References** of and then pick the **Symbol Definition** you want to change the **References** to and all Symbols in the drawing update to the newly selected **Symbol Definition**. The **Symbol Definitions** are not effected.

- **Purge Unused Symbol Definitions**. Quickly **eliminate Symbol Definitions** that have **no Symbol References** in the drawing. This excess baggage can make your drawing much larger than it needs to be. This is the case with most **AutoCAD DWG** or **DXF** drawing that you import.
- **Tooltips for icons**, **colors** and **property status window**s.
- **Adjustable pick cursor aperture size in PIXELS**.
- **Old file formats automatically update** on load or part insertion — no longer a two step process.
- **EXTENTS** info command. Shows the x, y, z dimensions of the active drawing
- **KEEP LIKE** entity modifer to set all properties of a specified entity as current.
- **Full 3d/2d drawing capabilities with rendering.**
- **Open multiple separate drawings files at the same time**.
- **Open multiple Overlay and Reference drawing**s that share the same coordinate space.
- **Draw and define camera paths for animated fly-throughs of your 3d drawin**g.
- **Use multiple Symbol Catalog viewers** to select **Symbols** from two or more **Symbol Catalog** files **.FS7** at the same time.
- **TABLE** command allows you to quickly define a **TABLE** of **ROWS** and **COLUMNS** of **TEXT** like a spreadsheet. The **TABLE** is dynamically editable by clicking in a **CELL** and typing or editing the text already there. **Column width** is dynamically adjustable by dragging the vertical devider with the mouse left or right. **CSV** data may be exported from a **TABLE** ot a spreadsheet or imported from spreadsheets into a **TABLE.**
- **Clone and Edit Symbol Definitions** for automatic updates of existing **Symbol References** in the drawing. Eliminates the need to draw and define a new **Symbol** just for minor changes.
- **Quick access to all properties from on-screen status window drop-down menus. If you click the drop-arrow at the right end of a status w**indow, you may pick the property from a menu. If you click in a status window the full dialog box appears for that property.
- **Customization of icon bars now supports standard Windows .ICO i**con **files instead of the previously used** .**BMP format**. This will allow variable size icon selection in future updates of **FastCAD v7**.
- **Greatly enhanced Macro language** now allows entity selection by entity type as well as the use of hot keys from inside **Macros**. **FastCAD 32 Macros** may now be run in **FastCAD v7** by a setting in the **Options** dialog.
- **Each drawing window now has its own set of icons that control its view settings with hidden line on/off, perspective on/off, render view on/off, make active window, move window, minimize/maximize and close.**.

- **Dynamic pan, zoom, roll, picth and yaw the drawings view using the left mouse button and one keyboard button.**. By clicking the center button of a three button mouse or the wheel switch on a wheel mouse, you are in dynamic pan and zoom mode. Hold the left mouse button down while moving the mouse and the drawing pans in real time. Hold the **SHIFT** button down and zoom or out by moving the mouse forward and backward. Hold the "**R**" key down and spin the mouse to roll, the "**P**" key to ptich up or down, the **Y**" key to yaw left or right and the "**ALT**" key to move the 3d view closer to or further from the 3d entities.
- The **Current Time, Current Date** and **Current Drawing** may be inserted as **constantly updating TEXT entities**. As you work the **TIME** and **DATE** are updated so when you **PRINT** the drawing you will have the **exact TIME** and **DATE** on the printed sheet. If you want them to remain fixed they may be **Exploded.**
- **HEXAGONAL GRID SYSTEM** allows for quick **isometric** drawing.
- An ***entire directory of individual drawings*** may be inserted into a drawing and ***automatically be defined as*** **Symbols.** The drawing name of each is used as the **Symbol Name** and its original **0,0 ORIGIN** is used as the **Symbol's ORIGIN** as its handle when placing a **Symbol** from the **Symbol Catalog Viewer.** The drawings may be in **.FCD, .ECW, FCW, .DXF** or **.DWG format** and **will be converted on the fly into the new .FS7 Symbol Catalog file format.** The **.FS7** file when saved may be opened in the **Symbol Catalog Viewer** and each **Symbol** picked from the **thumbnail pictures**.
- **Extrude 2d entities** into **3d space. Define Workplanes** in **3d space** and draw **2d entities** on them and **Extrude** them into the **Z axis** as if the **Workplane** was on a **2d XY plane**. This makes drawing in **3d** much easier. You can build from the **2d XY plane** into **3d space.**
- **Animate a 3d model**. A **Spline, Circle** or other entity may be drawn in a **3d model** and defined as a **Camera Path.** An **animated fly through** with a defined number of steps may be viewed along this **Camera Path**.

FastCAD Command Menus

FastCAD's commands are grouped in the pull-down menus according to their basic functionality.

File commands manage drawings and files.

Clip commands give you access to undo and the Windows Clipboard.

View commands control what you see in the drawing window.

Draw commands add new entities — like lines and circles — to your drawing.

Draw3 commands add new **3D** entities — like **3D** lines, Cylinders, and Spheres — to your drawing.

Dimensions commands to draw linear and angular dimensions of entities.

Edit commands modify existing drawing entities.

Copy commands let you create multiples of existing entities.

Insert commands manage symbols and parts.

Viewports commands allow you to draw an enclosed entity such as a **polygon** or **circle** and defin it as a viewport and copy a zoomed view into it. This is great for displaying enlarged detail insets os areas in the drawing

Render commands control the rendering environment for **3D** objects.

Animate commands control the real-time fly-through and orbit of **3D** objects.

Specs commands control your basic drawing environment.

Info commands report on entities and perform calculations and measurements.

Macros commands let you load, edit, and save your own routines.

Mod commands work within drawing or editing commands to precisely select existing points or entities.

Tablet commands control the use of a digitizing tablet with **FastCAD**.

Help commands access help files and information for using **FastCAD**.

If you are new to **CAD**, read the introductory chapters, then try the quick start tutorial. When completed, you'll be well equipped with the basic fundamentals to tackle your own, more challenging projects.

If you have experience with other **Evolution Computing CAD** products, or even other **CAD** systems, you may prefer to experiment on your own, using the **FastCAD v7** manual as a reference.

Either way, you will soon learn why **FastCAD v7** has earned its reputation as the world's fastest and easiest-to-learn precision **CAD** program.

Chapter 2

Communicating with FastCAD

The Interface: Telling FastCAD what you want it to do

Better than a large drawing board, **FastCAD v7** gives you a virtually unlimited canvas to create your drawings. You'll notice that most of your **FastCAD v7** screen consists of the drawing window, maximizing your ability to see your work.

You communicate with **FastCAD v7** through your mouse and keyboard, and through the numerous screen tools visible around the drawing window. These tools comprise the "**interface**", or your means of communicating with **FastCAD**.

Most on-screen drawing tools are in the form of "bars" that contain icons, buttons, or indicators that either provide information or perform actions. Each screen element listed below is described in detail in this chapter.

The ***Title*** *bar*, located across the top, is a standard Windows screen component that displays the current drawing name and contains controls to minimize, maximize, or exit the application.

The ***Menu*** *bar*, located just beneath the ***Title*** *bar*, provides you with access to almost all of **FastCAD**'s operating commands through the use of pull-down menus.

The ***Status*** *bars*, located beneath the ***Menu*** *bar*, as parts of the default set up are called: the ***Tracking*** *bar*, which provides indicators for cursor tracking, font name, text height, pen thickness and line width; the ***Style*** *bar*, providing the current text style, dimension style, coordinate system, and sheet name; and the ***Properties*** *bar*, indicate drawing file name, current layer, current line style, and current file style. You can left-click on any indicator to change current modes.

The various ***Icon*** *bars* are located along the left and right edges of the screen and contains icons for drawing, editing, zooming, specs, and 3D view control, and more.

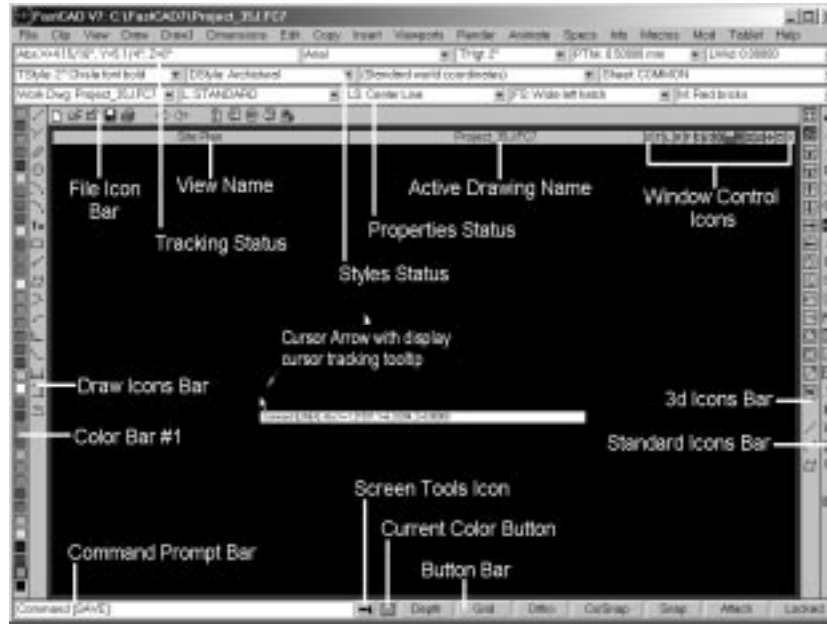
The ***Color*** *bar*, located along the left edge of the screen, allows you to change the current drawing color with a simple click.

Screen Tools icon

The ***Button*** *bar*, located on the bottom edge of your screen, contains the command prompt for keyboard entry of commands, and buttons that allow you to customize the drawing environment. **FastCAD v7** also has a unique ***floating command bar*** that follows the cursor as you move it across the screen .You can customize your screen by showing or hiding the ***Status*** *bars*, ***Color*** *bar*, and ***Icon*** *bars*. To do so, use the ***Screen Tools*** *icon* located on the ***Button*** *bar* and check the desired options ***on*** or ***off***.

The FastCAD v7 Screen

When you start **FastCAD v7** for the first time, you see a screen with a ***Menu*** *bar*, three horizontal ***Status*** *bars*, ***a Color*** *bar*, four ***Icon*** *bar*s and a ***command prompt*** bar with ***Buttons*** surrounding a large drawing window that contains an arrow-shaped cursor.



The ***Title*** *bar* runs across the very top of the **FastCAD** window. It displays the name of the current drawing you are working on, as well as controls to minimize, maximize, or close the program. If the **FastCAD** application window is not maximized, you can click on the ***Title*** *bar* and reposition **FastCAD** on your Windows desktop.

The ***Drawing*** *window* is most important part of your screen. This is where you view, draw, and edit your drawing. **FastCAD** allows you to open multiple windows for each drawing, and shows you different levels of detail. You can also open mutliple files. Multiple files can appear in the same drawing windows (as overlays) or in completely separate drawing windows.

The ***Menu*** *bar* across the top of the screen contains the titles of the pull-down menus from which you can choose almost every **FastCAD** command.

Most of the remaining screen elements can be toggled off using the **TOOLS** command **[Specs > Screen Tools…]**, also accessible by choosing the hammer icon at the bottom of the screen.

The ***Tracking*** *bar*, located beneath the ***Menu*** *bar*, displays the current coordinates of the cursor and up-to-date drawing information such as the current font name, text height, pen thickness, and line width. You can quickly change these settings by clicking any indicator on the ***Tracking*** *bar*.

The ***Styles*** *bar* beneath the ***Tracking*** *Bar*, shows text styles, dimension styles, workplane coordinates, and the current sheet name.

The ***Properties*** *bar* beneath the ***Styles*** *Bar*, shows whte current file name, Layer name, line style, and fill style.

The ***Color*** *bar* on the left shows the colors available on your display. The current color is marked with a heavy black outline. You can use the Color bar to select a different color at any time, even in the middle of drawing or editing.

The ***Draw*** *Icons bar*, located next to the Color bar, offers you quick access to the most commonly used drawing commands.

The ***Edit*** *Icons bar* offers you quick access to common editing commands.

The ***Standard Icon*** *bar*, located to the right, gives you quick access to several useful commands such as zooms, layers, and entity properties.

The ***3D Icon*** *bar*, located to the right, gives you control over the **3D** views and **3D** drawing environment of **FastCAD v7**.

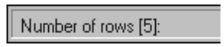
The ***command prompt*** along the bottom lets you and FastCAD communicate. Usually, the prompt reads “Command:”. **FastCAD** is asking, “What is your command?”

The ***Button*** *bar* is located at the right of the command prompt. These buttons enable you to quickly toggle features on or off. In some cases, you can edit the properties of the feature by right-clicking on the button.

Use the cursor to choose commands from the pull-down menus and to select screen locations. The cursor is a small arrow which changes shape during drawing and editing, usually to the crosshairs, stretching line (rubber band), stretching rectangle, or pick box.

The Command Prompt

Some examples of FastCAD’s prompts:



The ***Command*** *prompt* lets you and **FastCAD** communicate through the keyboard. You can enter commands (telling **FastCAD** what to do), or enter input during a command (answering questions **FastCAD** asks). When the ***Command*** *prompt* reads “**Command:**”, **FastCAD** is awaiting your next action. All menu commands have text equivalents that may be alternatively typed on the ***Command*** *prompt*. The previous command is displayed in [**brackets**]. To auto-repeat the previous command, either click the mouse button, press the SPACEBAR, or press ENTER.

In mid-command, the ***Command*** *prompt* may ask for any additional information (like locations, angles, numeric values, etc.) it needs to complete the current action. You can either type a response, or indicate it visually with the cursor, depending on what is asked. Press ENTER or right-click after typing the prompt input. Right-clicking without entering a keyboard response will cause **FastCAD** to accept the default, which is displayed in [**brackets**].When “[**dialog**]” is displayed, you have the option of submitting input via a dialog box. Right-click to display the dialog box appropriate to the active command.

Using the menus

FastCAD's entire command set is accessible through the pull-down menus. The menu is always located at the top of the drawing screen, and it cannot be disabled.

To invoke a menu command, move the arrow cursor to the ***M*enu** bar. When the cursor is over the title of the menu you want, left-click. A pull-down menu appears with several commands. With the mouse, move the highlight up and down until it is over the command you want, then left-click again. If you pull down the wrong menu, de-select it by left-clicking the menu title, or pressing the ESC key.

FastCAD's menu commands are grouped according to their function type:

- **FILE**—manage drawings and files.
- **CLIP**—undo and Windows clipboard commands.
- **VIEW**—control what you see in the drawing window.
- **DRAW**—add new entities (like lines and circles).
- **DRAW**—add new 2d entities (like lines and circles).
- **DRAW3**—add new 3d entities (like lines, circles, spheres and cylinders).
- **RENDER**—controls rendering environment for 3D entities.
- **ANIMATE**—acontrols real time fly-throughs and orbits for 3D entities.
- **DIMENSIONS**—add measurements.
- **EDIT**—modify entities that you have already drawn.
- **COPY**—duplicate existing entities.
- **INSERT**—manage symbols and parts.
- **SPECS**—control your drawing environment.
- **INFO**—perform calculations and measurements.
- **MACROS**—load, edit, and save your own routines.
- **MOD**—work within commands to refer to existing entities.
- **TABLET**—calibrate and configure your **WinTAB** compatible digitizer.
- **HELP**—access **FastCAD**'s on-line **Help** facility.

Some menu choices open sub-menus which provide additional command options. **FastCAD** never nests deeper than this level.

Menus can be modified with any text editors. For more information on menu customization, please consult **FastCAD**'s on-line **Help [Help > Contents**].

The Button Bar

The ***Button bar*** is located along the bottom edge of the screen. In the course of a typical drawing, you will likely access these buttons on a frequent basis.

Because the ***Button*** *bar* provides critical user input and feedback functions, you cannot toggle off its display.

The ***Command*** *prompt* is located on the left edge of the ***Button*** *bar*. The ***Command*** *prompt* lets you and **FastCAD** communicate through the keyboard.

The **Screen Tools** icon lets you select which screen elements will be displayed. Toggle items on or off using the ***Select Screen Tools*** *dialog box*. This button is the same as selecting **[Specs > Select Tools]** from the menu.

Color shows the current entity drawing color, and clicking on it brings up the **256 color** selection dialog box.

Depth toggles the **3D Depth mode on** and **off**. Activates the **3D Depth mode**. This mode lets you pick points above or below the window surface.

Grid turns the grid pattern display on or off . The Grid is a drawing aid consisting of nonentity dots displayed at regular, user-defined intervals. Right-click on this button to change grid spacing.

Ortho limits entity creation or displacement to the horizontal or vertical axes. For example, lines drawn with **Ortho** cannot be drawn diagonally.

CsrSnap (Cursor Snap) works only when the **Snap** button is on. If **Snap** is on, you can only place entities along the subdivisions of an established grid. When **CsrSnap** is on, your cursor will visually "**jump**" to the next snap points as well.

Snap places entities along the subdivisions of your established grid system. Right-click this button to alter the snap point intervals.

Attach engages the selected modifier for use in draw and edit commands. Right-click this button to change or inspect the active modifier. When ***Attach*** *mode* is on, a pick box cursor is displayed whenever a point is requested. If an entity is picked, the requested point honors the attach modifier. For example, if you are drawing a line with ***Attach*** *mode* **on**, and the current mode is **Endpoint**, the cursor will ***attach*** to the *endpoint* of any picked entity. For more information on **Attach Mode**, see **[Specs > Attach Mode]** in Chapter 12.

Locked toggles group effects **on** or **off**. When unlocked, the label changes to "**Unlocked**" to show the current state. When groups are ***unlocked***, they have no effect and their component entities can be freely edited. Toggle to "**Locked**" to restore group effects.

Icon bars

FastCAD provides you with up to eight ***icon*** *bars* that you can selectively toggle on or off or reposition in any combination:

- ***Standard Icon bar***
- ***3D Icon bar***
- ***Draw Icon bar***
- ***Edit Icon bar***
- ***Files icon*** *bar*


Screen Tools icon

To display or hide the icon bar, use the **TOOLS** command **[Specs > Screen Tools…]**, and click in the ***Control****; box*. The position of the ***icon*** *bar* can be customized by clicking on the desired control box with the arrow pointing to the side of the screen you want the ***icon*** *bar* displayed.

To see what function any icon performs, hold the cursor over the icon for a moment. A ***tooltip*** appears, telling you which command is attached to the button. Move the mouse away from the icon to hide the ***tooltip***. To turn tooltip display on or off, use the **OPTIONS** command **[Specs > Optioins]**.

Any of the icon bars can be customized. The ***Custom Icon*** *bars* contain no default icons, so you (or third party developers) can use these and still retain use of the ***Standard***, ***Edit***, and ***Draw Icon*** *bars*. For more information on creating your own icons, see the on-line Help topic on "**Customizing Icons**" **[Help > Contents]**.

The ***3D Icon*** *bar* contains tools for use with **3D**. Any possible view in **3D** space can be accessed and drawn in using the **3D** view controls in the ***3D Icon*** *bar*. **3D** drawingcommands are also easily available.

Standard Icon Bar

The ***Standard*** *Icon bar* gets detailed mention since it contains the most commonly used shortcut icons. It gives you quick access to many drawing properties you might often modify during the course of your work.

Zoom In
Zooms in 2.0 times, showing less of the drawing
[View>Zoom In]

Zoom Out
Zooms out 2.0 times, showing more of the drawing
[View>Zoom Out]

Zoom Center
Select new view center
[View>Zoom Center]

Zoom Window
Zoom into user-defined box
[View>Zoom Window]

Zoom Extents
Recalculate and zoom to extent of drawing
[View>Zoom Extents]

Zoom Text
Zoom to specified text
[View>Zoom Text]

Dynamic Pan
Activate real-time pans and zooms
[*middle-click mouse*]

Zoom Last
Restore to previous view
[View>Zoom Last]

Layer
Manage your drawing layers
[Specs>Select Layer]

Line Style
Manage the line styles FastCAD uses to draw entities
[Specs>Line Style]

Fill Style
Manage the fill styles FastCAD uses to draw entities
[Specs>Fill Style]

Dimension Style
Manage dimension formatting and options
[Specs>Dimension Styles]

Text Properties
Manage text properties, such as fonts and character styles
[Specs>Text Properties]

Select Color
Access the 256-color pallette. Customize or select colors.
[Specs>Select Color]

Edit Properties
Edit entity properties of a single, selected entity.
[Edit>Properties]

Help
Display the FastCAD on-line help file contents
[Help>Contents]

Select Active Window
When multiple windows/files are open, change the active drawing window
[no menu equivalent]

New Window
Create a new drawing window
[no menu equivalent]

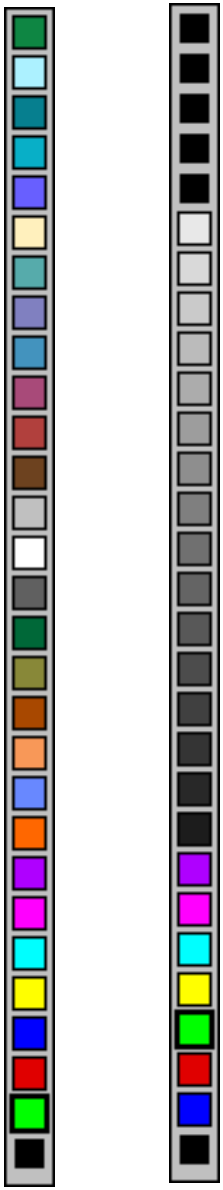
Multi-Drawing Control
Manage open multiple drawings, overlays, and references
[File>Multiple Drawings>Multi-Dwg Ctl]

Sheet
Manage sheets (add, delete, rename, reorder)
[Insert>Sheets]

Color Bar

Select Color icon

Color bar
The active drawing color has a black border. In this illustration, color #1 (green) is the active color. To the right is the *2nd Color bar.*

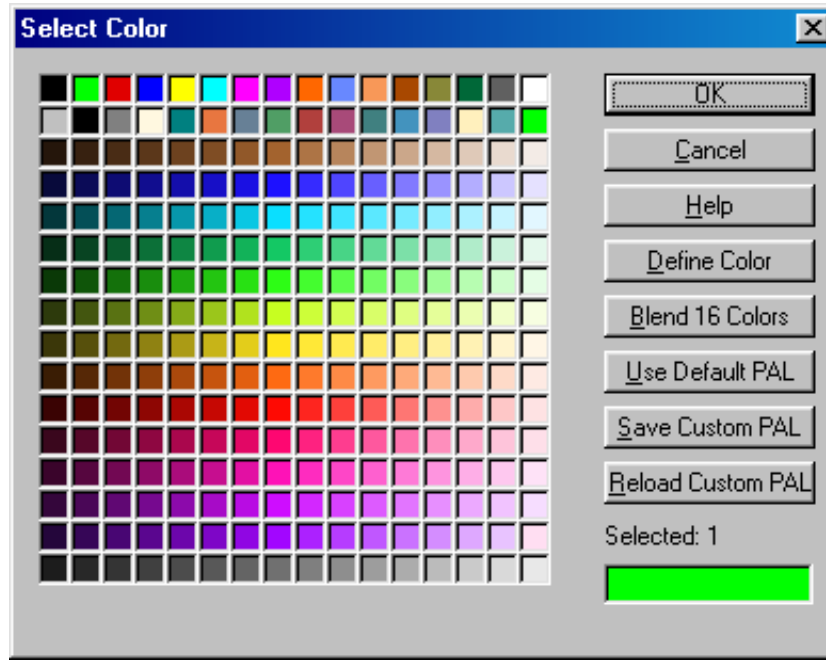


The ***Color*** *bar*, located on the left side of your screen, consists of available colors you can use in your drawing. **FastCAD** displays as many colors as your screen and resolution will allow. The ***Color*** *bar* display is anchored at the bottom (**the color black**). The active color is shown with a heavy black border (the active color is also displayed on the ***Status*** *bar*). Any entities you create are drawn with the active color.

The ***2nd Color*** *bar*, located to the right of the ***Color*** *bar*, expands your choices and flexibility of color selection. It contains the 8 basic drawing colors, plus 16 levels of Gray and 5 open postions for custom blended colors.

All 256 available colors are numbered from **0** to **255**. **Black**, always located at the bottom, is **#0**. **Color #0** (**black**) and **#15** (**white**) cannot be changed.

Below is the entire palette of **256 colors**. To see the **256 color palette**, select the **[Specs > Select Color]** command, or click the ***Select Color*** *icon*. The ***Select Color*** *dialog* lets you to choose from **256 colors**, create custom palettes, and create new blended colors.



Status Bars

Tracking bar
Styles bar
Properties bar

The three ***Status*** *bars* above display current (or active) drawing settings. The three default ***Status*** *bars*, are the ***Tracking*** *bar*, the ***Style*** *bar*, and the ***Properties*** *bar*. The individual Status boxes that make up the bars can be configured in any order or combination. To custom configure these bars, see **"Customizing FastCAD."**

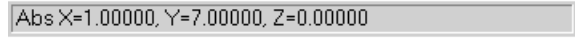
The settings displayed are in most cases, the active *entity properties* that apply to any entity you draw or create in the *active* drawing. These properties include current file name, sheet name, color, pen thickness, line width, layer, line style, and fill style, font name, text height, text style, dimension style, and current coordinate system.

In addition to the heads-up information it displays, the ***Status*** *bars* provides a very quick means of changing your current settings. Just left-click on any indicator wondow (even in mid-command) to select new settings through dialog box input. Left-click the ***drop arrow*** at the right end of most indicators to select from a menu of settings or styles used by the *active* drawing.

Tracking Status

The ***Tracking*** *status* window shows you the current coordinates of your cursor. Click on the window to cycle through the three *tracking* modes. The fourth click turns ***tracking off***.

Absolute



Relative

Polar

When on, the tracking display continuously updates as you move the mouse. Click on the status indicator to cycle through the various tracking modes: absolute, relative, polar coordinates, and off.

"Abs" indicates absolute tracking; **"@"** indicates relative tracking; and **"<"** indicates bearing angle and distance (relative polar tracking). Tracking can also be turned off. To set the display format and decimal places, use the **UNITS** command **[Specs > Units]**.

Color Status

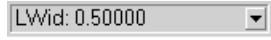
The ***Color*** *status* button displays the ***current color***. All new entities are drawn with the indicated color.This tool is especially helpful if you choose to work with the ***Color*** *bar* toggled off. To change the ***current color***, click directly on the ***Color*** *status button* to display the **256-color *Select Color*** *dialog*, then choose the ***new color***.

Pen Thickness Status

The ***Pen Thickness*** *status* window displays the ***current pen thickness***. ***Pen thickness*** is the ***display*** and ***printing*** of ***width*** for entities. All new entities are drawn with the specified ***pen thickness.*** Click the ***drop arrow*** for a ***menu*** to quickly choose one of the ***last five pen thicknesses used*** in the ***active drawing***. ***Pen Thickness*** always ***prints*** the same ***thickness*** at any scale factor.

To specify a ***new pen thickness***, click directly in the status window to open the ***Pen Thickness*** *dialog* and specify a ***new pen thickness*** in **millimeters** or **inchs**. The ***maximum*** is **2.54 mm** or **1/10".** You must type **mm** or **"** after the new value or the current value will not change..You may also ***map a different pen thickness*** for each of **FastCAD's** 256 colors. The new ***thickness*** will also appear in the drop down menu.

Line Width Status



The ***Line Width*** *status* window displays the ***current line width***. All new entities are drawn with this ***width***. Click the ***drop arrow*** for a ***menu*** to quickly choose one of the ***last five line widths*** used in the ***active drawing***.

To specify a ***new line width***, click directly in the ststus window to open the ***line width*** *dialog* abd type a ***new line width*** value. This value will appear to the drop down menu.

Layer Status

The ***Layer*** *status* window displays the ***active drawing's current layer name***. All new entities are drawn on the displayed ***layer name***. Click the ***drop arrow*** for a ***menu*** to quickle select one of the ***layers*** in the ***active drawing.***

Click in the window to open the ***Layer Management*** *dialog box* to ***add***, ***delete***, ***freeze*** *or* ***thaw*** layers. There is a ***Hide all but current layer*** check box which allows you to quickly see what entities are on the ***current layer.***

Line Style Status

The ***Line Style*** *status* window displays the ***current line style***. All new entities are drawn with the ***line style***. Click the ***drop arrow*** for a ***menu*** to quickly select one of the ***line styles*** used in the ***active drawing***.

To ***add*** a ***new line style***, click directly in the status window to open the ***Line Styles of*** *dialog box*, then select a new ***line style***. It will also appear in the drop down menu.

Fill Style Status

The ***Fill Style*** *status* window displays the ***current fill style***. All new entities are drawn on the displayed ***fill style***. Click the ***drop arrow*** for a ***menu*** to quickly select one of the ***fill styles*** used in the ***active drawing***.

To ***add*** a ***new fill style***, click directly in the status window to open the ***Fill Style*** *dialog box*, then select a ***new fill style***. It will also appear in the drop down menu.

Font Name Status

The ***Font*** *status* window displays the ***current text font***. All new text entities are drawn with the displayed ***text font***. Click the ***drop arrow*** for a ***menu*** to quickly select one of the ***fonts*** used in the ***active drawing***.

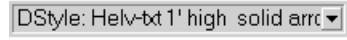
To select a ***new text font***, click directly in the status window to open the ***Text Properties*** *dialog.* Click the ***More Fonts*** *button* and choose one of the install Windows **TrueType** ***fonts***, a **FastCAD .FNT** vector font ar if installed, an **AutoCAD .SHX** vector font. The new font will also appear in the drop down menu.

Text Height Status

The ***Text Height*** *status* window displays the ***current text height***. All new text entities are drawn in this ***text height***. Click the ***drop arrow*** for a ***menu*** to quickly select one of the ***last five text heights used in the active drawing***.

To ***set*** a **new** ***text height***, click directly in the status window to open the ***Text Properties*** *dialog box*, then set a **new** ***text height*** value. The new ***text height*** will also appear in the drop down menu.

Dimension Style Status



The ***Dimension Style*** *status* window displays the ***current dimension styl****e*. All new dimensions are drawn with the selected ***dimension style***. Click the ***drop arrow*** for a ***menu*** to quickly select an availablr ***dimension style*** definted in the ***active drawing***.

To ***define*** a ***new dimension style***, click directly in the status window to open the the ***Dimension Style*** *dialog box*, then set the desired properties and add a ***new dimenstion style*** to the ***active drawing***.

Work Drawing File Name

The ***Active Work Drawing*** *file name* window displays the ***active drawing file name***. All new entities are drawn on the ***active work drawng file name***. Click the ***drop arrow*** for a ***menu*** to quickly select different open drawing as the ***active drawing file***. You may select a drawing opened as a ***separate*** or ***overlay*** file to make it the ***active drawing***.

Click directly in the status window to open the ***External Reference (Xref) Management*** *dialog box.* Use it to ***close open files*** with the ***Delete*** *button*,r ***freeze*** or ***thaw overlay*** and ***reference files*** with the "**F**" icon and ***hide*** or ***show*** them with the **H**" icon to make them available for editing.

Workplane Status

The ***Workplane*** *status* window displays the ***current workplane***. All new entities are drawn on the ***new workplane***. Click the ***drop arrow*** for a ***menu*** to quickly select a different ***workplane*** defined in the ***active drawing***.

Use the **WPDEF** command **[Specs > Define Workplane]** to define a **new** ***workplane***.

Sheet Name Status

Sheet: Sire plan

The ***Current Sheet*** *status window* displays the ***active drawing sheet name***. All new entities are drawn using on this ***active drawing sheet***. Click the ***drop arrow*** for a ***menu*** to quickly select a different ***sheet name*** defined in the ***active drawing***

To ***add***, ***delete***, ***hide, show*** or ***rename sheets***, click directly in the status window to open the ***drawing sheets*** *dialog box.*

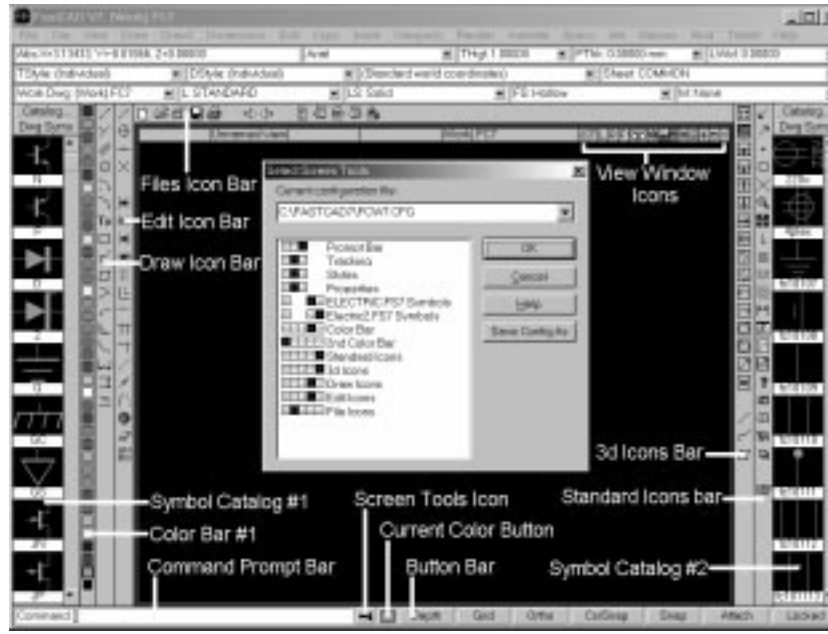
Material Name Status

M: Stone block

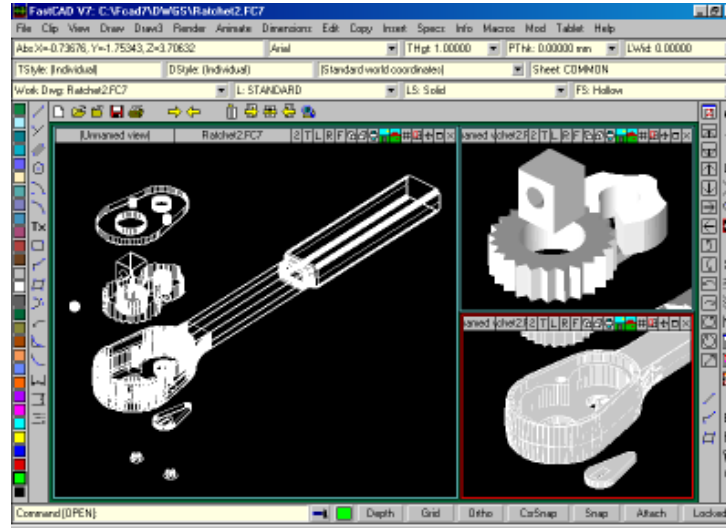
The ***Current Material Name*** *status* window displays the ***current rendering material*** for use with the **LightWorks** renderer. All new **3d** entities are ***created*** with and ***rendered*** using this ***material.*** Click the ***drop arrow*** for a ***menu*** to quickly select a different ***material***

All Status Bars, Icon Bars and Symbol Viewers On

The screen illustration seen below shows the **Screen Tools** dialog box with all of **FastCAD V7's Icon bars**, **Status bars** and two **Symbol Catalog Viewers** turned on. You may turn any or all of these off to maximize the drawing screen area if you like.



Multiple Drawing Windows



FastCAD can split your screen into a any number of windows, displaying different views of the drawing, or different drawings. Each window has its own zoom factor. Each window zooms and pans independently, but all windows are active for drawing and editing. For example, you can draw a line that starts in one window and ends in another.

Use multiple windows when you need to:

- Work simultaneously in widely separated areas of your drawing.
- Work on details in a zoomed in view while viewing the results in an overall view.
- Work with multiple (separate) drawings. Note that overlay and reference drawings always open in the same coordinate space as the base drawing, and therefore do not require new drawing windows.

To Add New Drawing Windows

Use the ***New Drawing Window*** *icon* on the ***Standard Icon*** *bar*. Click the icon and you will be prompted to provide the two corners that define the new window. Alternatively, you can type in **NEWW** at the command line. New windows can be added anywhere on the screen, even on top of one another.

Select Active Window icon in the Standard Icon bar

Make active window icon in upper right corner of each view window

To Select an Active Window

To make any drawing window *active*, you can either:

- Click the ***Make active window*** *inon* in the target window's *title bar*.
- Use the ***Select Active Window*** *icon* from the ***Standard Icon*** *bar*, then click anywhere on the target window.
- Type **SELAW** at the command line, then click anywhere on the target window.

The ***active window*** is the window affected by zoom magnification or view center changes. Note that the active window is drawn with a 1-pixel width red border just inside the main border.

To Name a Window

When a new window is created, it is given the default name "**Unnamed view**". Sometimes you may wish to give these views more descriptive names to facilitate printing, view sets, zoom names, etc. To give a drawing window a name, or to rename a window, use the **SET VIEW NAME** command **[View > Set View Name]**. This command can be accessed by typing **SVNAME** at the command line or by selecting the command from the ***View*** *menu*.

Typical window title bar. The view name in this case is "front view".

Minimize window icon in upper right corner of each view window

Maximize window icon in upper right corner of each view window

To Size or Move Windows

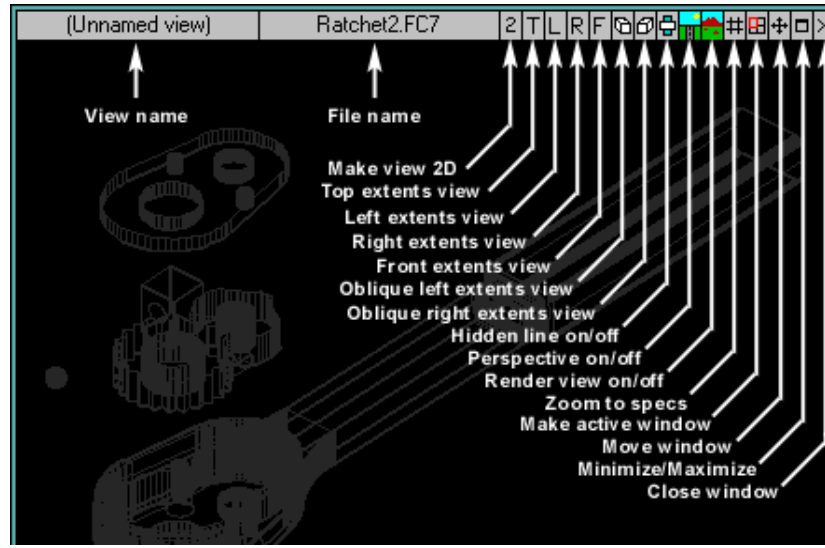
To resize a window, simply use the standard Windows procedures: Move the cursor to the edge or corner of the window border (the standard cursor changes to a two-headed arrow) and drag. To move a window, click on its ***Move wondow*** *icon* in the window's title bar of the drawing, hold the left mouse button, and drag. To ***minimize*** or ***maximize*** a window, click on the window's ***minimize/minimize*** *icon* in its title bar.

To Close a Window

Close window icon in upper right corner of each view window

To remove a window from the screen, click on the title bar's ***Close window*** *icon*. If the drawing window had a view name, the name is not deleted. The named view can be recalled in the ***active view window*** if it was saved with the **SET VIEW NAME** comand or when printing the drawing. Once the ***view window*** is closed you will need to use the ***New Drawing Window*** *icon* to create another.

Window Control Bar Icons



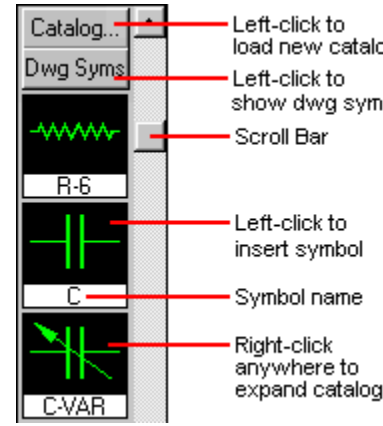
A powerful feature that is new in **FastCAD v7** is the individual drawing window control bar. Every window opened into a drawing or multiple drawings has this control bar across its top. Displayed across the bar are the following information and control icons:

View name - current view name
File name - current file name
Make view 2D - changes current view to standard 2D window
Top extents view - zooms to the extents of all objects from above
Left extents view - zooms to the extents of all objects from left side
Right extents view - zooms to the extents of all objects from right side
Front extents view - zooms to the extents of all objects from front
Oblique left extents view - zooms to the extents from a left angle
Oblique right extents view - zooms to the extents from a right angle
Hidden line on/off - toggles hidden line mode for 3D objects on or off
Perspective on/off - toggles between perspective view and ortho view
Render view on/of - toggles between render mode and wireframe view
Zoom to specs - zoom to specifications selected within view dialog box
Make active window - makes window active for zoom and rendering
Move window - allows window to be drug to a new screen location
Minimize/Maximize - toggles between full screen and small window
Close window - closes current window

Symbol Catalog Viewer

For a more detailed discussion about symbols, please see Chapter 11, Insert commands

FastCAD's *Symbol Catalog viewers* makes ***symbol management*** and ***insertion*** very easy.You can create ***symbol libraries*** and dynamically load and unload them from multiple viewers. Symbols can be selected from the loaded catalog and inserted into your drawing using a "**drag and drop**" type of interface.



The actual ***Symbol Catalog Viewers*** may be displayed veretically on the left or right side of the screen. Its display may be toggled on or off using the **SCREEN TOOL** command **[Specs > Screen Tools]** or its icon equivalent.

SCREEN TOOLS icon located in the button bar.

To insert a symbol from the catalog:

Move the cursor over any ***Symbol Catalog*** *viewer*, roll the scroll wheel, if you are using a wheel mouse, or right-click for a full screen of ***symbols*** and left-click over the desired ***symbol*** to select it. Move the cursor over the drawing screen and a ***ghost image*** of the ***symbol*** will appear. The *command line prompt* reads the following:

```
(CTRL=scale CTRL+SHIFT=rotate) Origin/Scale [INS=specs]:
```

At this point you may left-click to place one or more ***symbol references*** in your drawing, type a ***scale facter*** to change the size of the next ***reference*** you place in the drawing, or you may dynamically ***scale*** or ***rotate*** the ***symbol*** prior to placing each ***reference*** in the drawing If you press the **"Ins"** key on the keypad, the ***Insert Parameters*** *dialog box* opens You may set a ***scale factor*** and a ***rotation angle*** for the next ***symbol reference*** placement in the drawing. **FastCAD** allows you to open two ***Symbol Catalog*** *viewers* at the same time displaying to different **.FS7** ***Symbol Catalog*** libraries.

At any time you may select a ***symbol*** for insertion even in the midst of another ***symbol*** insertion.The new ***symbol*** wil aquire any ***scaling*** or ***rotation*** applied to the previous ***symbol***.

Symbol Definitions are loaded from the ***catalog*** to your drawing as soon as you left-click a thumbnail in the ***viewer***. You do not need to place a ***reference*** in the drawing. This is a quick way to make a new ***symbol catalog*** file from one or more more complex ***symbol catalog*** files.

To load a different Symbol Catalog in the viewer:

The sample catalog **ELECTRIC.FS7** demonstrates the ***Symbol Catalog*** functionality. The **FastCAD** distribution **CD** contains a number of other ***catalofs*** you may use.To load a different ***symbol catalog*** click the ***Catalog*** *button* at the top of the *viewer* to display the ***Load Catalog*** *dialog box*. The file list is automatically filtered to display only **.FS7** ***symbol catalog*** files. Simply select the desired file name and click the ***Open*** *button* or double left-click directly on the file name.

At the command prompt you may type **CATALOG** then specify a ***catalog file name*** This command is most useful in a ***custom menu***. where you can specify a menu entry to load a specified ***catalog***.

The last ***catalog*** used will be the one in use when a new **FastCAD** session is started.

To toggle between drawing symbols and catalog symbols:

The ***Symbol Catalog*** *viewer* can display either the ***symbol definitions*** in the loaded catalog file or those ***definitions*** contained within the ***active drawing only***. ***By default, the viewer displays the symbols in the currently loaded catalog***. To force display of symbols in the ***active drawing***, click the ***Dwg Syms*** *button* located at the top of the viewer. To switch back to the catalog view, click the ***Catalog*** *button*. Only one view can be active at any time.

To create a new symbol catalog file:

A s***ymbol catalog*** is a drawing file that only contains ***symbol definitions***. You can create a ***symbol catalog file*** by creating the ***symbol definitions*** in a drawing and saving it as an **.FS7** file. ***Symbol catalogs*** should use ***no grouping*** and contain nothing but ***symbol definitions***. A **FastCAD FS7 Catalog** file may be opened from the **OPEN** *command from the* ***Load Drawing*** *dialog box* and saved as a **"FastCAD FS7 Catalog" file** using the **SAVE AS** command from the ***Rename & Save*** *dialog box*.

To create symbols that draw in the color active during insertion:

Note that the symbols included in the sample catalog, **ELECTRIC.FS7** change color when you select a new drawing color. To create symbols that draw in the current drawing color active during insertion In your custom **.FS7** catalog file:

1. Draw the entities that will comprise the intended ***symbol definition***.
2. Use the **CHGCBB** command **[Edit > Change > to Color of SymRef]**, and select the defining entities. You can also use the **PEDIT** command **[Edit > Properties…]** and select entities one at a time. If you use this method, check the **"Color is color of Symbol Reference"** option in the ***Entity Properties*** *dialog*.
3. Immediately use the **DEFSYM** command **[Insert > Define Symbol]** to define the ***symbol***.

The entities in that ***symbol*** will be drawn in the ***current color*** at the time the ***symbol reference*** is inserted.

Default Values

For many commands, **FastCAD** displays a prompt followed by a default or prior value in brackets, like this: "**radius [1.50]:**". This is a ***suggested*** response based upon the previous value used for the same command. When you start a new drawing, **FastCAD** suggests a standard or "**normal**" response—like 0.2 units for text height—for each command that uses a default value. After you use each command, most default values change to the prior value you used. For example, each time you draw a circle, the default center is the center of the last circle you drew, making it easy to create concentric circles.

To accept the default or ***prior value***, right-click or press the SPACEBAR or ENTER key. To override the default value, input a new value using the keyboard or screen pointer.

Message Boxes

If you give **FastCAD** a command it does not understand, or if the program needs to alert you to a potential problem, a mid-screen window or message box appears containing **FastCAD**'s message. Choose the **OK** button or press ENTER to clear the message and go on with your work.

Depending upon the context of their appearance, Message boxes can also be called Alert boxes or Warning boxes.

Dynamic Cursors

For some commands that position or construct entities on the screen, **FastCAD** displays special ***dynamic cursors*** that show you exactly how the new entities will look. Dynamic cursors typically appear as moveable "**ghost**" or white outline copies of your entities.

The command line will display **"|SHIFT=move CTRL=scale CTRL+SHIFT= rotate|"** to let you know that you are in a dynamic cursor mode. Whenever you see the **"|SHIFT=move CTRL=scale CTRL+SHIFT= rotate|**:" prompt, you can use the SHIFT, CTRL, and ARROW keys as shortcuts to further modify some of these commands.

For example, draw some text by typing "**TEXT**" at the command line (from the menu, you can alternatively select **[Draw > Text > Single Line]**). Type in anything for text and it appears on the screen as you type. Before placing the textt the command line prompt reads "**|SHIFT=move CTRL=scale CTRL+SHIFT= rotate|**:". Try moving the mouse while holding down the following key combinations:

SHIFT: typically lets you dynamically **reposition** text and centers of conic entities like circles, arcs, etc.

CTRL: lets you dynamically **resize** (or scale) entities.

CTRL+SHIFT: lets you dynamically **rotate** entities.

ARROW keys: dynamically rotate at **0°**, **90°**, **180°**, and **270°**.

Selecting Entities

Whenever you invoke an editing command, you first tell **FastCAD** what you want to do, and then you pick the target objects. This second step is called entity selection.

FastCAD uses an efficient and flexible method of selecting entities. It requires considerably fewer keystrokes than entity selection methods used in previous versions of **FastCAD**, yet provides more powerful refinement. Configuration options let you determine the startup behavior.

FastCAD lets you select entities by directly picking them, or drawing a window around or through them. By holding down the **SHIFT** key, you can de-select entities using the same process.

Importantly, entities are also selectable by their entity properties. In other words, you can select entities by their color, or by their layer, or by almost any other identifiable property that you'd find in an entity's **LIST** window. This feature, and the ability to combine selections, are what make **FastCAD**'s entity selection process very powerful. For instance, you can select all red circles on the layer named "**PLAT**".

Note again that the selection process follows the command process. First you choose the command, and then you choose the items you wish to apply that command to. This verb-noun sequence works the way you would verbalize it (in English, that is). For instance, if you want to erase a line, you first choose the **ERASE** command, and then select the line you wish to erase.

The General Process

As always, watch **FastCAD**'s command prompts after you start a new command. The prompts guide you through every procedure you might encounter. After invoking an editing command, **FastCAD** displays the following prompt:

Select entities (H=help) [0 selected]:

Whenever you see this, **FastCAD** is asking which entities you want to copy, edit, move, etc. At any time during the selection process, you can press "**H**" to display the selection help screen. Or you can right-click to display the ***Entity Selection*** *popup menu*. Both the help screen and the popup menu provide options to complete, quit, or refine your selection.

Notice too that the cursor changes to a "**pick box**". To specify individual entities for selection, simply click directly on them, making sure that part of the entity is in the pick box aperture. **FastCAD** provides you with a method to change the ***aperture size*** from the **[Specs > Options]** dialog box. As entities are selected, they are redrawn in the marking color (**gray be default**). You can continue picking individual entities. The prompt displays a running count of the number of selected entities.

If you select a point that does not contain a valid entity, the cursor changes to a selection window. Like drawing a box, you can pick an opposite corner to define a selection rectangle. Any items within or crossing the selection window get selected.

You can hold down the **SHIFT** key and ***de-select entities*** by picking or drawing windows around them.

When you right-click, you can confirm, cancel, or refine the selection.

Confirming the selection

When you are done selecting entities, you must let **FastCAD** know. As soon as you do, **FastCAD** immediately continues with the requested command, affecting only those entities you have selected.

To confirm your selection, right-click and choose **[Do It]**. The right-click causes display of the ***Entity Selection*** *popup menu*, and the **[Do It]** option is located at the top of the menu. Optionally, you can also type "**D**" at the command line.

Refining your selection

This method works fine for visually picking entities on the screen. But how do you refine the selection by specifically including or excluding entities based on their properties (like **color**, **layer**, etc.)? **FastCAD** provides very powerful selection refinement options.

Entity Selection options

The Default Selection Method

FastCAD defaults to a ***popup*** *menu* selection method. In this mode, you select entities with picks or a selection window, then right-click to confirm ("**Do It**") or refine the selection via the ***popup*** *menu*. In the following ways, the ***popup*** *menu* option provides you with the most versatility:

- **Multiple Each** allows you to select entities with consecutive picks.
- It allows you to **de-select entities** if you hold down the **SHIFT** key while selecting a previously selected entity.
- If you do not pick a valid entity, then the pick cursor transforms into a ***selection window*** picking all entities that cross the window**..**
- If you toggle **OFF** the **"Enable entity-select right-button popup menus"** option in the ***Options*** *dialog box* **[Specs > Options]**, you can immediately confirm your selections with a right-click with no further refinement.
- You can refine your selections quicker by using key equivalent intercepts for ***popup*** *menu* mode (even when the menu isn't visible). These work by typing the shortcut key: **A=All**, **P=Prior**, **C=Color**, **L=Layer**, **S=Line Style**, **F=Fill Style**, **T=Entity Type**, **B=And (both)**, **O=Or, M=Dimension Style**, and **D** for **Do it** to end the selection with no menus or dialogs.

Automatically "DO IT"

You can configure **FastCAD** to eliminate the **right click/Do it** sequence altogether. In this mode, you simply pick or window the selected windows (using **SHIFT** to de-select, if necessary), then right-click to confirm selection.

To configure **FastCAD** in this manner, run the **OPTIONS** command **[Specs > Options]** and uncheck the "**Enable entity-select right button popup menu**" checkbox.

You can still access the advanced selection options by using the keyboard. For instance, if you wish to select all entities, simply press the **A** key at the "**Select entities:**" prompt, and then right-click. You can still combine selections using hot keys, and you can display the entire selection palette by pressing the H key.

Deselecting with SHIFT

If you hold down the **SHIFT** key and pick a previously selected entity, it deselects. Holding down the **SHIFT** key temporarily activates the [**Not**] mode. Deselected entities return to their normal color.

Entity Selection color

FastCAD allows you to change the entity selection (or marking) color. By default, the marking color is set to a light gray, which may be difficult to see when using similar background colors.

To change the marking color, type **MCOLOR** at the command line:

The prompt reads "**Color value [dialog]:**". To change the color:

- Type in a new color number (**0-255**);
- Pick a color directly from the Color bar, or;
- Right-click and select a color from the 256-color Select Color dialog.

The next time you use a command requiring entity selection, the selected entities will display in the new marking color.

This setting is saved in the registry, so it persists from one session to the next.

Note: This command is not on the standard menu.

Text equivalent: ***MCOLOR***

Changing pick box aperture

When you pick entities, **FastCAD** provides a pick box cursor. Any entities that are within or cross through the pick box are selected. The default size of the pick box is **4x4 pixels**. You can adjust the aperture size to suit your screen resolution, or the precision level of your work. To do so, use the **OPTIONS** command **[Specs > Options]** and specify a new in the "Pick Aperture size" window. The pick box is always square.

Advanced Entity Selection

FastCAD provides many powerful options to the entity selection process.

Whenever you invoke an editing command, **FastCAD** needs to know which entities you want to modify. You accomplish this through entity selection. In basic entity selection, you pick or draw windows around entities, and then right-click and **[Do It]**.

Review basic entity selection

However, **FastCAD** lets you refine or filter selection so that only a specified range of entities are selected, without even having to actually pick or window entities on the screen. The specified range can be very wide (e.g., all the entities) or very narrow (e.g., yellow circles on layer "Conduit"). Many of these options are based on entity properties, such as color, layer, fill style, etc. You can inspect the properties of any entity using the **LIST** command.

The advanced options are listed in the right-click ***Entity Selection*** *popup menu*. These options are explained in more detail in the ***Selection Hot Keys*** *help dialog*.

Combine selections with logical operators

You can use the logical operators **Both** (**And**), **OR**, and **NOT** to refine your selections until they include only the entities you want to edit.

The use of these operators may initially be confusing. For instance, if you want to select all green entities and all circles, you would use [**Color**] green [**Or**] [**Entity Type**] **2D** Circle. Contrary to what you might have expected, you use [**Or**] instead of [**Both (And)**]. To understand why, consider how **FastCAD** selects entities. It scans the entire drawing and looks for entities that are green or circles. If you had used [**Both (And)**] instead, then **FastCAD** would have selected only the green circles. Anything not **BOTH** green **AND** not a circle is excluded. **FastCAD** scanned the drawing, searching for entities that were **BOTH** green **AND** circles. [**Not**] works as you would expect. [**Not**] excludes any entities matching the criteria.

Mathematically speaking, logical operators tell **FastCAD** to apply two (**or more**) tests and compare the results. If you use [**Both (And)**], entities must pass all the tests to be included in the selection. If you use [**Or**], entities only need to pass one of the tests. If you use [**Not**], only entities that do not pass the test are included.

To use the Both/Or/Not operators, select one or more entities using any method, then choose [**Both (And)**], [**Or**], or [**Not**] and select using different criteria. To speed things up, you can bypass the popup menu by typing the hotkey equivalents. The hotkey shortcuts are as follows:

B Both (And)

O Or

N Not

Examples:

You might need to erase everything within a selection rectangle (**window**) except for a certain line. During the selection process, first draw the selection window, choose [**Not**], and then pick the line.

You might need to rotate all green text entities on layer "**Standard**", but not green text on other layers. Select by [**Color**] green, [**Both (And)**], and [**Entity Type**] "**2D Text**", [**Both (And)**], and [**Layer**] "Standard".

You might want to move all red entities on "**layer1**" and all text on "**layer1**" to another layer. Use the **CHANGE > LAYER** command. Then during the selection process, select by [**Color**] **red**, [**Or**], [**Entity Type**] "**2D Text**", [**Both (And)**], [**Layer**] "**layer1**".

In most cases, you could accomplish these tasks by selecting entities one by one or by repeating the editing command—the logical operators just make your work quicker and easier.

Entity Selection popup menu

During the entity selection process, you can right-click to display the ***Entity Selection popup menu***. Use this dialog to refine your selection, either by isolating selection to meet specified criteria, or combining the current selection with other criteria.

You can suppress display of the popup menu by selecting the **OPTIONS** command **[Specs > Options]** and ***unchecking*** "**Enable entity-select right button popup menu**". In this mode, a right-click acts as a **[Do It]**.

Entity Selection popup

[Do It] Ends selection and lets **FastCAD** proceed with the command, processing the entities you have selected.

[Cancel] Cancels both the selection process and the current command. **FastCAD** returns you to the "**Command**:" prompt.

[Continue] Closes the popup menu and continues pick/window selection.

[Both (And)] Restricts the selection. Selected entities pass both the first and second criteria. When you choose [**Both (And)**], **FastCAD** requires another set of selection criteria.

[Or] Adds to the selection (**default behavior**). Selected entities pass either the first or second criteria. When you choose [**Or**], **FastCAD** lets you add another set of selection criteria.

[Not] Restricts the selection. Entities selected with [**Not**] are excluded from selection.

[All] Select every selectable entity in the drawing, including those ***outside the current view***. Entities on ***frozen*** and ***hidden layers*** are not affected, unless they are part of

groups that include visible entities. ***Because groups* cannot *be restricted by layer, groups are always selected using All*. *Entities on hidden sheets are never selectable***.

[Prior] Select the entities used by the last successfully completed drawing, editing, or copying command so you can perform a series of operations on a collection of entities. Previous copy commands do not include the original entity in this selection.

[Window] Forces selection window (overrides pick), even if the first point contains a valid entity pick. Otherwise, draw a selection window as normal.

[Color] Select by primary entity color. The prompt reads "**Color value [dialog]:**". To specify the selection color: Pick a color from the ***Color*** *bar*; type a **color ID #** and press **ENTER**.Right-click to display the **256-color *Select Color*** *dialog*. Pick a color, then click **OK**.

[Layer] Select by ***layer***. Entities on frozen layers are not selected. To specify the selection layer, type a ***layer name*** or **ID #** and press **ENTER**.Right-click to display the ***Select Layer*** *dialog*. Pick a ***layer name***.

[Line Style] Select by ***line style***. To specify the selection ***line style***, type a ***line style*** name or **ID #** and press **ENTER**.Right-click to display the ***Select Line*** *dialog*. Pick a ***line style***.

[Fill Style] Select by ***fill style***. To specify the selection ***fill style***, type a ***fill style*** name or **ID #** and press **ENTER**. Right-click to display the ***Select Fill Style*** *dialog*. Pick a ***fill style***.

[Entity Type] Select by ***entity type***. **FastCAD** displays the ***Select Entity Types*** *dialog box* listing all available entity types. Click the checkbox next to ***entity type*** you want to edit and clcik **OK**.

[(Y) Font] Select text by ***font***. Only applies to text entities. To specify a selection font: Type a font name or **ID #** and press **ENTER**. Right-click to display the ***Select Font*** *dialog*. Pick a ***font***.

[Text Style] Select text by a defined ***text style***. Only applies to text entities. To specify a selection font. type a ***text style name*** or ID # and press **ENTER**. Right-click to display the ***Select Text Style*** *dialog*. Pick a ***text style***.

[Dimension Style] Select by dimension style. To specify the selection dinemsion style: Type a line style name or **ID #** and press **ENTER**. Right-click to display the ***Select Dimension Style*** *dialog*. Pick a ***dimension style***.

[Entity Tag #] Select by an entity's internally generated ***tag number***. Tag numbers are transparently assigned to new entities as they are created. Choosing by tag # is usually useful only if you have already determined the target's **tag #** using the **LIST** command **[Info > List]**. This method of selection can be helpful for choosing otherwise

difficult to select entities. For instance, this is the only way to select one line for erasure if you copied it directly onto itself.

[Entity Type #] Select by ***entity type number***. This works similarly to [Entity Type] except that the type is specified by the **type #** rather than a dialog. Each entity type (line, circle, etc.) has its own unique type #.

[(Z) Sheet] Select by ***sheet name***. To specify the ***sheet***, type in the name and press **ENTER**, or right-click and pick a name from the ***Select Sheet*** *dialog*. ***Entities on hidden sheets are never selected.***

[Help] Display the selection ***Help*** *dialog.*

Chapter 3

General Concepts

The building blocks to creating CAD drawings...

FastCAD is one of the easiest **CAD** programs you'll ever have the pleasure of using. Like a good tool, **FastCAD** is designed to help you get the job done, without forcing you to expend excess time and energy figuring out how to use it.

Admittedly, **FastCAD** is a bit more complex than a hammer or pair of pliers—maybe *quite* a bit more if you're totally new to **CAD**! But ultimately, it's more forgiving than using a pile of T-squares, triangles, protractors, pencils, and erasers, and it will likely change your whole approach to creating and modifying drawings.

Think of your drawing screen as a sheet of graph paper, with one critical exception: the **FastCAD** drawing screen is virtually unlimited in size. Until you print your drawing, you never have to worry about scale. If you want to draw a line that's two inches long, draw a two inch line. Do you want to add a three hundred foot line to the same drawing? Go ahead and add it...full size! It all fits because you continually zoom in and zoom out to navigate your drawing. You can peer into the most minute detail, then a moment later see a bird's eye view of everything.

If you can grasp that simple concept, then most of the battle is won.

In the last chapter, you learned about the specific screen components. In this chapter, the concepts are broader and usually applicable to every action you perform in **FastCAD**. For anyone who is an old hand at **CAD**, this chapter may seem rather fundamental. But if **FastCAD** represents your first experience with **CAD** software, this primer might help.

- ***Entities*** are the things you draw, such as **lines**, **circles**, **text**, etc.
- ***Units*** are the basic measuring unit in your drawing. Appropriate units should be determined and set before any entities are added to your drawing.
- ***Coordinates*** are how locations in your drawing are addressed.
- ***Bearings and angles*** can be measured and input using different conventions.
- ***Navigating your drawing screen*** using zoom commands is a vital **CAD** fundamental.
- ***Layers*** help organize your entities and what you see.
- ***Precision drawing tools*** ensure your drawings are accurate yet easy to create.

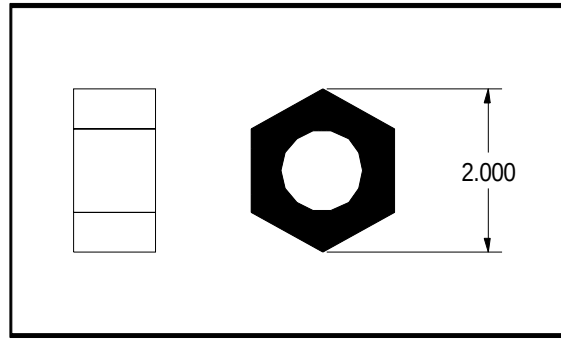
Entities

Each **FastCAD** drawing is made up of any number of ***entities***. This term refers to anything that is included in your drawing, such as **lines**, **circles**, **text**, and **dimensions**. Usually, your entites are visible, but they could be intentionally hidden from view. Using the pencil and paper analogy, an entity is anything you would draw. Each **FastCAD** drawing file is basically a sophisticated database of entities.

Just some of the entity types **FastCAD** makes available to you are:

lines	***points***	***circles***	***ellipses***
arcs	***polygons***	***text***	***multipolys***
paths	***splines***	***arrows***	***dimensions***

FastCAD allows you to draw these entities with precision and flexibility. For instance, there are six different ways to draw an arc. Don't let this overwhelm you — you may only use two methods 95% of the time, but the other methods are available should you need them.



In this simple example, there are nine entities (not counting the border):

Lines (6) make up the side view on the left.

The front view is made from a regular polygon (1) and circle (1) entity. They have been combined into *multipoly* and filled.

The dimension entity (1) makes nine.

You can combine entities together into **groups**, so they act like a **single entity**. A primary feature of **CAD** is its ability to save groups of entities for use over and over again. **FastCAD** lets you create **symbols** and **parts** for this purpose.

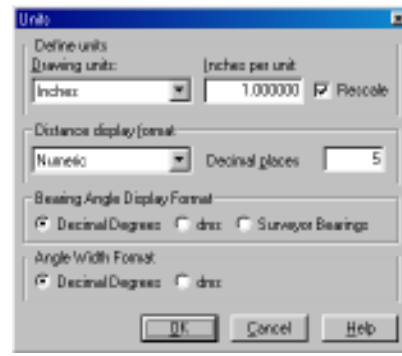
Compared to the paper method, **dimensioning** is a task made much easier by **CAD**. When you add **dimension entities** to a drawing, **FastCAD** automatically calculates the length of the selected distance and adds it to your drawing. **Dimensions** can be applied to **length**, **angle**, **radius**, and **diameter**.

You can manipulate, or **edit**, any entity. **FastCAD** provides a wide arsenal of editing commands for this purpose.

You can easily **scale** and **rotate** your drawing entities with **CAD**. Or, let's say you have drawn the perfect floor plan and want to create a **mirror image**. **FastCAD** can make use of your original image to eliminate the need to draw the same image again.

Units

FastCAD measures **coordinates** and **distances** with a user-definable system called ***units***. **Unit** is a general name for a type of measurement. **Feet**, **inches**, and **meters** are common examples of **units**, but you can create your own, like **miles**, **microns**, or **parsecs**.



Units dialog box

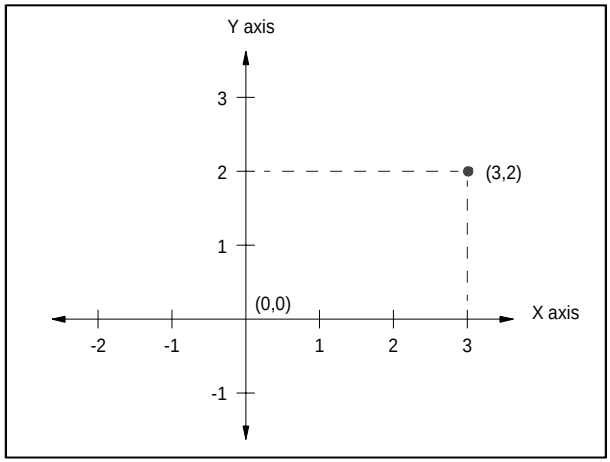
Determine suitable **units** before you start drawing entities. Think of your drawing as a life-size model. For example, if you design a bridge, you might tell **FastCAD** that each **unit is equal to one foot** before starting your drawing. For a machine tool, you might set your **units** to **inches** or **millimeters**. Each time you plot the drawing, **FastCAD** reproduces **(prints/plots)** your design at whatever **scale** you choose when you **print** it.

Dimensions are expressed in the **units** you select.

When you start a **new drawing**, **FastCAD** assumes that **one unit is equal to one inch**. You can set a different value using the **UNITS** command in the the ***Specs*** *menu*. **FastCAD** also provides a facility for file **templates**, which lets you pre-set almost all new drawing properties, including **units**.

You should always draw at **full scale**. The only time you should be concerned about **scale** is when you **PRINT** your drawing**.** You will tell **FastCAD** what distance to **print** on the paper and what distance it represents in the drawing such as **a 1/4" line printed on the paper represents or equals a 1 foot line in the drawing**. This is commonly called **quarter scale** in architecture.

Coordinates



Cartesian coordinate system

Each position in your drawing has its own unique address, composed of an ***x*** (**horizontal**) and ***y*** (**vertical**) value. To find specific locations, or coordinates, **FastCAD** measures the **horizontal** and **vertical** distances from a **common origin**. The coordinate address is written with a comma between the two numbers, like this: ***x,y***. Mathematicians call this the "**Cartesian**" **coordinate system**, after **Rene Descartes**, who developed it in the 17th century.

When you create a new drawing, coordinate (**0,0**) is located at the lower left of your screen. Coordinate (**0,0**) is called the ***origin***. Coordinate (**1,1**) is **one unit up** and to the **right** of this point, while coordinate (**-1,-1**) is **below** and **to the left** of the (**0,0**) point. In the illustration, the point is **three units** in the ***x*** direction, and **2 units** in the ***y*** direction: (**3,2**). As you add to your drawing, it can extend in all directions from the **origin**.

Entering Coordinates

When **FastCAD** asks you to select a point for a drawing or editing operation, you can do it one of three ways, mixing methods within a single command for maximum efficiency and convenience.

- ***Mouse***—Move the cursor to a screen location with the mouse, then left-click.
- ***Numeric coordinates***—You can type the exact coordinates at the keyboard, using **absolute**, **relative**, **polar**, or **relative polar** coordinates. To enter numeric coordinates, type a pair of numbers separated by a comma, with no space between the characters.
- ***Modifiers***—Select an item from the ***Mod*** *menu*, then use the cursor and left button to select an existing entity. **FastCAD** locks the cursor to a specific point (**center**, **endpoint**, etc.) on the existing entity. See the section on ***Mod*** *menu i*n Chapter 15.

Four kinds of numeric coordinates give you precise control over where an action takes place in your drawing. **FastCAD** measures absolute coordinates in units from the (**0,0**) point in your drawing, while the other three kinds are measured from the **reference point** (**the last point used in drawing an entity, or the last point selected in an editing operation**).

FastCAD can display the **current cursor position** on the ***Status*** *bar*. This function is called "**tracking**". Just click on the ***Tracking*** *window* to cycle through the four tracking modes.

1. **Absolute coordinates** are simply Cartesian coordinates, counted in **FastCAD** units from (**0,0**). Enter them by typing a pair of numbers separated by a comma (with no spaces between the characters). For example: **6,7**.

2. **Relative coordinates** tell **FastCAD** to draw from the reference point (last selected point) to a coordinate location a relative x.y distance away. Type the "**at**" **sign** (**@**), followed by the **x,y distance** to the desired point. For example, start a line at coordinates (**3,2**). Type "**@3,5**" to finish the line at (**6,7**). **FastCAD** draws the second point three units to the right and five units up from the first point. Use negative numbers to indicate movement down or to the left.

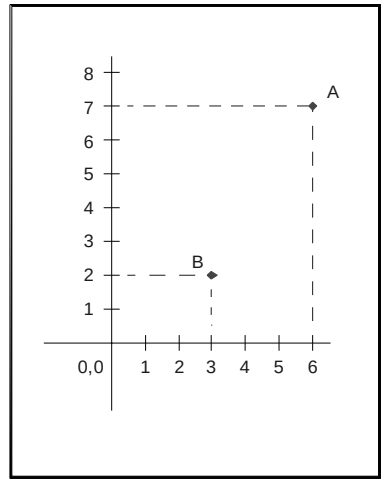
3. **Polar coordinates** use a **bearing angle** and **distance** (rather than a pair of x,y coordinates) to select a location in your drawing. To use **polar coordinates**, enter the "l**ess than**" **sign** (<), **an angle**, **a comma**, and **a distance**. For example, "**<90,2**" tells **FastCAD** to place a point **90°** (**direction**) and **two units** (**distance**) from the last selected point.

   Here's a real life example of **polar coordinates**: When you give driving directions to a friend over the telephone, you might say: "At the gas station, turn north and travel two miles." In **FastCAD**, the "gas station" is the reference point (last point used in drawing or editing), "north" is **90°**, and two miles could be two units in the polar coordinate **<90,2**.

4. **Relative polar coordinates** are like polar coordinates except that you use a **tilde** (**~**) instead of the "**less than**" **sign** (<), and **the angle** you enter **is relative to the angle defined by the last two points you used**.

   For example, suppose you just drew a line from coordinate (**1,1**) to coordinate (**1,3**) — a vertical line two units high, with an angle of **90°**. If you are still in the **LINE** command and type "**~90,1**" the resulting line has an angle of **180°** (**horizontal**) and is one unit long.

   To continue our real-life example, you might tell your friend to "turn left and go one mile." If he or she had just driven two miles north, turning left (**~90**) would mean heading west (**180°**).



Using the diagram to the left, we can see how different coordinate systems can refer to the same point.

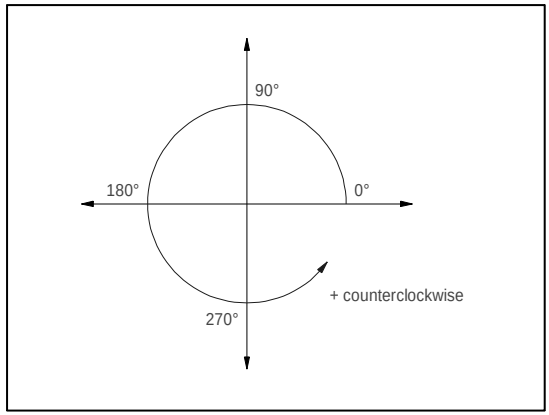
Absolute: Point A is at (6,7) and Point B is at (3,2).

Relative: Point A is at @(3,5) *relative* to Point B(3,2). To obtain absolute A, add B + @A = (3+3, 2+5) = (6,7)

Polar: The angle from (0,0) is to A 49.4°. The length is 4.61. If we were drawing a line from (0,0) to A, we'd enter the second point as <49.4,4.61.

Relative Polar: The angle from (0,0) to B is 33.7°. The angle from B to A is 59.0°. The relative bearing is 59.0-33.7 = 25.3. The distance from B to A is 2.92. So if we drew a line from (0,0) to B, and then to A, we would enter the third point (from B to A) as ~25.3,2.92.

Bearings and Angles



Measuring angles

In **FastCAD** , angles range from **0°** (z**ero degrees**) to **360°**, **measured counterclockwise from 0° at "3 o'clock"**.

You can use decimal fractions to specify angles precisely. Angles can be negative, in which case they are measured clockwise. In other words, **-45°** is the same as **315°**.

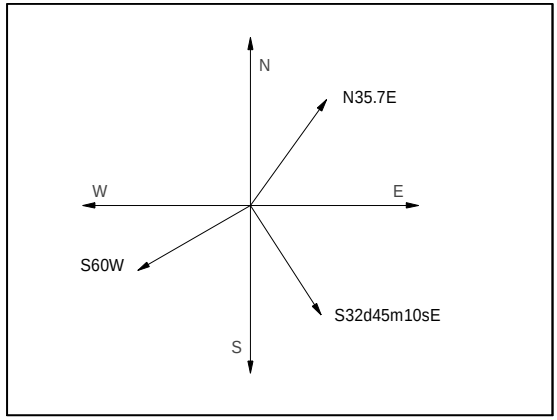
FastCAD uses this system of angles for many commands, including **ROTATE** and part/symbol insertion.

FastCAD uses this system of angles (also called bearing angles) for the **ROTATE** command, for inserting Text, Parts, and Clipboard items at an angle, for polar coordinates and for the **ANGLE TO** and **BEARING** modifiers. Use the **BEARING** command **[Info > Bearing]** to find the angle of an imaginary line between two points.

DMS and Surveyor Bearings

You can also specify **angles** or **bearings** in terms of **degrees/minutes/seconds** (**dms**). To do so, use the appropriate unit indicator after the decimal value:

<***degrees***> <**d|D|°**> <***minutes***> <**m|M|'**> <***decimal seconds***> <**s|S|"**>



Measuring dms and surveyor bearings

Examples of degree/minute/seconds.

- **10d3m2s** (**10 deg, 3 min, 2 sec**)
- **10°3'2"** (**10 deg, 3 min, 2 sec**)
- **10D3M2.75S** (**10 deg, 3 min, 2.75 sec**)

Surveyor bearings are used by architects, cartographers, etc., to express direction. Direction is referenced from the north-south compass axis (positive y-axis is "**north**"). They are expressed in dms format using a **N/S prefix** and an **E/W suffix**.

To use s**urveyor bearings**, use the same format as dms input with the appropriate compass reference:

<**N|S**><***degrees***><**d|D|ø**><***minutes***><**m|M|'**><***decimal seconds***><**s|S|"**><**E|W**>

Examples of surveyor bearings:

- **N10d4m3sE**
- **S10d4m3sW**

Entering Distance Values

Many drawing and editing commands ask you for distance values, such as a **coordinate pair, radius**, or **text height.** When you enter a measurement of distance in **FastCAD**, you can:

1. Type the measurement in **FastCAD** units, or **decimal fractions** of **units**. Some examples:
   - **Absolute coordinates** **3.5,2**
   - **Text height** **.75**
   - **Radius of a circle** **3.375**
2. Type the measurement in conventional **fractions** of **units**, using a **plus sign** (+) to separate whole numbers from fractional parts. Do not include spaces in the value or coordinate pair. Some examples:
   - **Absolute coordinates** **3+1/2,2**
   - **Text height** **3/4**
   - **Radius of a circle** **3+3/8**
3. Type the measurement in **feet-and-inches**. Use an **apostrophe** (') for **feet**, and a **quote** (") for **inches**. Do not include spaces in the value or coordinate pair. Some examples using feet and inches:
   - **Absolute coordinates** **3'6",24"**
   - **Text height** **9"**
   - **Radius of a circle** **3'4+1/2"**

   FastCAD automatically converts from **feet** and **inches** to whatever unit system you have defined. For example, if drawing **units** are defined as **FEET** and you type coordinates (**24",18"**), **FastCAD** converts it to (**2,1.5**).
4. Single values like **radius** or **offset**: Select distance visually. For example, instead of typing a value for diameter, use the cursor and left button to select a point anywhere on the screen. Now the prompt reads "**Distance to:**". Use the cursor to select another point at the desired distance from the first point. **FastCAD** measures the distance between the two points and uses that value, just as if you had typed it.

Entering Displacement Values

FastCAD often asks you to indicate movement through displacement. Commands like **MOVE** and **STRETCH** ask for "**from**:" and "**to**:" points. When prescribed, the reference manual topic for the calling command details the steps and options for entering displacement.

Navigating the Drawing Screen

Think of the computer screen as a camera view of your drawing. You can zoom in, allowing you inspect the tiniest detail. Or you can zoom out, letting you survey the entire drawing. At any zoom level, you can pan your view.

FastCAD allows two basic mechanisms for panning:

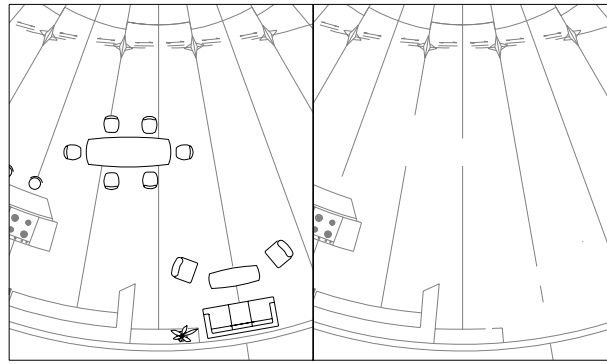
- **Dynamic panning**, which locks screen view to your mouse movements.
- **PAN** or **ZOOM CENTER** commands, which let you displace or center the view about a selected point.

Three other zoom commands will see frequent use in your drawing sessions. **ZOOM EXTENTS** zooms out to include everything visible in the drawing (**entities on hidden layers or sheets are never shown**). The extents view is a good starting point for a **ZOOM WINDOW**. The **ZOOM WINDOW** command lets you draw a zoom box on the screen; the view magnifies to that box. To quickly toggle between two views, you can use the **ZOOM LAST** command. These **zoom** commands have icons on the ***Standard*** *Icon* bar for quick access.

For ultimate view control, you can ***save named views*** and recall them. ***Named views*** can be handy when you ***print*** your drawing.

Layers

Layers are like **transparent**, **overlapping sheets** that you can use to organize your drawings. For example, you could assign one layer of a house blueprint to structural members, another to electrical work, and a third to plumbing details. **FastCAD** provides an unlimited number of layers that you can descriptively name.



This before-and-after example shows how the furniture items can be quickly hidden from view. The chairs, tables, and plants were drawn on a unique layer. When that layer is hidden, the furniture disappears from view. They still exist though, and can be made to reappear just as easily.

Multiple layers simplify editing of complex drawings. **FastCAD**'s editing commands let you select or exclude entities by layer. You can also protect layers by hiding or freezing them.

- **Hidden layers** don't show on-screen or when you plot your drawing. In the blueprint example, you could hide all but the second layer to see just the electrical work.
- **Frozen layers** are **write-protected**—you can see and refer to entities on a **frozen layer**, but you **can't erase them or modify them**.

Use the **LAYER** command to **add**, **delete**, **hide, show** or **make current** any **layer**.

Precision Drawing

While drawing with the mouse or other input device may be a good starting point, eventually you will want some shortcuts to more precise control of the distances in your drawing.

FastCAD gives you a few tools to help with precision:

Grid and Snap **FastCAD** can display a **grid** of dots to help you keep track of relative distance and size. **FastCAD** grids can be set on linear or circular patterns. Once you have set up a **grid**, you can choose to have the **cursor snap** to the nearest grid point. **Grid** and **snap** functions can be turned on or off at any point.

Switches give you **on/off** control over certain aspects of your drawings. These are usually manifested in the form of buttons, but they often can be toggled by using commands or key combinations. For instance, uou can temporarily force new lines to be **horizontal** or **vertical** by choosing the ***Ortho*** *button* on the ***Button*** *bar*, or by typing "**orthon**" or "**orthoff**" at the command prompt. Most switches can be activated in the middle of another command. For instance, you could start drawing a line, choose the ***Ortho*** *button*, and then finish drawing the line.

Modifiers are issued *within* commands to refer to existing entities, removing any guesswork or approximation. You can select the mathematically exact endpoint of a line, center of a circle, etc. Modifiers help you connect new entities to existing ones and edit with precision. Select **modifiers** from the ***Mod*** *menu*, or by using the appropriate key combination. **See *Chapter 15: The Mod menu*** for more detail.

Chapter 4

3D Concepts

The building blocks to creating 3D CAD drawings...

Viewing in three dimensions

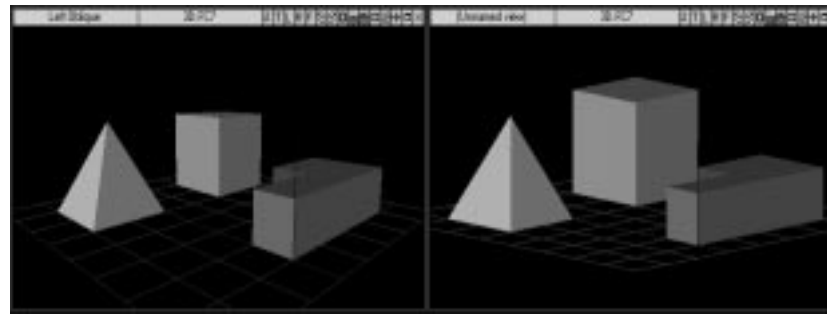
Orthographic and perspective projection

The process of representing three-dimensional objects on a two-dimensional surface (like drafting paper or a computer screen) is called **projection**. Over the centuries, draftsmen have developed several projection methods, of which **FastCAD** uses **orthographic** and **perspective projections**. Use the ***Perspective*** *icon* to toggle individual windows between the two kinds of **projection**.

Perspective drawings resemble the real works as we see it. Objects look smaller the farther they are from the viewpoint, and parallel lines seem to converge in the distance.

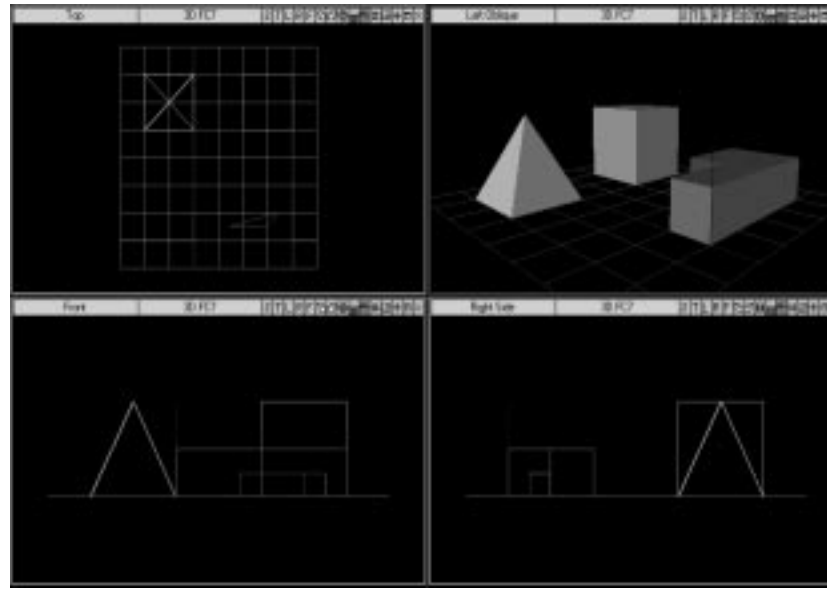
Orthographic comes from a Greek word meaning "**right-angle drawing**." In orthographic drawings, all objects are represented at the same scale no matter how far they are from the viewpoint. In other words, **FastCAD** draws a 40-foot tree on the far side of a large field the same as one that's right in front of you. These properties make **orthographic** drawings useful for accurate measurements. Plans and elevations are examples of **orthographic** drawings.

The example below shows front views of objects in both **perspective** and **orthographic projection**.



In the **perspective** at the left, you can see more of the tops and sides of the objects, giving an impression of depth. In the **orthographic** view, it isn't clear whether the objects are two- or three-dimensional. The difference is even more striking in the angled views at the bottom.

In summary, **perspective** views look more realistic and are easier to understand. Use an **orthographic** view when you plot a drawing intended for scale measurement. **Orthographic** views are easier to work with in windows that you use to select points for drawing or editing. With multiple windows, **FastCAD v7** can display both kinds of views at the same time. For drawing and editing, we recommend four windows with **orthographic** top, front, and side views, plus an angled **perspective** as seen below.



Drawing Windows

FastCAD v7 can display any practical number of views. Think of each window as a lens in a frame held in front of your face. The viewpoint is the center of the window. You can move and rotate the window with the ***3D icons***, or change the lens magnification with **Zoom In**, **Zoom Out**, and other ***Zoom*** *icons*.

Because most points that you select go on the surface of the drawing windows, it's important to know just where the window pane cuts through your **3D** model. **ZSPEC** lists this information in terms of **x,y,z coordinates** and **angles**, but a visual cue is often more useful. Here's how to get one: Click the ***Make active window*** *icon* in any window to make it active. Click the ***Zoom Window*** *icon* and place the first window corner just within the ***active window*** border. Now stretch the window cursor until it fills the ***active window***-you'll see a stretching rectangle representing the ***active window*** in all the other views where it would be visible. Right-click to cancel the zoom.

Standard Views

Although **FastCAD** can display your three-dimensional model from any angle, you can often show all the important visual information about a model with just three views: Top, Front, and Right side, especially if you add a pictorial or overall view. In engineering drafting, these are called the standard views of a model, as shown below.

FastCAD's **ZSTD** command [View > Std. 3d views] displays orthographic wireframe views of your entire model as shown on the next page. In the top view, the drawing window is level with the highest point in your model (maximum z coordinate). The front and side views line up with the farthest front (minimum y) and farthest right (maximum x) coordinates. The overall view has the viewpoint at the front upper-right corner of the model, looking toward the back lower-left corner.

In an empty model, ZSTD sets up these same views around an imaginary

Note: For consistency, the term "**top view**" **refers to the view that looks down on the XY plane**. However, there's no reason that you need to work that way. For example, you may find it more useful to think of the **ZX plane as a plan view** (**with the Y axis representing up and down**).

Selecting a view

The easiest way to select a view for the **active drawing window** is to start with one of the **standard views**, then use **FastCAD**'s ***3D icons*** to move and turn the viewpoint.

Icons that move the **active drawing window** through the model, relative to its current position. For example, the ***Zoom Up*** *icon* moves the viewpoint away from the ground if you are looking at a **side view**, or north if you are looking down on a typical map. In each case, your viewpoint moves as specified by the icon, so **objects in the window appear to move the opposite direction**.

Moving around the outside of your model usually involves a series of combined moves and turns. (For example, use the ***Zoom Up*** *icon* followed by the ***Pitch Down*** *icon* to move up and over a building.)

You can also specify a view numerically. To display a **3D** view, **FastCAD** needs to know magnification, the x,y,z coordinates of the viewpoint, and a direction of view. **ZSPEC [View > Zoom Specs]** specifies the direction with a combination of three rotations of the **XY**, **XZ**, and **YZ** planes. The ***Look From*** *icon* specifies a viewpoint and point to look towards. The ***Zoom Plane*** *icon* locks the window surface to a plane defined by three points, then asks for a direction to look.

Turning the viewpoint

The terms pitch, yaw, and roll come from the world of aviation. Like an airplane, **FastCAD**'s viewpoint can move anywhere in three-dimensional space, tilting and turning up and down, round and round, or from side to side. The terms are relative-in an airplane, "**pitch up 30°**" means lifting the plane's nose **30°** from whatever angle it's pointing now.

The **Pitch**, **Yaw**, and **Roll** icons change your direction of view without moving the viewpoint. Pitch is like tipping your head up or down, yaw is like turning your head around, and roll is like leaning your head from side to side. You can also use the text equivalent commands **PITCH**, **ROLL**, and **YAW** to turn the viewpoint by a specified angle.

The illustration shows a drawing window with viewpoint and what **FastCAD** displays n the window, before and after using the ***Pitch Down*** *icon* with a delta of **30°**.

Moving the viewpoint

Use motion icons like **Zoom Up** and **Zoom Forward** to move the viewpoint through your model. These icons don't change your direction of view. To move the viewpoint a long way in one step, use the text equivalent command for one of the motion icons (for example, **ZFWD** 100 moves the viewpoint forward 100 units). The **MOVE VIEW** command lets you move the viewpoint in three directions at once. It works like relative coordinates-you specify distances to move up, right, and forward (negative distances mean down, left, and back). The ***Zoom Center*** *icon* also moves the viewpoint.

Dynamic Panning

You can forgo the icons and text commands by using the **Dynamic Pan** function. With **DYNPAN**, you can change your view using mouse movements in conjunction with key presses. To invoke **dynamic panning**, click the wheel switch on a wheel mouse or the middle button of a three button mouse anywhere in a drawing window, or click the ***Dynamic Pan*** *icon* on the ***Standard icon*** *bar*.

During left-click and hold, drag:

Key down	*Action*
none	**Pan view**
SHIFT	**Zoom in/out**
CTRL [3D]	**Yaw & Pitch about center**
P [3D]	**Pitch about center**
Y [3D]	**Yaw about center**

R [3D]	**Roll about center**
ALT [3D]	**Move forward and back**
F [3D]	**Move forward**
B [3D]	**Move back**
H or ?	**Help**

ESC or U **Cancels this view change**

Hidden surface removal

When you start a new drawing (or use the **ZSTD** command), **FastCAD** displays **wireframe views in all windows**. Objects look like open models made of wire. You see all edges, including back edges that would be invisible in the real world. **FastCAD** can hide edges and objects that are behind opaque objects. Use the ***Hidden line on/off*** *icon* to switch between ***hidden line*** and ***wireframe*** views in individual windows.

With hidden surface removal turned on, **FastCAD** colors the surfaces of **filled 3D polygons** and **closed**, **solid** entities like **solids** ad **cylinders** with a related color for a realistic effect. You can choose hidden line removal instead, as illustrated at right above.

3D planes

Given three points in space, the flat surface that contains all three is called a plane. A plane can be tilted at any angle, and three-dimensional space contains an infinite number of planes. Because flat objects like lines and circles can lie in any plane, you will sometimes need to tell **FastCAD** what plane you are talking about.

- **You can determine a plane by indicating:**
- **Three points that aren't in a line**
- **A line and a point**
- **Two parallel or intersecting lines**
- **A point on the plane and a line perpendicular to the plane**

As an example of the latter method, you can draw a circle by selecting a center, a point on the circle, and a point on the axis (an imaginary line through the center, perpendicular to the circle). The axis determines the circle's tilt.

FastCAD often refers to the three special planes determined by the **X**, **Y**, and **Z** coordinate axes. In this manual, we'll call these the **workplanes**.

- The **XY** plane contains the **X** and **Y** axes. All points in this plane have a **z** coordinate of **0** (**zero**), so coordinates look like this: (**x,y,0**). Two-dimensional **CAD** systems use only the XY plane.
- The **YZ** plane contains the **Y** and **Z** axes. All points in this plane have an **x** coordinate of 0, so coordinates look like this: (**0,y,z**).
- The **ZX** plane contains the **X** and **Z** axes. All points in this plane have **a y** coordinate of **0**, so coordinates look like this: (**x,0,z**).

3D angles

Two-dimensional **CAD** systems measure angles around a center point, with positive angles tuning counterclockwise from zero degrees at 3 o'clock. In a three-dimensional model, angles are measured around a centerline called an axis.

Whether a positive turns counterclockwise or clockwise depends on the direction you are looking. For example, when you look at a clock from the outside, its hands travel clockwise. If you could stand inside the clock and look out through its face, you would see the hands travel counterclockwise. **FastCAD 3D** resolves this conflict by asking you to specify a center and axis endpoint for rotations. Positive angles turn counterclockwise if you look along the axis from the center toward the endpoint.

The zero-degree point for a **3D** angle depends on the command you are using. For example, you can draw a **3D** arc by selecting its center, starting point, and ending angle. The arc's zero degree point is simply the starting point that you picked.

Some features in **FastCAD 3D** use only **2D** angles. For example, the **INSERT PART** command allows **rotation only in the XY plane**. (Of course, you can rotate the parts in any plane after you have inserted them.) For such commands, angles behave exactly like angles in **2D FastCAD**-zero degrees is toward the **positive X axis**, and **rotation is counterclockwise** if you look down on the **XY plane**.

Entering 3D coordinates

Because your computer's screen is a two-dimensional surface, entering three-dimensional coordinates presents a challenge. **FastCAD** offers several ways to add the third dimension. You can place points on the window surface, or use depth mode to add elevation values.

On the Window Surface

When you select points with the left-click in a drawing window, FastCAD assumes you want points on the surface of the drawing window. You can l9ock the window surface to a specific plane with the ***Zoom Plane*** *icon*, then use the ***Zoom Forward*** and ***Zoom Back*** *icons* to move the window above or below the original plane. **Note:** If snap locking is turned on, **FastCAD** locks to the snap point nearest the window surface.

Depth mode

To place points above or below the window surface, use **FastCAD**'s ***Depth*** *button*. For each point you select, **FastCAD** gets two dimensions from the cursor position on the drawing window's surface, then adds an extra "**depth**" prompt and displays a rubber-band cursor. You can then raise or lower the point by moving the cursor to another window.

3D numeric coordinates

You can select points in **3D** space by typing numeric **x,y,z** coordinates at the keyboard, using **absolute** or **relative** coordinates. Type three values separated by commas, with no spaces, like this: **5,0,3.142** or this: **@0',24',6'6"**. You can also type just the **x,y** coordinates, leaving out the **z** coordinate. **FastCAD** assumes a **z** value of **0** (**zero**)-to enter (**1,1,0**) you can simply type **1,1** and press **ENTER**.

Note: In **FastCAD**, polar coordinates work with regard to the **XY** plane. For example, if the last point you selected was (**0,2,1**) typing **<45,1.41421** selects coordinates (**1,3,1**)-there's no change to the **z** coordinate. **Relative polar** coordinates (using **~**) have no meaning in **3D** context.

FastCAD can display the current cursor position on the ***Tracking*** *status* in terms of **absolute**, **relative**, or **polar** coordinates. **Polar tracking** displays an angle and distance that are parallel to the **XY** plane.

3D entities

3D entities are the primitive geometric elements from which you build a drawing or model. Location information for **3D** entities is stored in the form of **x,y,z** coordinates, so the entities can exist anywhere in three-dimensional space. Some **3D** entities, like lines, points, circles, and text, are flat. These objects act like the entities in **2D** space, although you can tilt them at any angle. You can build a **3D** model from these flat entities, but **FastCAD** offers a better way.

Surfaces and Solids

The most powerful **3D** entities are surfaces and solids. These entities, found for the most part in the ***Draw3*** *menu*, occupy space and give shape to your model. It's like the difference between building a model by taping together pieces of paper or by sculpting it with clay. These entities include:

- **Cylindrical shapes, including cones, tubes, and tapered cylinders.**
- **Tabulated profile-a series of edges connected with closed sides-for example, a curved tube or duct with bends.**
- **Rectangular solids**
- **Irregular solids**
- **Surfaces of revolution**
- **Meshes, ruled surfaces, and Coons surfaces**
- **Spheres, domes, and spherical segments.**

Note: **FastCAD**'s solids look hollow if you view them from the inside. However, they are single objects mathematically, solid in the sense that they are closed and occupy space.

Edges

Edges include entities like **2D** and **3D** lines, circles, and splines. Edges are tools **FastCAD** uses to build many of its surfaces and solids. For example, you create a ruled surface by selecting two entities to become its edges. **FastCAD** stretches a surface mesh of straight lines (**called rules**) between the edges. A tabulated profile extends a surface in the shape of a single edge. When **FastCAD** asks you to select an edge, you can pick a **2D** of **3D line**, **arc**, **circle**, **path**, **polygon**, **smooth polygon**, or **spline**.

2D entities in 3D space

You can also use regular **2D** entities like lines, circles, and arcs in **3D** drawing. These **2D** entities are flat (have no z coordinate), so they are limited to the **XY** plane. **FastCAD** processes **2D** entities faster because each point only has two coordinates instead of three. Note that **FastCAD** has several commands to convert **2D** entities into **3D** entities.

3D drawing and editing

3D Drawing Strategies

FastCAD has several commands that create 3D entities from scratch. For example, **SOLID**, **CYLINDER**, **SPHERE**, and **MESH SURF** create three-dimensional objects without referring to existing entities for their shape.

However, it's often easier to build a three-dimensional shape by drawing a flat entity, then extruding it or using it as an edge for a solid or surface. **EXTRUDE** and similar commands can start with **2D** or **3D** entities. If the edge you want to draw lies in the **XY** plane, start with a **2D** entity-it's less work because you don't have to specify a depth for each point.

FastCAD includes several commands that turn existing **2D** or flat **3D** entities into three-dimensional objects:

EXTRUDE adds depth to flat entities like lines, polygons, and circles, creating simple three-dimensional shapes. For example, **lines** become **polygons**, **polygons** become **solids**, and **circles** become **cylinders**. Extrusion works on both **2D** and **3D** versions of flat entities, and you can extrude in any direction.

SURF REV, **TAB PROFILE**, **RULED SURF**, and **COONS SURF** build solids or **3D** surfaces from edge entities that you select. For example, **SURF REV** revolves the edge, turning it into a surface of revolution. If the edge is a circle, the resulting solid looks like a **torus** (**donut**); if the edge is an angled line, the result may look like a cone or lampshade.

Selecting entities

You will often need to select entities for the commands discussed on the previous page, or for editing commands. If your model is complex, it's important to use multiple views when selecting so you can see exactly which entities will be affected by the command you're using.

When you select by **EACH**, **FastCAD** finds and highlights entities along an imaginary line extending into the model perpendicular to the window surface. Note: Picking by Each is easiest in orthographic rather than perspective views.

When you select by Window, you draw a selection block around the entities you want. You can select corner points in two windows, or use depth mode to draw the block. Note: If you draw a **2D** selection rectangle on the window surface (both corners in the same window or at the same depth), **FastCAD** only highlights entities that are on the window surface.

Editing commands

Most **Edit** and **Copy** commands work as they do in **2D** drawing except that you can

select points in **3D** space. Some commands may ask for additional information, particularly if **Depth mode** is **on**.

Displacement

When **FastCAD** asks you to select **3D** "**from**" and "**to**" points for commands like **MOVE** and **EXTRUDE**, **FastCAD** calculates the **x,y,z** distance, or displacement, between the points you select with the cursor.

Using workplanes

Std. 3d Views

Zoom 3d Workplane

A ***workplane*** is an invisible ***flat surface*** on which **FastCAD** projects **3d**entities. It's like the **XY plane,** but can be placed at any angle, anywhere in **3d** space. Like the **XY plane**, each ***workplane*** extends for an unlimited distance in all directions within a flat plane. A single drawing can contain many workplanes, each with a name to distinguish it from others.

When you draw **3d** entities, **FastCAD** locks them to the currently selected ***workplane***. This is the last ***workplane*** you ***defined***, unless you have selected a different one from the ***Workplane status drop-down menu*** .or you have just loaded a different drawing.

To reduce visual clutter, **FastCAD** doesn't display or plot entities that would be perpendicular to the ***current view***. For example, your model can contain entities on ***workplanes*** lined up with ***top***, ***front*** and ***side views***. In an ***overall view***, you'll see all the ***dimensions***. However, in the ***front view***, entities from the ***top*** and ***side views*** would be ***edge-on to the screen***, so **FastCAD** doesn't display them. **FastCAD** also ***hides*** entities, when ***Hidden line*** is toggled ***On for that view,*** on ***workplanes*** seen from behind - for example, entities in a ***right side view*** won't show in a ***left side view***.unless ***Hidden line*** is toggled ***Off*** for that view window To ***hide*** entities from perspectives and other angled views, you can keep them on ***separate layers*** from the rest of your model, then ***hide*** the ***layers***.

Define a Workplane by picking three points in 3d space

You set up a ***workplane*** by specifying its ***origin***, a "horizontal" direction for text and arrows in horizontal dimensions), a "**Point on workplane positive X axis**" to the right of the ***origin*** and a "**Point on workplane positive Y axis**" up from the **origin.** The easiest way to define a workplane is on the surface of, or parallel to, a drawing window. The **DEFINE WORKPLANE** command has defaults that make this very easy to do, as described below.

Before ***defining*** the ***workplane***, use **STD VIEWS** icon or the ***3D zoom*** *icons* to set up one of the view windows you want to use to draw in. For the illustrated example, you could select **STD VIEWS**, then use the lower-left window (**a front view**) to place points.

When you select **DEFINE WORKPLANE** the prompt reads "**New workplane name [none]:**". Type a ***descriptive name*** (it can include spaces and other punctuation) the press **ENTER**. The name should help you remember from which direction you intend to view the **3d** dimensions on this ***workplane***. For example, a ***workplane*** intended for drawing a ***plan*** could be named "**top view**." An ***elevation*** could be "**front**" or "**southwest**." or even "**Elivaation".**

The next prompt reads "**Workplane origin:**". This point will be a ***local origin*** (**0,0**) for entities in the ***worplane***. Select a point on an entity on the desired ***workplane***.

Now the prompt reads "**Point on workplane positive X axis:**". The default direction means that ***dimension text will read horizontally to the right*** in the window you used to place the ***origin***. Select a point to the right of the ***origin*** you picked..

The next prompt reads "**Point on workplane positive Y axis:**". Pick this point up vertically from the ***workplane origin*** you picked.

Drawing 2d entities on a 3d Workplane

Use the ***Zoom 3d Workplane icon*** *t*o immediately zoom the ***active view window to the surface of the*** workplane. You may now draw on the surface of the ***workplane*** and even use **2d entities** and then use the **EXTRUDE** command to give them depth**.** This method makes **FastCAD 3d** much easier for rawing in **3d** space. You can treat each ***workplane*** surface as if it is a **2d** drawing.

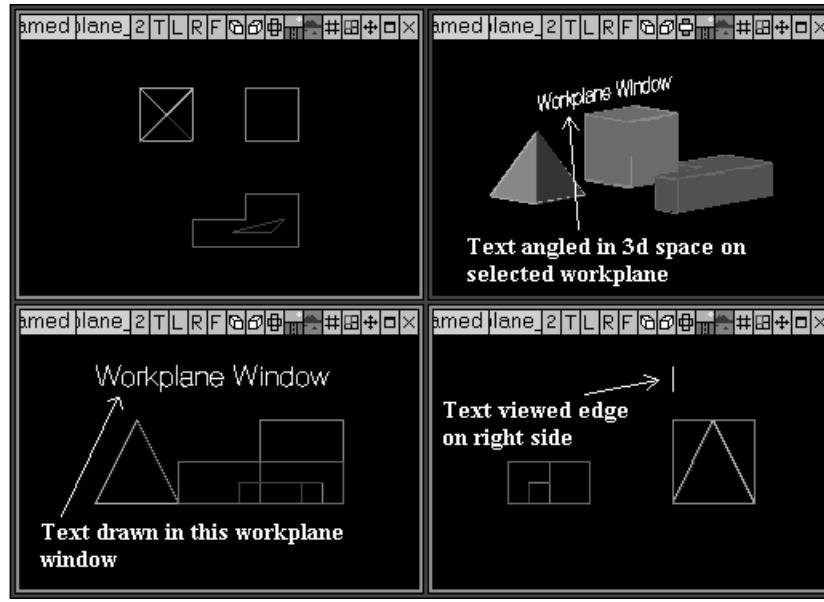
Right-click to accept the default, or select a point to orient the workplane toward some other direction.

FastCAD adds the new ***workplane*** definition to your model, making it the current workplace for new **3D dimensions**.

You can select a defined ***workplane*** by clicking the ***Workplane status window*** drop-arrow and click on one from the drop-down menu. It becomes the default ***workplane.***.

Define a Workplane by picking just the origin in a view window

The **DEFINE WPLANE WIN** (**Define Workplane Window**) allows you to select a ***workplane*** by picking the ***workplane origin point***i by clicking in any view window and that window becomes the ***active workplane*** surface. The lower-left window in the illustration below was defined as the ***workplane.*** Notice the "**Workplane Window"** thex was drawn on the surface of the lower-left window and is angled in the other two**..**



Depth mode

Depth button

When **depth mode** is on, **FastCAD** displays a default depth (the depth you select if you right-click or press **ENTER**). In addition to picking a point to indicate ***depth***, you can press **ENTER** to accept the default, or type a distance value. A positive value moves the ***depth*** forward (into the screen) from the first point you digitized. **FastCAD** uses an extra "**Depth**" prompt for each **3D** point you select. You can use another window to specify a depth or height for the point. In contrast, with depth mode off, points go on the surface of the drawing window in which you select them.

When **FastCAD** prompts you for a 3D point, pick a point in any window. The extra prompt reads "**Depth from window pt/value [prior]:**". Move the cursor into a different window and use that window to move the point to the depth or height you want (**FastCAD** displays a rubber-band cursor to show the effect of the depth).

Depth mode is easiest to understand in terms of two windows: a top view (looking straight down on the **XY** plane) and a side view. When **FastCAD** prompts you for a **3D** point, you can specify **x** and **y** coordinates in the top view but use the side view to specify the **z** coordinate. With other views, just remember that depth is measured perpendicular to the original window.

Here's a practical example: Suppose you display a one-story house in two windows-a top view and a front view. The top view looks straight down on the **XY** plane from ceiling eight, with north at the top of the window. The front view looks at the south side of the house from eye level. Adding furniture symbols to the house is easiest in plan view, but without depth mode you would be placing them on the ceiling! With depth mode on, you can specify **x,y** coordinates for each symbol in the plan view, then specify the z coordinate (distance from the floor) in the front view of the house.

Depth defaults: Instead of picking a point to indicate **depth**, you can right-click to accept the default depth. When you start a new command, the default is on the window surface. For subsequent points that display a rubber-band cursor, the default is at the same depth as the last point you picked.

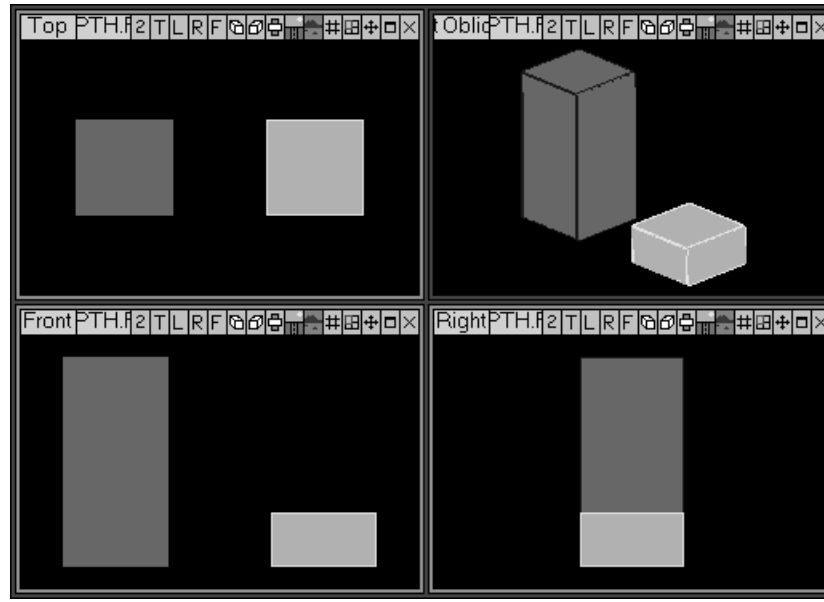
Note: Depth mode is only useful if you have more than one drawing window open. You may find it easier to use with ***perspective turned off***.

For a hands on **Depth Mode** tutorial, see the **FastCAD Help** file topic **"3d CAD Concepts > Working in 3d > Depth Mode Tutorial".** You will be working with the **FastCAD** drawing named **DEPTH.FC7 f**or the tutorial**.**

IMPORTANT: It is essential that you become proficient at using Depth Mode as well as Defining and useing Workplanes for drawing in 3d space.

Depth mode Tutorial

FastCAD's *Depth Button* helps you place points anywhere in three-dimensional space. For each point that you select, **FastCAD** gets two dimensions from the cursor position on the drawing window surface, then adds an extra "**depth**" prompt that lets you ***raise*** or ***lower*** the point in another view window. In this example, we'll use ***depth mode*** to draw a tilted rectangle connecting two blocks of different heights. The illustration below is a **FastCAD** sample drawing named **DEPTH.FC7** that you may open to performe the steps in this tutorial. This is the same tutorial contained inthe **FastCAD Help** file.



1. From the **FILE** menu select **OPEN** and click the **DWGS** folder and **open DEPTH.FC7** drawing. As seen in the illustration above, there are **four view windows**, **TOP view** [**upper-left**], **FRONT view** [**lower-left**], **RIGHT SIDE view** [**lower-right**], and **LEFT OBLIQUE view** [**upper-right**].

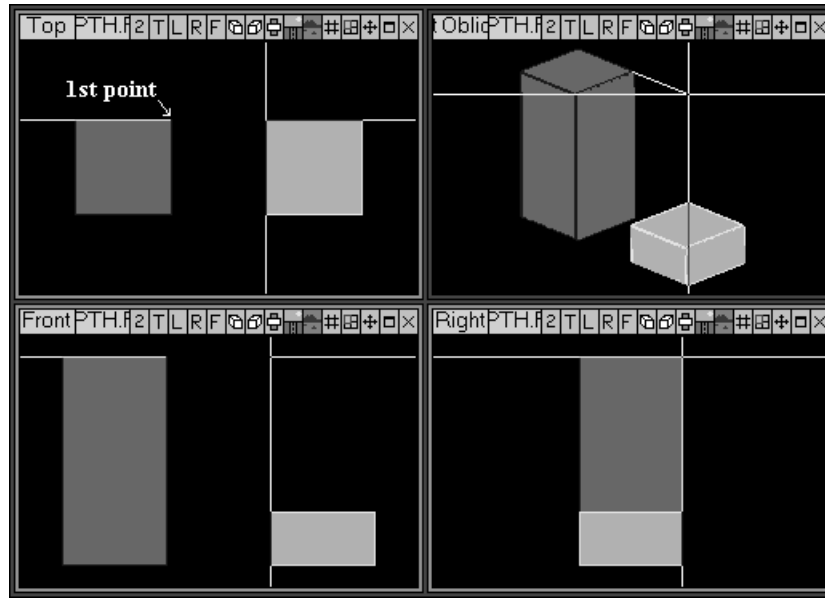
   There are two blocks: a **blue one 4" high and a yellow one 1" high**. The ***Depth mode button*** is turned **Off** to start with.

   The ***Snap*** and ***CsrSnap buttons*** are ***depressed*** or **On**. With these enabled, it is easier to see the **X,Y,X *coordinates*** update in the ***Tracking window*** as you move the cursor.

   Snap and ***CsrSnap*** cause the ***Tracking window*** to show each ***snap*** as you move the cursor. ***Snap divisions*** are set to **4 divisions per grid** so ***Tracking*** only updates every **quarter of a unit**.

You will now use the **upper-left TOP *view window* to pick 3d points over the XY *plane*** and the **lower-left FRONT *view window*** to pick the **Depth or Z height** for each point ***above*** the **XY *plane*.** The **upper-right LEFT OBLIQUE *view window*** will show each side of the ***polygon*** as it is drawn. You will alos see the cursors displayed in this window as you move and pick points. This may be a bit confusing at first but the more you work in **3d**, the cursors position will start to make sense.

Picking the 1st point of the Polygon



2. Select the **3DPOLY** command from the **DRAW3** menu**.**

The prompt reads 1st point:

Move the crosshairs around in the **upper-left TOP *view***, watching the ***Tracking status***. The **X** and **Y coordinates change** but the **Z coordinate is fixed at 4.0**. That's because this **window's surface is 4 units above the XY plane on top of the blue block**.

Left-click to place the **1st point at coordinates (2,2,4), at the upper-right corner of the blue block then move the cursor to the right.** The rubber-band cursor extends straight out from the **blue block at a height of 4 units**.

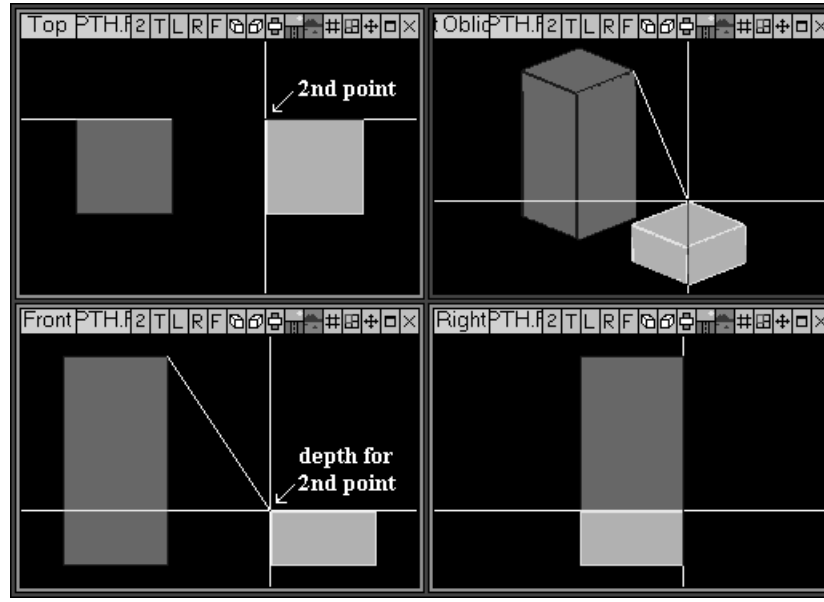
3. ***The prompt reads*** Next point [done]:

You need to place this **next point *below* the TOP *view window surface at the level of the top of the yellow block***. Here's where ***depth* mode** comes in.

With the rubber-band cursor still attached, drop down and **left-click** the ***Depth button*** on the ***button bar***.as illustrated below. ***Depth mode*** is now enabled.

The ***buttons*** appear ***depressed*** to show that they are switched **On** as illustrated above. The ***Grid, CsrSnap*** and ***Snap*** make it easier to see the ***Tracking indicator*** update the **XYZ *coordinates*** as you move the cursor**.** The ***coordinates* w**ill only change each **.25 of a drawing unit** because the ***snap divisions*** are set to **4 per unit..** When ***Depth mode*** is enabled the visual ***snap*** of the cursor is suppressed but ***snap*** is still in effect.

Picking the 2nd point and its Depth point above the XY plane



4. In the **upper-left TOP *view window*** Left-click at ***coordinates* (4,2,4),** to place the **1st point** at the **upper-left corner of the yellow block**.

The prompt reads 

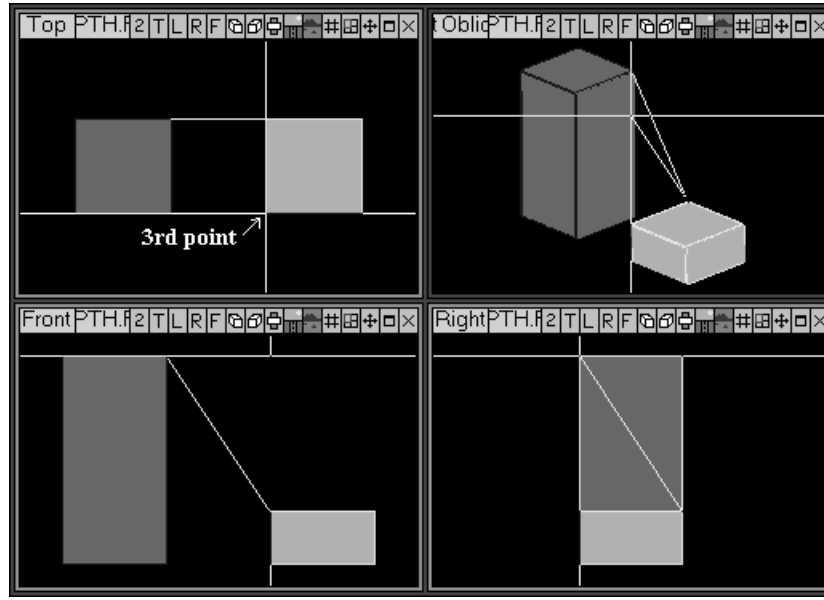
FastCAD took ***X*** and ***Y coordinates*** from the point you just picked over the **XY *plane***, but you still need to indicate a ***Z coordinate depth point above*** or over the **XY plane**.

Move the cursor down into **the lower-left FRONT *view window***. Notice that you can move the rubber-band cursor ***up and down***, but not ***from side to side***. That's because the **X** and **Y *coordinates*** for this point ***are already set***.

Left-click when the rubber-band cursor touches the **upper-left corner of the yellow block** at ***coordinates* (4,2,1)**. You will see the ***first green edge*** of the **3d *polygon*** in the **upper-right LEFT OBLIQUE *view window*** .

This **upper-right LEFT OBLIQUE** ***view window*** gives you a good visual indication of how things really look as you draw each edge of the **3d** ***polygon***.

Picking the 3rd point and its Depth point



The prompt reads Next point [done]:

5. For the **third point**, move the cursor up into the **upper-left TOP** ***view window*** and left-click **at** ***coordinate*** **(4,0,4)**, at the **lower-left corner of the yellow block**.

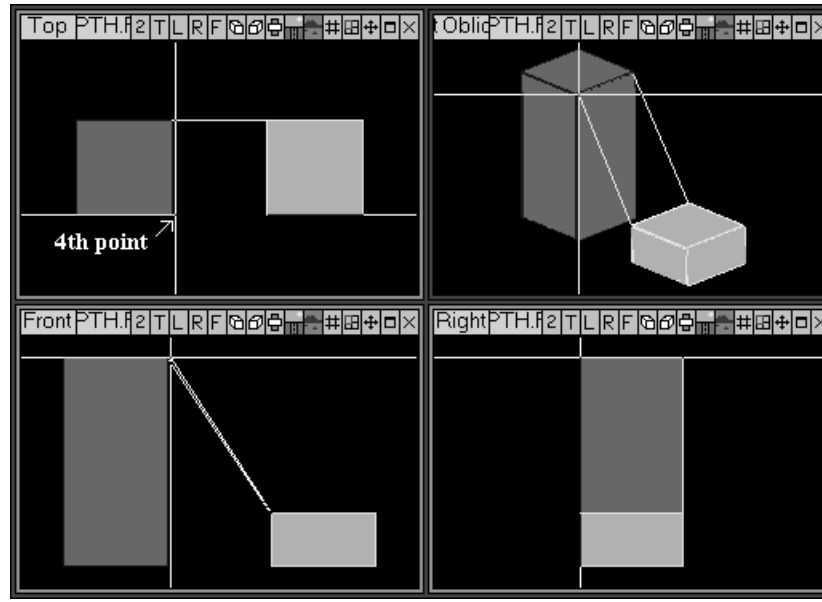
   The prompt reads Depth from window pt/value [prior]:

   When you use ***depth*** **mode** with entities like **polygons** that use the rubber-band cursor to draw a series of points, **FastCAD** stores the ***depth of each point for you to use on the next point***. We want this ***depth point at the same level as the second point***, so we'll use **FastCAD**'s ***default depth***.

   Right-click to accept the default ***depth*** **point** (**prior**) **Z coordinate of 1 unit**.

   The prompt reads Next point [done]:

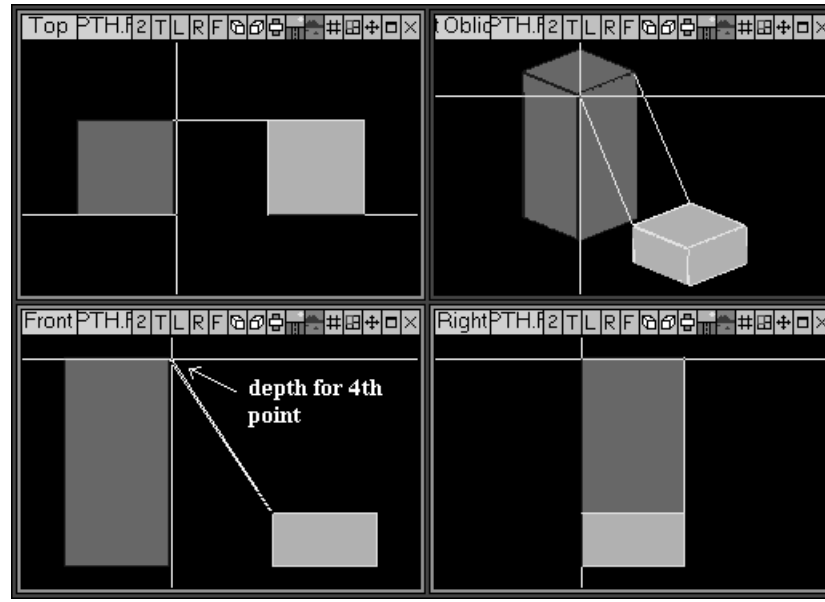
Picking the 4th point



6. For the **fourth point**, left-click at ***coordinate* (2,0,4)** in the **upper-left TOP *view window***, which is the **lower-right corner of the blue block**.

   The prompt reads Depth from window pt/value [prior]:

Picking the 4th point's Depth point

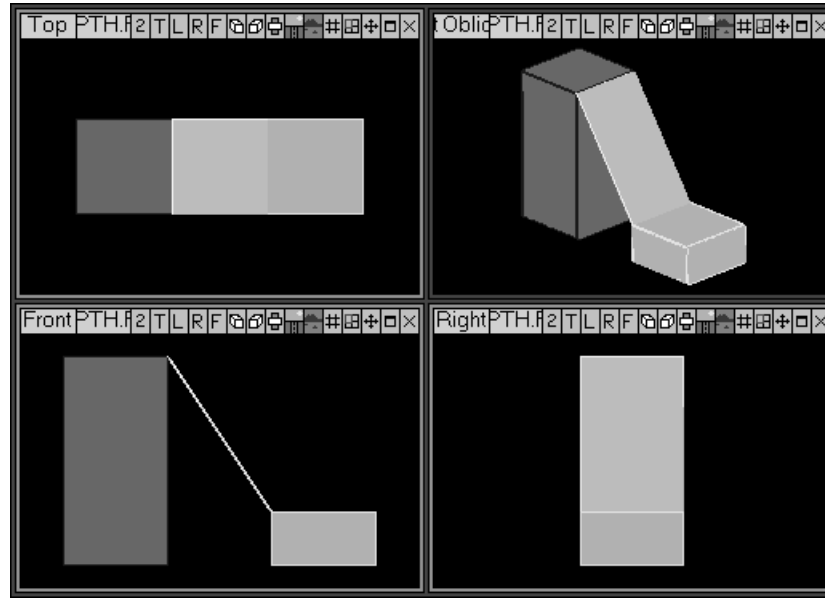


7. Drop down in the **lower-left FRONT** ***view window*** and left-click at ***coordinate* (2,0,4),** which is the **upper-right corner of the blue block,** for its ***depth point***.

   The prompt reads Next point [done]:

8. **Right-click** to **close the last edge and draw the 3d polygon**

FastCAD Draws the 3d Solid Filled Green Polygon



FastCAD draws a **green Solid Filled polygon** as illustrated above. It **filled automatically** because the **default** ***Fill Style*** in the drawing is set to "**Solid**". If the ***default Fill*** **Style** is set to "**Hollow**", you would see through the ***polygon***.

It takes some practice to become proficient at using ***Depth mode*** but this is a unique **FastCAD** tool to make drawing in real 3d space easier. This together with ***DefiningWorkplanes*** and ***Defining Workplane Windows*** and using them as drawing surfaces is essential for making **3d** drawings in **FastCAD.**

Chapter 5

File Commands

Managing your drawing files

The **File** menu includes general commands for managing drawings and other files.

Use the **OPEN** command to convert single files "o**n the fly**" from other formats into **FastCAD**. You can import **AutoCAD DXF** or native **DWG** drawing files, as well as **FCW** drawing files created in **FastCAD/EasyCAD 32 v5.215**, and **FCD FastCAD/ EasyCAD DOS** drawing files.

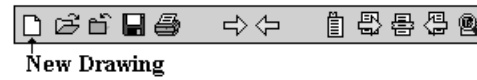
You can use the **SAVE AS** command to save your current drawing to any supported format.

If you have many files to import/export, use the **CONVERT** command to convert entire directories from one file format to another.

FastCAD v7 has multiple file capability. You can open multiple files independently of each other, or as *overlays* or *references*, so that they share the same coordinate space.

In addition to file commands, the File menu also displays the File History, which displays the last four files you have saved. Select any file on the list to load it into your current **FastCAD** session.

New Drawing



The* NEW DRAWING *command* [File > New Drawing] *closes the currently loaded drawing, and opens a new, blank drawing named* WORK.FC7.** If the drawing to be ***closed has been modified, **FastCAD** asks (**with an alert box**) if you want to save the current drawing before creating the new one. All drawing properties (like text and dimension specifications) are reset to that of the ***current template***.

Use the **SAVEAS** command **[File > Save As]** to give the new "**WORK.FC7**" drawing a unique name.

Text equivalent: ***NEW***

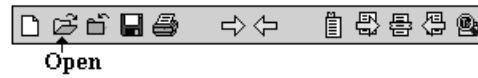
New 3d Drawing

The* NEW 3d DRAWING *command* [File > New 3d Drawing] *closes the currently loaded drawing, and opens a new, blank drawing named* WORK.FC7 and issues the STD. 3DD VIEWS command to display the four view window screen for TOP, FRONT, RIGHT SIDE, and LEFT OBLIQUE views for starting to draw in 3d.** If the drawing to be ***closed has been modified, **FastCAD** asks (**with an alert box**) if you want to save the current drawing before creating the new one. All drawing properties (like text and dimension specifications) are reset to that of the ***current template***.

Use the **SAVEAS** command **[File > Save As]** to give the new "**WORK.FC7**" drawing a unique name.

Text equivalent: ***NEW;ZSTD;***

Open



The* OPEN *command* [File > Open] *loads a previously saved drawing*. FastCAD** displays the standard ***Load Drawing *dialog box*, which lists available drawings of the file type you specify. To load a file, select a drawing filename from the list and click **OK**. If you type a name not listed in the dialog, a new drawing will be created with the specified name.

Use **OPEN** to perform "**on the fly**" file conversions from other formats to the native **FastCAD FC7** file format. When a non-fcw file is converted, a new fcw file is created, and the original file remains unmodified. The converted file has the same file name, but takes on the .fcw extension.

FastCAD v7 can import the following file types:

- **FCD FastCAD/DOS 2.72+ files.**
- **FCW FastCAD 32/EasyCAD 32 files.**
- **FSC FastCAD 32 Symbol Catalog files**.
- **FCT FastCAD 32 Template files**.
- **DXF AutoCAD Drawing Exchange files up to version 2000**
- **DWG AutoCAD Drawing files, up to version 2000.**

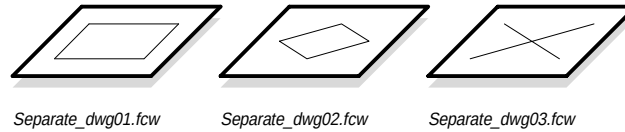
To specify a file type to import, click the Files of Type box and select a type from the drop-list. Specify the drive and path as needed. **FastCAD** filters the list window to display only those filenames that end in the matching extension. Select a name and click **OK**. **FastCAD** performs "**on the fly**" conversion into the **FC7** format and loads it into your current session.

Text equivalent: ***OPEN*** *or* ***LOAD***

Keyboard Shortcut: ***CTRL+O***

Multiple Drawing commands

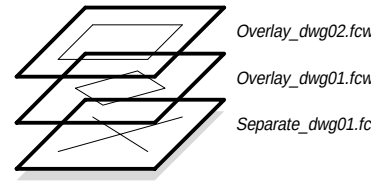
Separate Drawings



Separate drawings are all loaded into memory, but they remain independent.

Overlay Drawings

Overlay drawings share the same coordinate space as the first separate file.



FastCAD allows you to open more than one file at a time. Additional drawing files can be opened in two ways:

1. As **separate** drawings

   Files opened in this manner are independent of each other. For instance, a single line cannot be drawn from one drawing to another, nor can an entity be dragged from one drawing to another. The clipboard must be used to directly transfer entities between ***separate*** drawings.

2. As **overlay** (or **reference**) drawings

   After a file has been loaded using the normal **OPEN** command, ***overlays*** or ***reference*** drawings can be loaded atop it. ***Overlays*** are completely ***separate*** drawings, and always maintain their ***separate*** nature. When an ***overlay*** is opened atop a ***separate*** file, it never integrates itself with another drawing. Rather, it shares the same coordinate space. ***Overlay*** drawings can be opened and edited independently. A ***reference file*** is simply an ***overlay*** that is **read-only**. A **read-only** file is one that cannot be edited.

   Note that ***overlay/reference*** drawings can only be loaded on the drawing loaded with the **OPEN** or **NEW** command. Successive drawings loaded with the **OPEN SEPARATE** command ***cannot*** **possess** ***overlays*** for this session.

To understand ***overlays*** better, it may help to compare them with parts, their most similar traditional **FastCAD** feature. Both ***overlays*** and ***parts*** are maintained as ***separate*** regular drawing files. But ***parts*** differ from ***overlays*** in the following key aspects:

- Once a ***part*** is inserted into a target drawing, it becomes fully integrated into the target drawing. Every time the target drawing is loaded in the future, the ***parts*** contained therein also load. ***Overlays***, on the other hand, remain conspicuously discrete. ***Overlays*** must be singly loaded every time the primary file is newly loaded. ***Overlays*** never auto-load, and they share no natural association with any other file (unless they contain **XREF**s).
- Since an inserted ***part's*** elements are integrated into the target drawing, subsequent editing of the inserted ***part*** does not affect the original ***part file*** in any way. ***Overlays***, however, *are* the source file. If you edit an ***overlay*** while it is loaded, and then save it, the ***overlay file*** is modified to reflect those changes.
- Conversely, subsequent editing of the source ***part file*** will not affect ***parts*** that have already been inserted. There is no dynamic association between the source ***part file*** and ***inserted parts***. ***Overlays*** are always displayed in their current state.

As you can see, ***parts*** and ***overlays*** share some similar qualities. They are both ***separate*** drawing files that can be "**combined**" with others. But, functionally, they are completely different.

Parts, as their name suggests, are used to make drawing entities portable for use in other drawings.

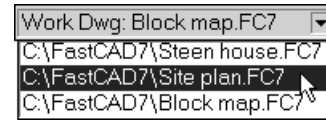
Overlays serve more of an administrative purpose. Use ***overlays*** to logically separate an otherwise large drawing into smaller, more manageable drawing tasks. An example might be that of a facility engineer who controls the master plant layouts, but allows sectional ***overlays*** to be modified by personnel relevant only to those areas. Or perhaps a single drawing is so large that it affects your computer system's performance. Separating the drawing into smaller, more efficient ***overlays*** will facilitate editing, while allowing all elements to be easily recombined when needed.

Open Separate File

***Use the* OPEN SEPARATE *command if you already have at least one file open, and you wish to load one or more drawings*.**

To select the **OPEN SEPARATE** command **[File > Open Separate…]** from the **FILE** menu. **FastCAD** displays the standard ***Load Drawing*** *dialog*. Choose a file as you would using the regular **OPEN** command. **FastCAD** loads the ***separate file*** as a full screen window over the current drawing. This drawing can be resized at any time using the standard Windows methods.

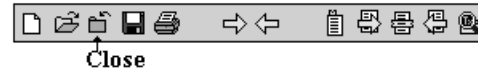
Separate files are listed in the ***Work Dwg:*** *status drop-down menu.* Click the drop-arrow at the right end of the status window and select a drawing. It now becomes the ***active drawing*** to which all new entities will be added.



Active Work Drawing status drop-down menu

You may use this method to *copy* selected entities from the ***active drawing*** to the Windows clipboard using the **CLIP > COPY** command **[Clip > Copy]**. Select another ***open*** drawing from the ***Work Dwg:*** *status drop-down menu* to be ***active*** and use the **CLIP > PASTE** command **[Clip > Paste]** to ***paste*** the enties into.

To ***close*** a ***separate* file**, you can select the desired file from the ***Work Dwg*** *status drop-down menu*, use the **CLOSE** command **[File > Close]** or you can click the **CLOSE *icon*** on the drawing window's ***Title*** *bar* as seen below.



How does **OPEN SEPARATE** differ from the regular **OPEN** command?

When a file is loaded using the regular **OPEN** command, the first drawing listed in the ***Work Dwg:*** *status drop-down menu* is closed, along with any of its associated ***overlays*** and ***references***. **OPEN SEPARATE** lets you load a drawing and keep the first drawing in memory.

Open Overlay

The* OPEN OVERLAY *command loads a selected file as an overlay, sharing the same coordinate space as other loaded overlay/reference drawings.

To use **OPEN OVERLAY,** type **OPENOVL** at the command prompt, or select **[File > Open Overlay…]** from the **FILE** menu. **FastCAD** displays the standard ***Load Drawing*** *dialog box*. Choose a file as you would using the regular **OPEN** command. The ***overlay*** loads into the same window as the first loaded drawing (loaded either by default when **FastCAD** started, or with the **OPEN** command).

You can load as many ***overlay files*** as your computer's memory allows.

All entities on an ***overlay file*** are editable as if they were in the main drawing. However, entities will ***not*** migrate between files. For instance, the **DRAG** command will only displace an entity within its source drawing, never to another drawing or ***overlay file***.

To draw entities on an *overlay* file, click the drop-arrow at the right end of the ***Work Dwg:*** *status window*.and select the ***overlay file*** name to be the *active* drawing. Draw new entities or edit the file as desired.

To ***save*** these changes, use the **SAVE** command **[File > Save]**. You can also use the **SAVE ALL** command **[File > Save All]** to save all loaded files.

Overlay as well as ***reference files*** are listed in the ***Work Dwg:*** *status drop-down menu* as well as in the ***External Reference [Xref] Management*** *dialog.* In the ***External Reference [Xref] Management*** *dialog* thay show the **OVL** label. If you select an ***overlay*** to be the *active* drawing, it will not appear in the ***External Reference [Xref] Management*** *dialog* until you select different ***active*** drawing.

To ***close*** an ***overlay file***, open the ***External Reference [Xref] Management*** *dialog* by clicking in the ***Work Dwg:*** *status window*, left-click the desired ***overlay***, and click the **DELETE** button.The ***Delete External Reference*** alert box appears asking if you really want to do this with **Yes/No** options If you use the **OPEN** command to load another drawing, all ***overlay*** and ***reference files*** will be closed.

A file opened with the **OPEN SEPARATE** command is not an ***overlay file***. They cannot be part of an ***overlay*** system unless they are ***closed*** and re-opened as such.

How does an ***overlay file*** differ from a ***reference file***?

A ***reference file*** is an ***overlay file*** that **cannot be edited**. That is, no entities on a ***reference file*** can be selected for editing. ***Reference files*** can be selected as *active* in the ***External Reference [Xref] Management*** *dialog*. ***Reference files*** can be made editable by checking the "**F**" [**Freeze**] control box changing its status from **FROZEN** to a selectable ***overlay***). Conversely, an ***overlay*** can be made a **read-only [FROZEN]** ***reference*** by checking its "**F**" control box.

Open Reference

The **OPEN REFERENCE** ***command loads a selected file as a*** **read-only FROZEN** ***overlay, sharing the same coordinate space as other loaded overlay/ reference drawings***. A ***reference file*** is exactly the same as an ***overlay file,*** except that entities on a ***reference file*** are **FROZEN and** cannot be modified.

To use **OPEN REFERENCE**, type **OPENREF** at the command prompt, or select **[File > Open Reference…]** from the menu. **FastCAD** displays the standard ***Load Drawing*** *dialog*. Choose a file as you would using the regular **OPEN** command. The ***reference*** loads into the same window as the first loaded drawing (loaded either by default when **FastCAD** started, or with the **OPEN** command).

You can load as many ***reference files*** as your computer's memory allows.

Entities on a ***reference file*** cannot be edited. They behave as if they were on a **FROZEN** layer. Modifiers such as ***endpoint*** will work with entities in ***reference files***.

Reference as well as ***overlay files*** are listed in the ***Work Dwg:*** *status drop-down menu* as well as in the ***External Reference [Xref] Management*** *dialog* with a darkened "**F**" **[FREEZE]** control box with the **OVL** ***overlay*** tag.

To ***close*** a ***reference file***, open the ***External Reference [Xref] Management*** *dialog* by clicking in the ***Work Dwg:*** *status window*, left-click the desired ***reference***, and click the **DELETE** button.The ***Delete External Reference*** alert box appears asking if you really want to do this with **Yes/No** options. If you use the **OPEN** command to load another drawing, all ***overlay*** and ***reference files*** will be closed.

A ***reference file*** is an ***overlay file*** that **cannot be edited**. That is, no entities on a ***reference file*** can be selected for editing. ***Reference files*** can be selected as ***active*** in the ***External Reference [Xref] Management*** *dialog*. ***Reference files*** can be made editable by checking the "**F**" [**Freeze**] control box changing its status from **FROZEN** to a selectable ***overlay***. Conversely, an ***overlay*** can be made a **read-only [FROZEN]** ***reference*** by checking its "**F**" control box.

Close File

The **CLOSE** ***command*** **[File > Close]** ***closes the active drawing without saving, and switches to another main working drawing***. If the ***active drawing*** is displayed in more than one drawing window, they are all closed. If no other drawing exists, a **new** drawing named **WORK.FC7** will be started as the ***active drawing***.

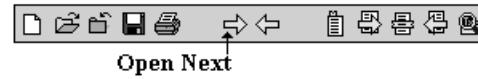
When using the **EDSYMDEF** command **[Insert > Edit Symbol Def]**, use the **CLOSE** command ***to save the modified symbol***.

Text equivalent: ***CLOSE*** *or* ***CTRL+W***

Files Icon Bar

The **Files Icon bar**, provides instant access to the most used commands for file manipulation. The icons from left to right are, **New Drawing, Open, Close, Save As Print, Open Next** and **Open Prior** as well as five ***Bookmark*** *icons* discused later in this chapter.

Open Next



FastCAD ***maintains an internal listing of the*** **last 10 files you've opened**. ***You can use the*** **OPEN NEXT** ***icon and*** **OPEN PRIOR** ***icon to respectively load the next or previous drawing file in the list.*** The file list is maintained in the registry, so it will persist even after closing and restarting **FastCAD** sessions.

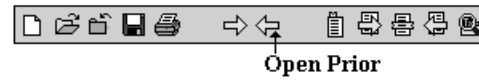
For fastest navigation, enable the ***File*** *Icon bar* using the **TOOLS** command, then use the **OPEN NEXT** *icon* to load the next drawing in the list.

If you use the **OPEN** command to load a file, and it already occupies a position in the file list, your "**current position**" will be set to that file's location. For example, you successively loaded *file1*, *file2*, *file3*, and *file4*. You then load *file2*. If you use the **OPEN NEXT** command, *file3* will automatically load.

Use ***bookmarks*** if you need more precise or flexible control over file sequencing.

Text equivalent: ***OPENND***

Open Prior



FastCAD ***maintains an internal listing of the*** **last 10 files you've opened**. ***You can use the*** **OPEN NEXT** ***icon and*** **OPEN PRIOR** ***icon to respectively load the next or previous drawing file in the list.*** The file list is maintained in the registry, so it will persist even after closing and restarting **FastCAD** sessions.

For fastest navigation, enable the ***Files Icon*** *Icon bar* using the **TOOLS** command, then use the **OPEN PRIOR** *icon* to load the previous drawing in the list.

If you use the **OPEN** command to load a file, and it already occupies a position in the file list, your "**current position**" will be set to that file's location. For example, you successively loaded *file1*, *file2*, *file3*, and *file4*. You then load *file2*. If you use the **OPEN PRIOR** command, *file1* will automatically load.

Use ***bookmarks*** if you need more precise or flexible control over file sequencing.

Text equivalent: ***OPENPD***

Save

The* SAVE *command saves your current drawing to disk. Use **SAVE** frequently to protect your work from power failure or computer malfunction. **SAVE** is also the ultimate protection from undesirable changes that you might make.

If you are working with multiple files (***overlays*** and/or ***separate*** files) and wish to save all drawings with one command, use the **SAVEALL** command.

Use this command explicitly when you complete your work. Even though **FastCAD** provides an **AutoSave** feature, it is not a substitute for the **SAVE** command. **AUTO SAVE** only saves the *active* drawing in in a file named **AUTOSAVE.FC7** in the **\FastCAD7** home directory. If you have a power failure or a system crash, you must immediatly restart **FastCAD** and use **OPEN** to open the **AUTOSAVE.FC7** file. Use **the SAVE AS** command and save it over the original file name.

You cannot perform any **REDO** or **REDOA** operation after a **SAVE or SAVE AS**.

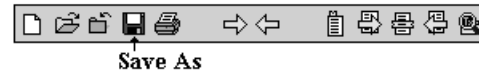
You can enable automatic file compression using the **OPTIONS** command.

Features and Functions that are attached to each drawing file when saved:

Portrait/Landscape printer orientation
Crosshairs ON/OFF
Layer names and their HIDE/FREEZE status
Grid patterns and Grid names
Line styles and names
Fill styles and names
Dimension styles and names
View sets and names
Button bar settings

Text equivalent: ***SAVE*** *or* ***CTRL+S***

Save As



The* SAVE AS *command saves a copy of your active drawing file under a new name or format. The new drawing file becomes your active drawing*. *Also, use* SAVE AS *to convert the active drawing to a different format (**DWG**, **DXF**, **EMF**, **BMP**.).

When you invoke **SAVE AS**, **FastCAD** displays the ***Rename & Save*** *dialog* box listing existing drawings. Select an existing drawing name with the cursor, or type a new name, then click the **OK** button. If you select an existing drawing, **FastCAD** will prompt for confirmation before replacing it.

To save the ***active drawing*** to a different file format, click the ***drop-arrow*** at the right end of the ***Save as type*** window and choose the desired output format from the list. **FastCAD** will automatically attach the appropriate file name extension.

You can use **SAVE AS** to preserve progressive versions of a drawing as separate files on disk. For example, if you start **FastCAD** with a drawing named plan1, you could use **SAVE AS** to make a copy with the name plan2 and then work on it.

Text equivalent: ***SAVEAS*** *or* ***CTRL+A***

Save All

The* SAVE ALL *command lets you save all opened files in one step. This includes all ***overlays*** and ***separate files***. ***Reference files are not saved because by their nature they are* read-only [FROZEN]**.

To use, type **SAVEALL** at the command prompt, or select **[File > Save All]** from the menu. ***All drawings save to their respective drives***. Drawing windows created by the **EDIT SYMBOL DEF** command that are still open will also be ***saved***.

You cannot perform any **REDO** or **REDO ALL** operation after a **SAVE ALL**.

To save a single ***overlay*** or ***separate*** file:

Click the drop-arrow at the right end of the ***Work Dwg:*** *status window*, select the an open file to make it the *avtive*.drawing and select the **SAVE** command **[File > Save].**

Text equivalent: ***SAVEALL***

Restore

The* RESTORE *command restores your drawing to the last version saved.

Before committing, **FastCAD** asks you "Is it **OK** to discard the changes to this drawing?". If you click **Yes**, **FastCAD** restores the drawing to its state at its last **SAVE** or **LOAD**. If you click **No**, **FastCAD** cancels the **RESTORE** command and returns you to the drawing screen without making any changes.

You can also use **RESTORE** to reverse the effects of a **DISCARD** command, provided that you did not **SAVE** the drawing after the **DISCARD**.

Discard

The* DISCARD *command clears all entities from the active drawing.

When you run **DISCARD**, **FastCAD** warns you: "T**his command will erase all entities in the current drawing and reset everything to the NEW drawing template, keeping only the drawing name. Is this what you want to do?**" To confirm, click the **Yes** button. If you click **No**, **FastCAD** cancels the command.

Until your next **SAVE** of the drawing, **RESTORE** can be used to recover the previous drawing state.

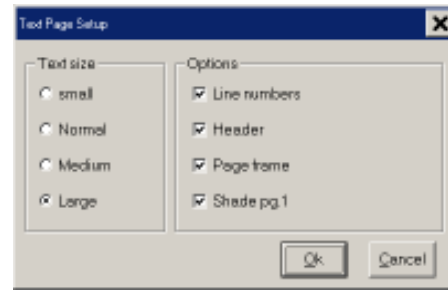
Print Setup

The* PRINT SETUP *command enables you to configure a printer before printing a* FastCAD *drawing. This command is similar to the Windows **Control Panel>Printers** option. Before you ***print*** a drawing, you must be sure the proper driver for your printer is installed. See your Windows documentation for information about installing printer drivers and setting up your printer. Parameters such as paper size, orientation, print resolution, and others (**depending on your printer**) can be specified here.

Text equivalent: ***PRINTSU***

Text Page Setup

TEXT PAGE SETUP *lets you change page set up options for printing from text windows*. Make your configuration choices in the ***Text Page Setup*** *dialog*.



FastCAD displays text windows for the following commands:

- **NOTES [File > File Notes]**.
- **EXTRACT attributes [Insert > Extract Attributes]**;
- **LIST [Info > List]**;
- **COUNT ALL [Info > Count All]**;
- **COUNT [Info > Count]**;
- **LIST VARIABLES [Macros > List Variables]**.

Text Page Setup dialog options

Text size This controls the size of the font when printing. Choose from small/normal/medium/large. Actual font sizes may vary from one printer to the next. All text from a text window prints in one font at one size.

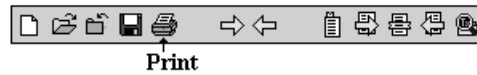
Line number Prints line numbers in the left margin space.

Header Prints printing information at the top of the page, such as the drawing filename, page number, and date/time stamp.

Page frame Prints a line border around the entire page (and line numbers and header if those are configured on).

Shade pg. 1 The header section of the first page is printed in a grayscale so the first page can be readily identified.

Print

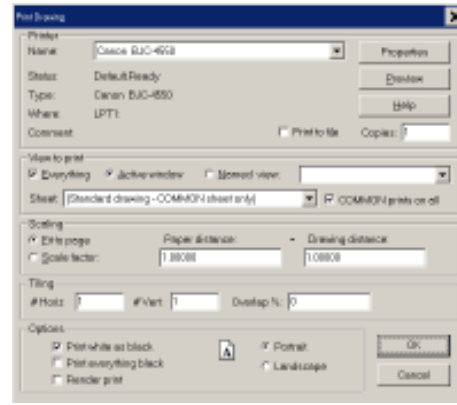


The* PRINT *command displays the Print Drawing dialog box enabling you to control how your drawing is printed. After you confirm your options and choose **OK**, **FastCAD** sends your print file to the Windows print spooler, and you will be able to continue your work before the print is finished.

Note: The terms "**print**" and "**plot**" are used interchangeably.

Text equivalent: ***PRINT*** *or* ***CTRL+P***

Print Drawing dialog box options



Choosing a new printer Click on the ***Name*** *control* to display a drop list of printers installed in your Windows system. Select the desrired printer.

Setting up the printer Choose the ***Properties*** *button* to access the printer driver settings. The exact appearance and available options of the printer property sheet will vary for each printer model and its associated driver software.

Seeing a print preview Choose the ***Preview*** *button* to see an accurate representation of your drawing as it will appear on your paper. **FastCAD** interrogates the Windows Printer Driver for information on the capabilities of the printer. Not all printers (and/or printer drivers) have the same ability to reproduce colors, fill patterns, and/or **TrueType** fonts.

Printing to a file Check the **Print to File** control to send output to a file. **FastCAD** displays the ***Print to File*** *dialog*, requesting a filename and destination.

Setting number of copies Type the number of copies you want to print into the **Copies** box.

Choosing the view to print You can choose to print all or a specified part of your drawing. Choose **Everything** to print the entire drawing. This includes space occupied by entities on hidden layers. (It's the same view you would see if you selected the **ZOOM EXTENTS** command or icon.) Hidden entities will not show in the printed drawing, but they do affect the print size and placement. Or choose **Active Window** to print the current view in the active drawing window. Or select a **Named View** from the drop list. The list displays all views that were named using the **SET VIEW NAM**E command **[View > Set View Name]**.

Scaling the printout You can choose one of two methods to size your drawing onto the page. **Fit to page** automatically scales the selected view for a best fit on your paper. Remember to set the proper paper orientation in the **Options** section of the dialog box. **Scale Factor** uses the relationship between paper distance and drawing distance to print a scaled version of the selected view, centered on the center of the view. For example, to print at "**1/8th scale**", many ways to describe it exist:

- **Paper distance** = **1/8"** and **Drawing distance** = **1'**
- **Paper distance** = **1/8** and **Drawing distance** = **12**
- **Paper distance** = **1'** and **Drawing distance** = **96**

Printing large drawings on multiple sheets Use the **Tiling** option to print large drawings over multiple sheets of paper. To specify the number of sheets, enter the values for **# Horiz** and **# Vert** (the total number of sheets = # Horiz x # Vert). The portrait/landscape orientation setting is honored. You can also specify the **Overlap %**. Having a small amount of overlap can help align your sheets during assembly. Use Preview to inspect your settings before you print. The entire print will be displayed with red lines demarcating each page's unique print elements.

Sheet Choose a sheet option for printing. Drawings constructed with sheets allow you to perform batch printing operations.

- **COMMON only (standard drawing)**
  Prints one page containing only entities on the COMMON sheet.
- **All visible sheets as overlays (single page)**
  Prints one page containing all entities on visible sheets.
- **All visible sheets as separate pages**
  Prints multiple pages, each visible sheet as a page.
- **All sheets as separate pages**
  Prints multiple pages, each sheet as a page regardless of hidden/visible status.

- The currently selected sheet

  Prints one page, only the ***current sheet***.

COMMON prints on all Choose if you wish entities on the **COMMON *sheet*** to be printed no matter what Sheet option you select.

Printing white colored entities Check the **Print white as black** option if your printer does not print white entities.

Printing everything black Check the **Print everything black** option to suppress color information. The drawing will print out only in black, even on color printers.

Setting Page Orientation Select the printing orientation either as **Portrait** or **Landscape**.

Bookmark commands

Bookmarks are file management aids that help you efficiently organize and navigate your drawing files. This is accomplished through the use of ***bookmark files***. A ***bookmark file*** is simply an editable list of files that work with the **Next/Current/ Prior *Bookmark*** navigation icons that reside on the ***File Icons*** *Icon bar* .

Additionally, ***bookmarks*** can be specifically created and distributed along with drawing files you send to other users. This is especially useful in conjunction with **View7**, which completely supports all **FastCAD *bookmark*** features. ***Bookmarks*** help you control what the person viewing the drawing sees—not only which files, but also the order in which those files are viewed.

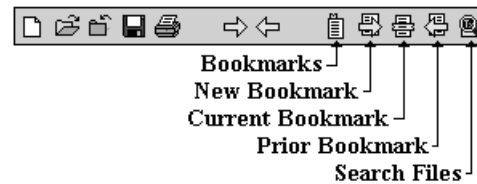
Files loaded with the ***bookmark*** commands replace the current drawing in memory. For example, if six files are viewed in succession using the ***bookmark*** commands, each load replaces the previous. In other words, **FastCAD** will *not* open six separate files simultaneously in memory. This keeps **FastCAD *bookmark*** behavior consistent with **View7** (which does not support multiple files).

Bookmark files are saved as **.BKM** files. They are regular text files that can be viewed, created, or modified with any regular text editor.

FastCAD v7 contains commands needed to either use or create ***bookmar***k ***files***.

To Use Bookmarks

To simplify ***bookmark*** navigation, use the **TOOLS** command **[Specs > Screen Tools…]** to activate the ***File Icons*** *Icon bar* by clicking the desired placement arrow box. There, you will find ***icons*** that let you navigate and manage your ***bookmarks***.



To Create or Edit Bookmarks

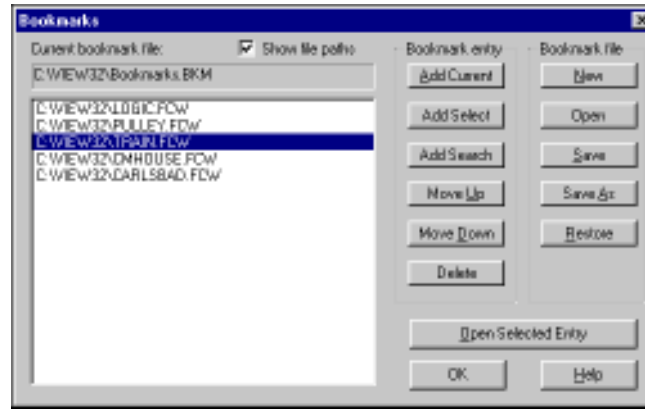
Use the **BOOKMARK** command (or choose the ***Bookmarks*** *icon*) to display the ***Bookmarks*** *dialog*. This permits creation or modification. You can manually ***order*** the files in the ***bookmark*** list, or you can ***add*** or ***delete*** files.

Edit Bookmarks

To ***create*** or ***modify*** a ***bookmark*** file, start the **BOOKMARK** command from either the ***Menu*** *bar* **[File > Bookmarks > Edit Bookmarks…]** or the ***FIles Icons*** *Icon bar*. Doing so will display the ***Bookmarks*** *dialog box*. Use the ***Bookmarks*** dialog to perform all modifications to your ***bookmark file(s)***.

Bookmark files are saved with a *.bkm* extension. A **.BKM** ***bookmark file*** is a regular text file that simply consists of a list of files (exactly as shown in the ***Bookmark*** *dialog* when **Show file paths** is toggled on). Therefore, you can use a regular text editor, such as **NED**, to create or edit a ***bookmark file***.

Also use this command to quickly jump to a selected ***bookmark file***, or to view the ***bookmark*** list.



Bookmark dialog box

Current bookmark file This displays the active ***bookmark*** list as displayed in the list window. You can load a new ***bookmark file*** by choosing the ***New*** *button*.

Show file paths When checked, the full path for each file, including the drive letter, is displayed in the list window. When unchecked, only the file names are displayed.

Add Current Adds the currently loaded drawing to the end position on the ***bookmark*** list. A file can be added more than once to the ***bookmark*** list.

Add Select Adds a file you select from the ***Load Bookmark*** *dialog*. You can change drives or directories and select a new drawing to be added to the end of the list. A file can be added more than once. A file you select using this method will not load into the viewer.

Add Search Lets you select files that meet matching criteria using the **SEARCH** command. Files that match the criteria are displayed in a dialog box, where they can be individually added to the end of the ***bookmark*** list.

Move Up Manually moves the highlighted file up one position on the list. In the illustration, doing so would move **TRAIN..FC7** between **LOGIC.FC7** and **PULLEY>FC7**

Move Down Manually moves the highlighted file down one position on the list. In the illustration, doing so would move *train.fc7* between *cmhouse.fc7* and *carlsbad.fc7*.

Delete Removes the highlighted file from the ***bookmark*** list. Note that deleting here only removes the filename from the ***bookmark*** list; it does not erase the actual drawing file from its source.

New Clears the current ***bookmark file*** from memory. You can then create a new ***bookmark file***, or load one from disk.

Open Loads a selected ***bookmark file*** (ending with a .bkm extension). Select the new ***bookmark file*** from the ***Load Bookmark File*** *dialog*.

Save Saves the current ***bookmark file*** to disk, using the name shown in the ***Current Bookmark*** file field.

Save As Lets you save the listed ***bookmark*** to a different name or path.

Restore Reverts the list to its condition as it was immediately after your last *save*. It basically undoes any unsaved modifications you may have made to the ***bookmark*** list.

Open Selected Entry Choose to load the highlighted drawing into your **FastCAD** drawing window. When new files are loaded via ***bookmark*** functions, the previous loaded file will automatically close (unless you modified the drawing, in which case you will be ask to confirm first).

Whenever a file is loaded in this manner, the ***bookmark*** pointer is set to the file position that you selected. The ***bookmark*** navigation buttons will operate from this position.

Text equivalent: ***BOOKMARK***

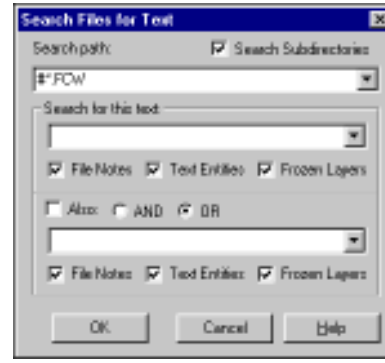
Search Files

The* SEARCH FILES *command lets you search for text matches in drawing files without loading them first. The ***search*** can be configured to ***search* FastCAD v7 .FC7** drawing files in a specified directory (and in subdirectories, if desired). You can configure the ***search*** to look in any combination of ***file notes***, ***text entities***, and ***text entities on frozen layers***.

Files that match the ***search*** criteria are displayed in a ***Bookmark*** dialog. You can ***edit*** and ***save*** the ***bookmark*** file.

The speed of the ***search*** depends upon several factors, such as the number of searched files, the size of the files, the speed of your processor, and the speed of your drives. ***It is highly recommended that searches be performed only on files residing on hard disks or drives similar in performance***. Searching a floppy disk will take many times longer than searching the same files on a hard disk.

The five most recent search paths and pattern strings are saved in the registry. When the ***Search Files for Text*** *dialog* appears, it is initialized to the latest settings.



Search Files for Text dialog box

Search Path Enter the drive, folders, and file filters you wish to search. Use the # character (**number sign**) as shorthand for the **FastCAD** working directory.

Search Subdirectories Check if you wish to search files contained in directories nested below the level specified in Search Path.

Search for this text Enter the text string you wish to search for.

File Notes/Text Entities/Frozen Layers Check these in combination to direct the text search. **File Notes** and **Text Entities** are actual items that are searched, while **Frozen Layers** is a refinement of **Text Entities**. If **File Notes** and**Text Entities** are both unchecked, the search will yield zero results.

Also: And/Or You can further refine your search by adding an additional search string. To do so, check the **Also** box, then specify whether the additional criteria is **And** or **Or**. If **And** is selected, ***a search will yield results that contain both the first text***

and the second text. If **Or** is selected, ***a search will yield results that contain either the first text or the second text.***

Text equivalent: ***SEARCH***

Index

The* INDEX *command can dramatically speed up* SEARCH *operations. This command creates an ***index file***, ***index.idx***, which can be specified as the file path in a later **SEARCH** command. The ***index file*** contains the information that the **SEARCH** command uses each time it is run; specifically, all text information in the targeted drawings.

When text entities are added or deleted from drawings, a new index file should be generated to keep it synchronized. The matching index file(s) for a drawing will not automatically update.

To generate an ***index file***, type "**INDEX**" at the command line and press ENTER:

1. The prompt reads "**Index drive:path*.fcw**:".

   Type in the drive and path specification. For instance, if you wish to index **FastCAD** drawings on your **D:** drive, in a folder named "**job4074**", enter "**D:\job4074*.fc7**". **FastCAD** will also process any folders contained in the target folder (subdirectories).

   FastCAD does not provide or accept default entries for this prompt.

2. **FastCAD** generates the ***index file***.

   This stage can take a while if you are processing many files. The Status bar displays the progress of the indexing. The ***index file*** is created in the specified drive and path, with the file name ***index.idx***. If this file already exists, it will be automatically overwritten.

After an ***index file*** is created, you may manually rename ***index.idx*** (it should end with an .idx extension).

File references contained in the *index* file are relative to the path of the *index* file, so it and its directory and subdirectory structure may be moved to other locations, such as a **CD-ROM** drive. You should not ***index files*** directly off of a **read-only** drive, as **FastCAD** will be unable to write the ***index file*** to that location.

Using an index file

An ***index file*** is used with the **SEARCH** command. In the ***Search Files for Text*** *dialog*, the i***ndex*** file should be specified in the **Search Path** option. The **SEARCH** command then processes the ***index*** file, which can be much faster than processing each drawing file "**on the fly**".

File Notes

FILE NOTES ***give you the ability to attach informational notes to any drawing. Each note has a name, and more than one note can be contained in any drawing***. Each set of named ***notes*** is saved with the drawing and can be viewed or edited after the drawing is loaded.

List Window Existing ***notes*** are listed alphabetically by name. To ***open***, ***edit***, or ***delete*** an existing ***note***, pick a ***note name*** and click the appropriate button.

OK Displays the contents of the selected ***note***. The *note* contents are displayed in a standard text window. From the text window, the ***note*** can be printed or saved to a text file. If you do not first highlight an existing *note*, **FastCAD** will return you to the drawing screen.

Cancel Ignores changes you have made, returning to the **FastCAD** drawing screen.

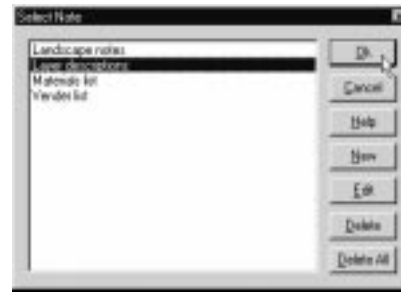
Help Displays the on-line help topic.

New Creates a **new** ***note***. First, enter a name for the ***new note***. Then enter the ***note*** text in the standard ***Edit Text*** *dialog box*. When you click **OK**, the note is created and the dialogs close, returning you to the drawing screen.

Edit Modifies the selected ***note***. The existing text is displayed in the standard ***Edit Text*** *dialog box*.

Delete Deletes the selected ***note***. **FastCAD** then closes the dialog and returns you to the drawing screen. **Note**: **UNDO** will not work with this command. A deleted ***note*** cannot be recovered past your last **SAVE**.

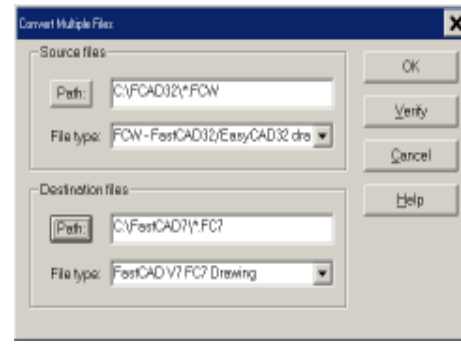
Delete All Deletes **All Notes** from the drawing at one time. **Note**: **UNDO** will not work with this command. A deleted ***note*** cannot be recovered past your last **SAVE**.



Text equivalent: ***NOTES***

Convert Files

CONVERT FILES ***allows you to perform batch file conversions***. All files that match the file filter you choose in the **Source File Type** list will be converted to the **Destination File Type**. Choose the **Path:** button to redirect the location of either the ***Source*** or ***Destination files***.



Convert Multiple Files dialog

If you only need to convert a single file *to* **FC7**, use the **OPEN** command and choose the appropriate source file type.

If you only need to convert a single file from **FC7** to another format, **OPEN** the **FC7** file, then use the **SAVE AS** command and choose the desired destination file type.

Text Equivalent: ***CONVF***

Protect

***The* PROTECT *command allows you to provide limited overwrite protection or output restrictions for any* .FC7 *drawing files you work on*. *The restrictions you select are honored by* FastCAD v7 *and the* View7** viewer.

File ***protection*** works by assigning a numerical **Key ID** to the drawing file. If you assign a **Key ID**, keep a record of it (written or otherwise). Once a **Key ID** has been assigned, and protection options have been set, the same **Key ID** must be entered to modify or disable those protection options.

To change the **Key ID** of a drawing, first enter the valid **Key ID**, uncheck all of the *protection* options, then choose **OK** to close the ***Change Drawing Protection Settings*** *dialog*.

To *protect* a file, select **[File > Protect]** from the *Menu bar*, or type **PROTECT** at the command prompt:

1. A dialog will appear, prompting you to enter a key number to change protection. If the drawing has never been protected before, enter a new value. The **Key ID** is required if you wish to change any protection settings in the future. If this file has already been protected, enter the file's known **Key ID** number.

   A **KeyID** should consist only of digits, and the value must not be zero. The **Key ID** is a 32-bit value, so it can be quite large (into the nine digits). Do not use alphabetic characters. The **Key ID** terminates at the first alphabetic or non-numerical value. For instance, if you enter "**23qwerty**" as the **Key ID**, it is stored as "**23**".

2. If the proper **Key ID** is entered, **FastCAD** displays the ***Change Drawing Protection Settings*** *dialog*. The settings can be toggled on or off in any combination: Change the settings and choose **OK** to accept the changes.
3. The drawing must be saved before the drawing is distributed.

Change Drawing Protection Settings dialog box

Preventing entities from being modified Choose **This drawing cannot be changed**. Style settings also cannot be changed. The current print settings are used. File

Notes can be edited when this option is checked. The drawing cannot be copied to the clipboard, nor can it be inserted as a part.

Note that layers can still be manipulated to the point that actual entities are not affected. For instance, layers can be created and deleted (if they do not contain entities). Layers can be frozen or thawed, hidden or shown. To prevent layer manipulation, choose **Layer status cannot be changed**.

Preventing a Drawing from being printed Choose **This drawing cannot be printed**. If someone attempts to print the file, **FastCAD** will display a warning box stating that the file is protected.

Protecting layer settings Choose Layer status cannot be changed to lock out users from the layer management controls and dialogs. Frozen and hidden layers will remain as such, and entities on those layers will behave and appear appropriately. If **This drawing cannot be changed** is unchecked, entities on shown and thawed layers can be edited.

Text Equivalent: ***PROTECT***

Menu

The* MENU *command loads a custom menu file for use with current drawing. **FastCAD**'s standard *menu* file is called **FCW7.MNU**.

FastCAD lets you select a ***new menu*** using the standard ***Select menu*** *dialog*. When used with the **MENU** command, the dialog displays files ending with the **.MNU** extension.

If you want to load a ***new menu*** for use during the entire current session (i.e., even after you load new drawings), use the **MENUD** command **[File > Set Menu]** instead.

Creating ***custom menus*** can greatly increase your productivity by tailoring **FastCAD** for the way you work. If, for example, you use many circles and arcs in your work, you could build a ***custom menu*** file with a ***menu*** item for each of the seven circle and six arc commands to bypass the **CIRCLE** and **ARC** sub-menus. Another ***custom menu*** could call some or all of the **CHANGE** commands directly, bypassing the **CHANGE** sub-menu. If you wish, you can add these menu items to the standard **FCW7.MNU** menu file, or to a customized menu with a different name.

You can also build alternate ***menus***, ***scripts***, and ***macros*** for complex or repetitive tasks, loading them instantly. The first addition you might make to a ***custom menu*** (**MYMENU..MNU**) could be a line (**or menu item**) that loads another ***menu***. As an example, we'll add a ***menu*** command to open a sample ***custom menu*** included with the installation, ***solar.mnu***. Add the new line between **Menu** and **Set Menu** under the "**File**:" heading:

Menu :|MENU;
Solar Menu :|MENUM solar.mnu;
Set Menu :|MENUD;

After this *menu* is saved and loaded, a click on the new *menu* item, "**Solar Menu**", will instruct **FastCAD** to load the menu named ***solar.mnu*** (bypassing the dialog box).

Auto-loading menus

Any drawing can have its own custom *menu*! A *menu* file with the same name as a drawing file will automatically load with the drawing, provided the **.MNU** file is located in the same directory as the **.FC7** file. For a working example of this, open the drawing ***solar.fc7*** using the **OPEN** command **[File > Open]**. Note that the standard **FastCAD** ***menu*** has been replaced by a ***custom menu*** named ***solar.mnu*** that facilitates the exploration of our solar system.

Text equivalent: ***MENU***

Set Menu

The* MENUD *command loads a menu file and makes it the default for the current session. Loading new drawings will not reset the menu back to the* FastCAD *standard menu (**the menu file named FCW7/MNU**). If you close **FastCAD**, and then re-start it, the ***standard menu* FCW7.MNU *loads*, *not the menu specified with* MENUD**.

The regular **MENU** command loads a ***new menu*** as well, but loading any new drawing resets to the ***standard menu***. **MENUD** persists for the session.

Text equivalent: ***MENUD***

Script

The* SCRIPT *command loads and runs a script file (**a series of FastCAD commands saved in a plain ASCII text file**). **FastCAD** displays the ***Load script to run*** *dialog* listing available ***script*** files (**those with the type extension .SCR**). Select a ***script*** with the cursor or type a ***script*** file name.

Use a script to automate repetitive tasks. ***Scripts*** can include ordinary **FastCAD** commands (**text equivalents of the ones in the standard menu**) or special programming commands.

Scripts and macros

Scripts are very closely related to ***macros***. In fact, almost all ***scripts*** can be converted into ***macros*** by inserting them between the **MACRO-ENDM** block. All rules that apply to creating ***macros*** apply to ***scripts***.

Scripts differ from ***macros*** in only a few ways. Each ***script*** resides as a separate file, while many ***macros*** can be consolidated into one or more *macro* files. ***Scripts*** run as soon as they are loaded, while ***macros*** run only when a ***macro*** name is invoked. Because ***macros*** can be much more easily "**bundled**," they facilitate complex tasks much easier than ***scripts***, especially with regards to subroutines and other conditional branching.

For rules governing ***script*** creation, please see the **Macro Command Reference**.

Running scripts

To use a ***scrip***t, invoke the **SCRIPT** command. Type the name of a ***script*** file, or select a file with the mouse. **FastCAD** reads and executes the text in the ***script*** file just as if you had typed it all at the keyboard.

You can use a *script* file to execute a series of **FastCAD** commands, but not to define ***macros***. Unlike **LOADMAC**, running a ***script*** does not clear existing ***macros***, although it does redefine any macros, variables, or labels with the same names.

Automatically running scripts

After **OPEN/LOAD** or **LOADM** commands, or after loading a drawing from the **FILE** menu's recently used file list, a ***script*** file present with the same name as the drawing ***will autorun***. ***Autorun is disabled in* View7**, ***and does not occur when multiple files are loaded*** (as ***references*** or ***overlays***).

Text equivalent: ***SCRIPT***
*Macro equivalent (**bypass dialog box**):* ***SCRIPTM***

Template

FastCAD TEMPLATE ***files are used as a starting point for new drawings that use units that are different from*** **FastCAD**'s ***default units*** (**inches**). ***Template*** files behave exactly the same as regular **FastCAD** drawing files, except that they have a different extension (**.FT7**) that renders them less susceptible to accidental overwriting.

FastCAD uses many scaleable properties, such as ***line styles*** and ***fill styles***, which are dependent on the current drawing units. Switching units from **feet** to **millimeters** can drastically alter their appearance. That is why **FastCAD** ships with two standard ***templates*****:** ***English*** (**for inches**) and ***Metric*** (**for millimeters**).

You can store many startup conditions in a ***template***, such as all your ***custom style properties*** (**line, fill, text, dims, etc.**), ***screen tools options*** (**scroll bars, tracking, icon bars, etc**.), and ***multiple windows***. A ***template*** can contain **symbols** or ***a pre-drawn border***. Basically, whatever you can store in a regular drawing, you can store in a ***template***.

Using a Template

Templates are not intended to be applied to a drawing after the drawing has already been created. ***Templates should be used before the file is created***.

1. To use a ***template***, select **[File > Template]** and choose a ***template***.
2. Immediately choose **[File > New Drawing]**. The new drawing uses the ***template*** as its starting point.

Creating a Template

FastCAD ships with two default ***templates***: **English** and **Metric**, named after the units they are based on. But you can create your own ***templates*** from any **FastCAD** drawing.

1. **OPEN** the **FastCAD** drawing you want to use as a *template* source.
2. **ERASE** any entities you do not wish to appear in your *template*.

B. Select **[File > Save As]**, then select file type "**FastCAD V7 FT7 Template**". The only way to save a ***template file*** is with **SAVE AS** (the regular **SAVE** command saves only **.FC7**, not **.FT7** files), therefore, the chances of inadvertently overwriting an existing ***template*** are slim.

Text equivalent: ***TEMPLATE***

AutoSave

AUTOSAVE ***automatically creates a backup of your active drawing in a file named*** **AUTOSAVE.FC7** ***in the*** **FastCAD7** ***home directory***. Several options are provided to adjust the timing increments and the appearance of on-screen prompts.

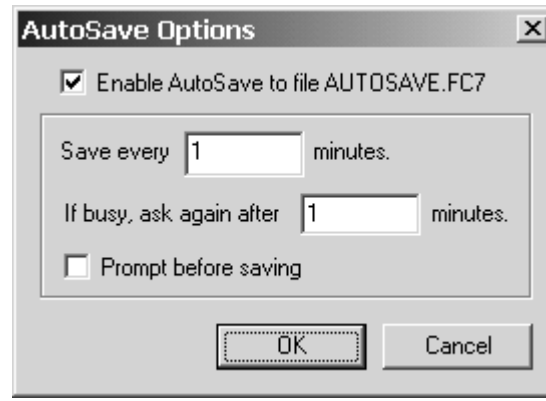
Usage

Important: ***You must still use the*** **SAVE** ***command while you are working and when you end your drawing session.*** **AUTOSAVE** ***is a precautionary feature that in no way replaces the*** **SAVE** ***command***. ***The Automatic backup is saved in a file named*** **AUTOSAVE.FC7**. ***AutoSave*** does ***not modify or update your original*** **.FC7** ***file***.

Use the ***AutoSave*** menu command **[File > AutoSave]** to access the ***AutoSave Options*** *dialog box*. From the dialog, you can turn ***AutoSave*** **on** or **off** and adjust other options.

In order to prevent interference with running ***macros*** or ***scripts***, ***AutoSave*** occurs immediately and only after user input through keyboard entries or mouse clicks. If no direct user input occurs for the duration of an ***AutoSave*** interval, the ***AutoSave*** will take place immediately after the first subsequent keyboard entry or mouse-click.

AutoSave Options dialog box



Enable AutoSave to file AUTOSAVE.FCW Check this option to enable the **AutoSave** feature. The drawing in memory is saved to a file named **AUTOSAVE.FC7**. The original drawing is never overwritten. When this option is checked, the menu entry **[File > Autosave…]** displays a check mark to remind you that it is active.

Save every *n* minutes Specify the ***AutoSave*** interval.

If busy, ask again after *n* minutes Specify the interval **FastCAD** will wait before asking again when instructed to **Ask Later** (in the ***Time to Save Your Work*** *dialog box*).

- **Save Now**
- **Ask Later**
- **Disable AutoSave**

Prompt before saving

If you find this reminder prompt annoying, you can toggle off its appearance with this option. ***AutoSave*** will then operate completely in the background.

Recovering a Drawing

In the event your current drawing session unexpectedly terminates (**due to a crash, power failure, etc**.), you can recover your entire drawing up to the most recent ***AutoSave***.

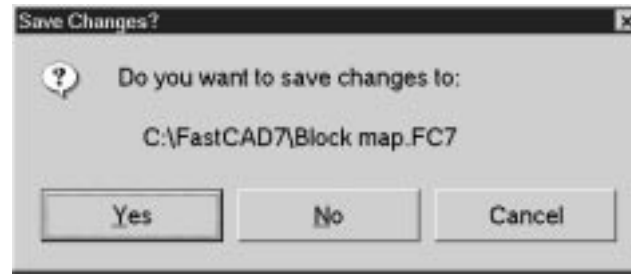
To recover, restart* FastCAD *and* IMMEDIATELY *open the file* AUTOSAVE.FC7 *from the* \FASTCAD7 *home folder*. *Then use the* SAVE AS *command to save the recovered drawing with its original filename or a newfilename.

Text equivalent: ***ASAVE***

Exit

The* EXIT *command closes* FastCAD *and returns you to the Windows desktop.

If you have made any changes since the last **SAVE** of your drawing, **FastCAD** displays the ***Save Changes*** *dialog box*:



If you do want to quit without saving, choose **NO** or type "**N**". Choose **Cancel** to return to **FastCAD or** press **ENTER,** click the ***Yes*** *buttom or type a "**Y**"*.

Text equivalent: ***EXIT*** *or* ***QUIT*** *or* ***END***

Chapter 6

Clip Commands

Undo errors, and cut and paste with the Windows clipboard

FastCAD's **Clip** *menu* contains commands commonly found in the "**Edit**" menu of most Windows programs (**FastCAD** uses the term "**Edit**" to denote entity modification). Such commands include the undo and clipboard functions.

Undo Functions

FastCAD includes an efficient multiple undo facility, allowing you to backtrack draws and edits up to your last ten commands, or your last save, whichever occured last.

Clipboard Functions

The clipboard commands let you **COPY** or **CUT** all or part of your drawing to **PASTE** into other Windows applications. For instance, you can insert your drawing as clip-art into a word processing or desktop publishing program. You can use the **PASTE** command to import text you may have composed, spell-checked, and grammar-checked in a word processing application.

Undo or CTRL+Z

The Last Ten Steps

Why does FastCAD only allow you to undo the last ten commands?

The answer lies in your computer's memory, or RAM. As you perform operations, entity modifications are buffered. For instance, if you arrayed 10,000 circles, and then erased them, FastCAD could restore them with UNDO because the circles' properties were retained.

As you can see, if FastCAD allowed unlimited UNDO, you could rapidly use up your computer's available memory, especicially if you performed multi-entity operations on a very large drawing. If Windows runs out of system memory, a lock-up results.

FastCAD provides a multiple level undo facility. At almost any point in your drawing, you can backtrack your steps until you reach the point of your last **SAVE**. Using the auto-repeat feature (mouse-click to repeat commands), you can sequentially see the effect of each undo.

UNDO will work successively for the previous ten (**10**) commands, or the last save, whichever is encountered first.

UNDO works on changes made to the drawing database. That is, drawing and editing operations that affect entities. As a general rule, **UNDO** does not affect changes made to the drawing environment, such as the current color and layer.

To redo an **UNDO** (undo the **UNDO**), select **REDO**. Doing so will redo the last step undone. You can continue to redo until you reach the last draw/edit you performed. The **REDO** command only works if it immediately follows an **UNDO** command. For instance, if you **UNDO**, draw a line, and then attempt a **REDO**, the **REDO** will have no effect. (In this case, the drawing database has changed since the prior **UNDO**.)

To redo a series of undo commands, use the **REDOA** command. As long as the **UNDO** commands were performed sequentially, the original commands can all be restored with **REDOA**. Like the regular **REDO** command, there cannot be an intervening command.

If you wish to toggle between the original and undone states, use the **FLIP** command. After invoking **FLIP**, you can auto-repeat by mouse clicking anywhere on the drawing. Each click will flip back and forth between the "**before**" and "**after**" versions of your drawing.

After you are satisfied with the **UNDO** or **REDO**, you can **SAVE** your drawing to compress it, permanently removing the unused versions of the entities. This is important if you've performed many drawing or editing commands that involved large numbers of entities. The database compression that automatically occurs with a **SAVE** command can free up useful chunks of memory.

Text equivalent: ***UNDO***
Keyboard Shortcut: ***CTRL+Z***

Redo or CTRL+Y

REDO will reverse the last command that was **UNDO**ne. If a series of **UNDO**s have occurred, you may continue to redo commands up to the initial **UNDO** by repeating the command.

Note: **REDO** only works if it is applied immediately after one or more **UNDO** commands. If another command besides **REDO** is invoked after an **UNDO**, a subsequent **REDO** will have no effect.

To undo a **REDO**, select **UNDO**. You can also select the FLIP command to toggle between the **UNDO/REDO** drawing states.

Text equivalent: ***REDO***

Keyboard Shortcut: ***CTRL+Y***

Redo All

Use the **REDO ALL** command to redo a series of undo commands in one step. As long as the **UNDO** commands were performed sequentially, the original commands can all be restored with **REDO ALL**. Like the regular **REDO** command, there cannot be any other intervening commands.

If **REDO ALL** was used to restore multiple **UNDO** commands, to then return to the pre-**REDO ALL** state you must invoke that number of **UNDO**s.

Text equivalent: ***REDOA***

Flip

The **FLIP** command acts as a toggle between the last **UNDO** or **REDO** command. After invoking **FLIP**, you can auto-repeat by mouse clicking anywhere on the drawing. Each click will ***flip*** back and forth between the "**before**" and "**after**" versions of your drawing. This action is similar to the **FastCAD/DOS** style of single-level undo/redo.

Text equivalent: ***FLIP***

Copy or CTRL+C

The **COPY** command works with the Windows clipboard. It places selected entities temporarily on the clipboard. You can then paste the selected entities into any other Windows application. Original entities remain unchanged.

FastCAD makes three versions of data file formats available on the clipboard:

- **FastCAD** format (**.FC7**)
- Windows **E**nhanced **M**eta**F**ile format (**.EMF**)
- Windows Bitmap format (**.BMP**)

Don't confuse this command with the **COPY** command found in the **COPY** *menu*.

Text equivalent: ***CLIPCOPY*** *or* ***CTRL+C***

Cut or CTRL+X

CUT places entities on the clipboard and then removes them from the drawing, just like **ERASE** (***Edit*** *menu*). You can then paste the selected entities into any other Windows application.

Text equivalent: ***CLIPCUT*** *or* ***CTRL+X***

Paste or CTRL+V

PASTE copies valid clipboard content from the Windows clipboard into the current drawing.

If the clipboard data consists of **FastCAD** entities, then it will paste in as a **FastCAD** ***part***, using the clip data origin as its insertion point. ***Parts*** are discussed in **Chapter 11**.

If the clipboard data is text from a word processing program, for instance, **FastCAD** converts it to a text entity, using the current text specifications. Text formatting infomation governing original formatting, fonts, etc., is ignored.

FastCAD cannot import raster images from the clipboard. Raster images include file formats such as **.BMP**, **.GIF**, **.JPG**, **.TIF**, etc. Instead, use the **PICTURE** command to directly place such images into your drawing.

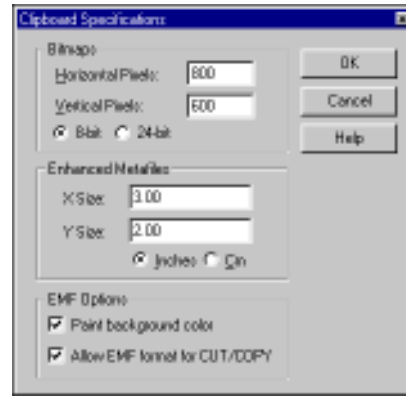
The **CTRL+V** keyboard shortcut **PASTES** entities if the clipboard contains a drawing file, but pastes text *into the input stream* if it contains text. In other words, clipboard text will be entered at the command prompt, not as a text entity.

Text equivalent: ***PASTE*** *or* ***CTRL+V***

Clip Options

When you select **OPTIONS**, **FastCAD** displays the ***Clipboard Specifications*** *dialog box*. You can select parameters for sizing either **bitmap** (**.BMP**) or **enhanced metafile** (**.EMF**) images. You can force output to either one of the formats.

The **bitmap** and **enhanced metafile** size parameters have relevance when using the **PASTE** command to insert your entities into other Windows applications. Some target applications have a "**Paste Special**" command that allows explicit determination of the paste format (**.BMP** or **.EMF**).



Clipboard Specifications dialog box

Bitmaps

Horizontal Pixels Sets the horizontal bitmap size. **FastCAD** offers a default value of 200 pixels in the **X**-direction (horizontal). A value of 1 to 999 may be specified.

Vertical Pixels Sets the vertical bitmap size. **FastCAD** offers a default value of 200 pixels in the **Y**-direction (vertical). A value of 1 to 999 may be specified.

Enhanced Metafiles

X Size Sets the horizontal enhanced metafile size. **FastCAD** offers a default value of 3.0 Inches for the horizontal dimension. Note that the largest value you select for either the **X** size or **Y** size will be the size your clip entities fit into. The aspect ratio of the selected entities is never altered (that is, the entities will never appear stretched or squashed).

Y Size Sets the vertical enhanced metafile size. **FastCAD** offers a default value of 2.0 Inches for the vertical dimension.

Inches / Cm Sets the units of measurement for the metafile only.

Options

Paint background color Copies the opaque background color as seen in **FastCAD**. If not checked, the copied entities will be on a white background.

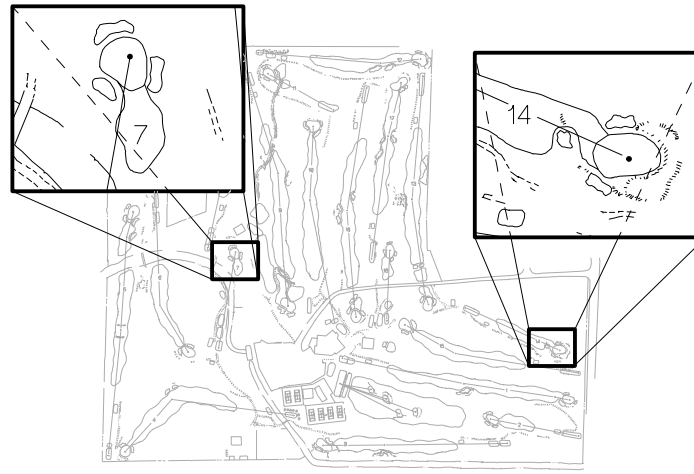
Allow EMF format for CUT/COPY Forces clip output to the enhanced metafile format. If not checked, clip output will be in the bitmap (.bmp) format. Bitmap output provides greater compatibility, particularly with older or 16-bit Windows applications. However, unless a high bitmap resolution is specified, bitmaps may appear blocky when resized.

Text equivalent: ***CLIPOPT***

Chapter 7

View Commands

Controlling what you see on the screen



Use Zooms to inspect or edit large or complex drawings.

Keep your drawing in perspective

One common mistake newcomers to CAD make is drawing too much detail, because suddenly you can see something 100 times its normal size without straining your eyes.

Be aware of what will really be seen on the printout. Use the ZOOM ALL or ZOOM EXTENTS command to keep your drawing in perspective.

The **View** commands control what you see in the active drawing window. Because they allow you to navigate, survey, and see detail in your drawing, you will use these commands often in the course of a typical **FastCAD** session.

The **Zoom** commands are especially important since they determine the relationship between the drawing and the drawing window. Like the zoom lens of a camera, the zoom commands are used to move your view closer or further away from the drawing. Although the objects appear to get bigger or smaller, their actual size does not change. This ability lets you see areas beyond your drawing's boundaries, or magnify in and see the finest details.

FastCAD permits ***dynamic panning***. This feature lets you pan or zoom your drawing view in real time, following the movements of your mouse.

A powerful feature to remember is the "**nestable**" nature of **FastCAD**'s ***zoom*** commands. A zoom command can be activated in the middle of another command. For instance, if you start a line command and realize you need to see more detail, you may invoke a zoom command and still finish your line.

Redraw

REDRAW recalculates your drawing from the database, then redraws the screen.

When you use editing commands, **FastCAD** erases affected entities by drawing over them in the background color before drawing the changed versions of the entities. This can leave apparent holes in other entities and in the grid. **REDRAW** cleans these holes up.

Redraws also occur automatically when you zoom, pan, resize the drawing window, change layer states, etc.

If you have a large or complex drawing that results in lengthy redraws, try using layers or sheets. Entities on ***hidden* layers** or **sheets** do not redraw.

Text equivalent: ***REDRAW***

Customizing Working Colors

FastCAD's default background color is black (#0). If you decide to use a new color, keep in mind that the defualt entity selection (marking) color is light gray (#16). If you choose a gray background color, use the MCOLOR command (type at command prompt) to change the marking color. Otherwise, selected entities will be difficult to see.

If you go with a light-colored background, you may want to then customize your palette so that more darker colors are available on the *Color bar*. Colors like yellow (#4) or cyan(#5) are difficult to see on a white background.

Background

BACKGROUND changes the background color for the current drawing. The new background applies only to the current drawing. Separate drawings in other drawing windows are not affected by this command. If the current drawing is visible in multiple drawing windows, all windows are updated to display the new background color.

After invoking the **BACKGROUND** command, the prompt reads "**Color value [dialog]:**". Type a color ID#, pick a color from the ***Color*** *bar*, or right-click to pick a color from the **256-color Select Color dialog**. **FastCAD** then redraws the drawing window in the new color.

Typically, this command affects screen display only-**FastCAD** leaves the background blank when you plot. Exception: if you check the **Paint background color** option in the ***Clipboard Specifications*** *dialog box* **[Clip > Options]**, the background color will be retained when the selected entities are pasted into other applications.

When drawings are opened as a reference or overlay drawings, or inserted as **XREF**s, the background color of the host drawing is honored.

Text equivalent: ***BKGND***

Change Origin

CHANGE ORIGIN redefines the location of coordinate (**0,0,0**) in the current drawing. Use **CHANGE ORIGIN** before saving a drawing for use as a part. **FastCAD** uses the (**0,0,0**) point as a handle to align with the insertion point when you later add the part to another drawing with **INSERT PART**, **PART ARRAY**, or **CIRC PART ARRAY** commands in the **INSERT** menu.

When you select **ORIGIN**, **FastCAD** displays a crosshairs cursor and the prompt reads "**Origin:**". Pick a point or by type numeric coordinates. (The center or lower-left corner of the part is usually a good choice.) **FastCAD** moves everything in your drawing so that the specified point is at coordinates (**0,0,0**).

UNDO does ***not*** work after the **CHANGE ORIGIN** command has been invoked.

Text equivalent: ***ORIGIN***

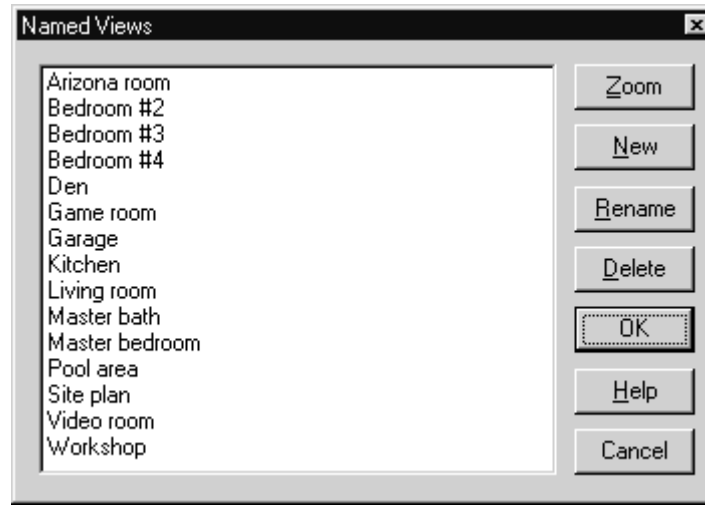
View Names

VIEW NAMES are a drawing navigation aid that allows you to quickly display a stored ***view*** of your drawing. A ***view name*** is for a single window. Creating a ***view name*** is akin to taking a ***snapshot*** of the drawing screen, yet you retain complete drawing and editing functionality when the ***view*** is recalled. Note: Layer properties (hidden, frozen, etc.) are not saved or altered when using ***view name***.

Each ***view name*** window retains its ***size***, ***name***, and ***zoom factor*** at the time it was saved, even if they are subsequently modified. If a ***named view*** window is closed, it will re-open when you select the **[View > Zoom Named]** command and you either type in the ***view name*** or select it from the ***'Named Views'*** *dialog.*

VIEW NAMES cannot be printed as such. However, named drawing window view can be printed by selecting the ***'Named View'*** in the ***'Print Drawing' dialog***.

When you select the **VIEW NAMES**, command the ***'Named Views'*** *dialog* appears.



To *display* a view Name Simply highlight the ***view name*** and click **ZOOM**.

To *create* a view name **Zoom** to the desired view in the actice drawing window. Click the ***New button*** and type the ***view name*** in the ***'Rename View'*** *dialog*. The view name can be up to 31 characters long and can contain spaces and punctuation marks. Press **OK** The ***view name*** will appear in the ***Title*** *bar* of the ***active drawing window***.

To *rename* a view Select a ***view name*** from the ***Named Views*** *dialog* and click the ***Rename*** *button.* Type a new ***view name*** in the ***Rename View*** *dialog*. The default ***view name*** appears in the ***Rename View*** *dialog.*

To *remove* a view name Select a ***view name*** and click the ***Delete*** *button.*

Text equivalent: ***VNAMES***

Set View Name

SET VIEW NAME allows you to give a name to any drawing window. The view's center, height, and width are stored with the name and can be recalled at any time using the **ZOOM NAMED** command **[View > Zoom Named]**.

View names are also helpful when used in conjunction with the **PRINT** command.

To ***name*** a drawing window,

1. If more than one window is open, use the mouse and click the ***Make Active Window*** *icon* at the right end of the desired window you wish to name (or rename).

   A red border appears around the ***active*** window. Zoom to the desired view. If only one drawing window is open it is the *active* window by default..

2. Select the **[View > Set View Name]** command. The prompt reads "**View name:**"
3. Type in a new *view name* and press **ENTER**. The *view name* can consist of up to 32 characters and contain spaces and punctuation. The *view name* appears in the ***Title*** *bar* of the *active* window.

To ***delete* a *view name*** use the **ZOOM NAMED** command, select the *view name* in the ***Named Views*** *dialog*, then click the ***Delete*** *button*.

Text Equivalent: ***SVNAME***

Dynamic Pan

DYNAMIC PAN allows you to navigate your drawing in real time. When **DYNAMIC PAN** is active, the entire drawing window responds to your pointer movements. You can scroll in any direction, and by holding down certain keys, you can dynamically zoom or change your **3D** view. If you pan the screen using the scroll bars, the screen will track with the scroll bar movement.

To start the ***dynamic pan*** mode, choose the ***Dynamic Pan*** *icon* on the ***Standard Icons*** *bar*.

If you have a three-button mouse or a wheel mouse, clicking the center button or the wheel switch will activate **DYNAMIC PAN** mode.If your mouse's third button or wheel switch is configured for another option (for example, double-click or the Start menu), you must re-configure it to its "**default**" behavior. **FastCAD** will not override the driver-configured function.

After you invoke the dynamic pan mode, **FastCAD** displays a pointing cursor. Just click an hold the left mouse button and move the mouse to drag the view within the *active* window. You can continuously specify the pan origin, which can be useful when panning across multiple screens. To re-specify the pan origin, release the mouse button and reposition the cursor. Then click-hold the left button and move the mouse again.

Remember that when you are using this command, you are never changing entity positions or sizes! The only changing element is your view of the drawing.

To end the Dynamic Pan/Zoom mode Right-click to finish dynamic panning. To cancel any view changes, **press ESC** or the "**U**" key before you right-click.

OtherDynamic Pan Options

FastCAD also lets you ***Zoom in*** or ***Zoom out*** in **DYNAMIC PAN** mode. For **3D** work, you can also ***yaw*** and ***pitch*** your views, as well as move in or move out.

If you press the "**H**" key anytime during ***dynamic pan***, **FastCAD** displays a popup Help window that describes the keystrokes required for each action:

During left-click-hold and drag:

Key down	Action
none	Pan view
SHIFT	Zoom in/out
CTRL	[3D] Yaw & Pitch about center
P	[3D] Pitch about center
Y	[3D] Yaw about center
R	[3D] Roll about center
ALT	[3D] Move forward and back
F	[3D] Move forward
B	[3D] Move back
H or ?	Help
ESC or U	Cancels this view change

If you are not creating **3D** drawings, it is advisable that you avoid using the available **3D** options. The view may rotate off the **XY** plane, making it seem as if part your drawing has disappeared. If you're view has been rotated "**off the screen**", click the ***Make view 2d*** *icon* (or run the **ZSPEC** command) and enter zero values for all three of the **View Rotation** settings.

Text equivalent: ***DYNPAN***

Pan View

P**AN VIEW** shifts your drawing under the drawing window (like *scroll bars* do, but with more precision).

To pan the view, select **[View > Pan View]**:

1. The prompt reads "**Pan from [displacement]:**". The "**Pan from**" point is used in conjunction with a second point you specify to move your view. You can:
   - Pick a start point in the drawing window, or type numeric coordinates.**FastCAD** displays a rubber-band cursor anchored at the specified point.
   - Right-click to enter a displacement value. The prompt reads "**X,Y** Displacement:" Enter in a numeric coordinates. The view will change by this displacement relative to the origin and the command will terminate.
2. The prompt reads "**Pan to:**". Pick the second point in a drawing window, using the stretching cursor as a visual guide.

   FastCAD moves your drawing so that the "**Pan from:**" point appears at the "**Pan to:**" point in the drawing window.

Text equivalent: ***PAN***

Zoom Window

ZOOM WINDOW lets you draw a selection rectangle or "**window**" to view part of your display in the drawing window. The specified zoom is always displayed in the current drawing window, even though you can specify the zoom box in another drawing window. The two corner points can even be in different drawing windows; the stretching cursor will show you if any pick is valid.

To zoom into a window, select **[View > Zoom Window]**:

1. The prompt reads "**First corner:**". To specify the first corner of the desired view, pick a point in the drawing window or by type in numeric coordinates.

   FastCAD changes the cursor to a stretching rectangle.
2. The prompt reads "**Opposite corner:**". To specify the opposite corner of the zoom window, use the stretching rectangular cursor as a visual guide and pick a point.

   FastCAD redraws your screen to show the part of your drawing that fits within the rectangle you drew.

Text equivalent: ***ZWIN***

Zoom Center

ZOOM CENTER moves the *center* of the drawing window to a new spot in the drawing. This command has only one prompt, so it's often more convenient than **PAN**. The specified zoom is always displayed in the current drawing window, even though you can specify the view center in another drawing window.

This command does not magnify the view.

To **ZOOM CENTER,** select **[View > Zoom Center]**:

. The prompt reads "**View center:**". To specify the center point of the new zoom, pick a point in a window, or type numeric coordinates. (**Tip:** Use numeric coordinates to shift the view center to an off-screen point.)

FastCAD moves your drawing view to center it on the point you selected, without changing the zoom factor.

Text equivalent: ***ZCEN***

Zoom In

ZOOM IN magnifies the view in the drawing window. (Entities in the drawing window appear larger, and you see less of your drawing.) As the view magnifies, the center view stays fixed.

Like all zoom commands, **ZOOM IN** changes the view within the current drawing window. It does not resize the drawing window, nor does it change the view in other drawing windows.

To **ZOOM IN**, select **[View > Zoom In]**:

The prompt reads "**Scale factor [n]:**". Type a number to indicate how much to magnify the view and press ENTER.

FastCAD multiples the current view dimensions by the supplied factor and zooms accordingly.

Text equivalent: ***ZIN***

Zoom Out

ZOOM OUT widens the view in the drawing window. (Entities in the drawing window appear smaller, and you see more of your drawing.)

Like all zoom commands, **ZOOM OUT** changes the view within the current drawing window. It does not resize the drawing window, nor does it change the view in other drawing windows.

To **ZOOM OUT**, select **[View > Zoom Out]**:

1. The prompt reads "**Scale factor [n]:**". Type a number to indicate how much to widen the view and press ENTER.

FastCAD multiples the current view dimensions by the supplied factor and zooms accordingly.

Text equivalent: ***ZOUT***

Zoom Width

ZOOM WIDTH tells **FastCAD** how much of your drawing to display in the drawing window, using the specified horizontal width.

. The prompt reads “**New view width:**”. Enter a value in current drawing units (you can include decimal or conventional fractions).

  FastCAD redraws the drawing window, horizontally sized to the specified number of units.

When you create a new drawing, **FastCAD** uses a default view width of 10.5 units, centered on coordinates (5,5).

Text equivalent: ***ZWID***

Zoom Height

ZOOM HEIGHT tells **FastCAD** how much of your drawing to display in the drawing window, using the specified vertical height.

. The prompt reads “**New view height:**”. Enter a value in current drawing units (you can include decimal or conventional fractions).

  FastCAD redraws the drawing window, vertically sized to the specified number of units.

Text equivalent: ***ZHGT***

Zoom Extents

ZOOM EXTENTS displays your entire visible drawing in the current window, shrinking or enlarging the view to fit. Entities on hidden layers are ignored.

This command is one of the most useful zoom commands available. To quickly **ZOOM EXTENTS**, click the ***Zoom Extents*** *icon* on the ***Standard icon*** *bar.*

TIP: When working with **FastCAD**, it is easy to get caught up in unnecessary details. Sometimes it is a good idea to take a step back and view the entire drawing. This command will also show if you have inadvertently drawn or moved something out of your expected drawing area.

Text equivalent: ***ZEXT***

Zoom Named

ZOOMED NAMED displays a named view you previously created using the **SET VIEW NAME** command **[View > Set View Name]**. When you invoke the command, **FastCAD** asks you to type in the name of the saved view, or you can right-click to display the ***Named Views*** *dialog box*. Select a name and click **OK**. FastCAD replaces the current view with that of the named view.

Text equivalent: ***ZNAME***

Zoom Text

ZOOM TEXT command zooms the view to center on drawing text that matches the text string you specify. The actual scale of the zoom depends on the height and length of the found text entity. **FastCAD** determines the extents of the text, then zooms to four times (4x) those values, centering on the extents.

The prompt reads "**Zoom to text pattern [prior]:**". Type in a text string to search, or right-click/press ENTER to accept the previous search string.

If **FastCAD** finds only a single match, it will zoom to center that text in the view. If more than one instance of the search string is found, **FastCAD** displays a dialog displaying the matching text entities. Select the desired target, or click **Cancel** to abort.

Use the **ZTEXTD** variant of this command to display the ***Zoom to Text*** *dialog*. The operation sequence is exactly the same as **ZTEXT**, except that the string is entered in the dialog and confirmed by clicking **OK**.

Note that the pattern match is case insensitive. Also, there are implied wildcards at the beginning and end of the search text you enter. For instance, "Street" matches "103rd Street NW" and "A street" and "STREETS".

The = character (equal sign) forces the search string to be an exact match of the entire text entity (not case sensitive). For instance "=street" matches "Street" or "STREET", but it does not match "3rd street" or "streetsweeper".

The ? character (question mark) matches anything except end-of-string (there must be a character to match). For instance, "103?C" matches "aa103xc".

Text equivalent: ***ZTEXT***

Zoom Workplane

ZOOM WORKPLANE places the ***active drawing window*** surface on the currently selected ***workplane***. The ***workplane*** must have been defined previously with the **[Specs > Define Workplane]** command or the **[Specs > Define WPlane Win]** command. This zoom function may be performed by clicking the **ZOOM WORKPLANE** icon in the **#d Icons bar**.

Text equivalent: ***ZWPLANE***

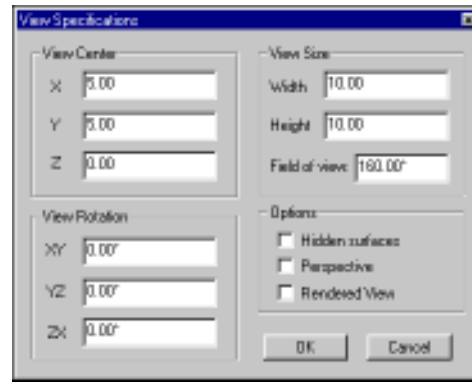
Zoom Last

ZOOM LAST redraws the drawing window with the previous zoom magnification and position, even if you have used other commands since the last zoom. If you repeat this command, it will toggle between the two last views.

Text equivalent: ***ZLAST***

Zoom Specs

Use **ZOOM SPECS** to parametrically define the view specifications of the active window. Many of these options can be specified by other means, such as with dynamic panning, or the control icons on each drawing window.



View Center Specifies the (x,y,z) coordinates of the drawing center's window. For **2D** drawings, the z value should always be zero.

View Rotation Specifies the rotation of the view in each standard plane. The rotation is specified in bearing notation. To rotate 2D drawings, modify the **XY** value only; the **YZ** and **ZX** values should always be zero.

View Size

Width The minimum horizontal length of the view in drawing units. This stays fixed even if you resize the drawing window. The actual width may be greater, depending on the specified **Height** and the aspect ratio of the drawing window.

Height The minimum vertical length of the view in drawing units. This stays fixed even if you resize the drawing window. The actual height may be greater, depending on the specified **Width** and the aspect ratio of the drawing window.

Field of view Specifies the angular width of the view in the rendered view. The viewpoint is determined by the specified field of view and window width.

Options

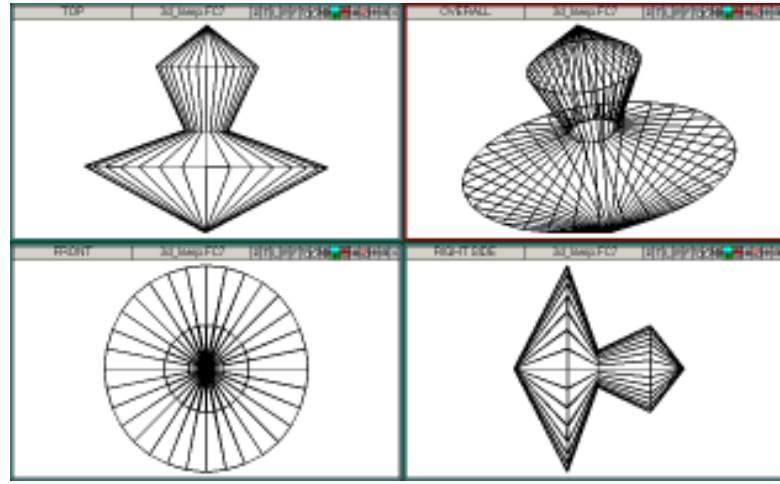
Hidden Surfaces Toggles between a wire-frame view and a "**solid**" hidden surface 3D view. This is the same as clicking the ***Hidden Line*** *icon* on any drawing window's title bar.

Perspective Toggles between a perspective and non-perspective view. The perspective view looks more realistic, as it more closely approximates the way you would see the entities in real life. This is the same as clicking the ***Perspective*** *icon* on any drawing window's title bar.

Rendered view Toggles between a regular wire-frame/hidden line view and a rendered "photo-realistic" view. Rendered views utilize light sources and shading to give added depth and realism to your 3D entities. This is the same as clicking the *Rendered View icon* on any drawing window's title bar.

Text equivalent: ***ZSPEC***

Standard 3D Views command



STD 3D VIEWS divides your screen into four windows and displays the entire (current) drawing in top, front, right side, and overall views as shown above. The overall view is from above the front right corner of the model, angled to display the entire drawing.

Text equivalent: ***ZSTD***

Chapter 8

Draw Commands

Adding 2d entities and objects to your drawing

The **Draw** commands add **2d** entities to your drawing. **3d** entities are discused in ***Chapter 10.*** As you might guess, these commands provide the essential building blocks of **FastCAD** . For most of the **Draw** commands, this chapter contains a step-by-step look at how to construct these entities using mouse or keyboard entry.

Keep in mind that when you create new entities, they acquire the *entity properties* you have specified as current. Entity properties include color, layer, line width, line style, and fill style. The current state of these is always displayed on the ***Status*** *bar*. Text and Dimension entities also have their respective current settings. These settings can be changed at any time, even in the middle of a draw command.

Don't fret if you draw some complex entities with the wrong properties. The edit commands in the next chapter tell you how to adjust those. For instance, you can draw a complex component all in green, then later change the colors of the entities using the proper edit commands.

FastCAD lets you create just about all the entities you might ever need to produce professional-level **CAD** drawings:

Lines Regular lines, double lines, and three types of constrained lines.

Circles Seven methods to draw circles, plus a donut command.

Arcs Four methods to draw arcs, plus chained arcs and a wedge command.

Text You can enter text or import it from a text file.

Paths and Polys Box, path, polygon, and regular polygon.

Waall Networks Intelligent **Walls** with auto-healing intersections

Splines and SPolys Complex curves and polygons using Bezier, cubic B-, and parabolic methods.

Ellipses Three methods, plus an elliptical arc command.

Point One simple method.

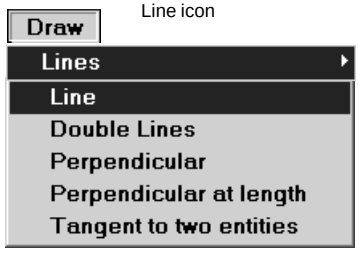
Fillets/Chamfers/Offsets Powerful cornering and duplicating methods.

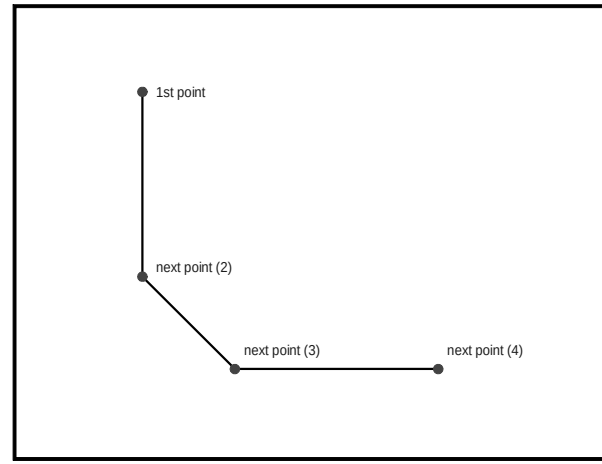
Arrows Create arrowed or double-arrowed lines, paths, arcs, and splines.

Multipolys Collects existing entities and turns them into a new single entity.

Line

Line icon





Lines sub-menu

When you select [**Draw > Lines**], a sidebar menu appears with five ways to draw lines. Use the cursor and left button to select the method you want to use.

When you see the "**Command:**" prompt after leaving a **LINES** command, you can either left/right-click or press **ENTER** to draw more lines of the same type, bypassing the sub-menu. To change to a different type of line, re-select [**Draw > Lines**] from the menu and select the desired type.

LINE ***lets you add 2D line entities to your drawing. You can draw a single line, or you can add a chain of lines that are connected endpoint to endpoint.***

The **LINE** command is listed both by itself and in the **[Draw > Lines]** sub-menu.

To draw a **line**, select **[Draw > Line]**.

1. ***The prompt reads*** 1st point:

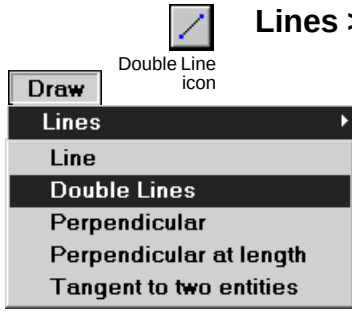
   To specify the line's starting point, pick a point, or type numeric coordinates.

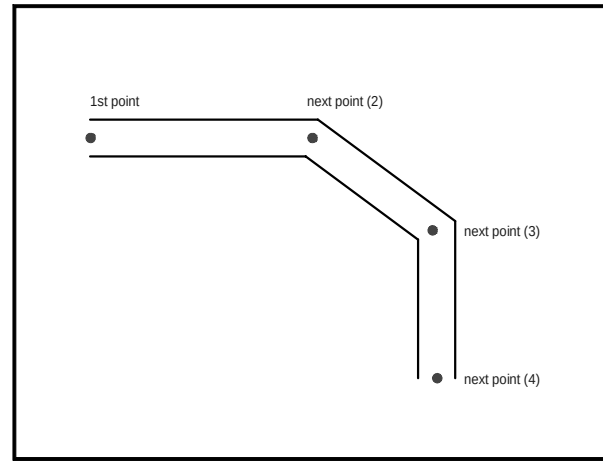
2. ***The prompt reads*** Next point [done]:

   The cursor becomes a rubber-band (stretching) line anchored to the first point you selected. To specify the line's end point, pick a point in the drawing window or type coordinates. **FastCAD** draws a line entity to replace the cursor, then stays in the **LINE** command so you can draw a series of connected lines. Right-click to end the command.

Text equivalent: ***LINE***

Lines > Double Lines





LINES > DOUBLE LINES ***adds parallel lines to your drawing. These parallel lines are separated (or offset) by a specified distance.***

To draw a **double line,** select **[Draw > Lines > Double Lines]**.

1. ***The prompt reads*** Separation [1.00000]:

   Separation is the distance between the double lines. To specify the separation:

   - Right-click or press **ENTER** to accept the default.
   - Type a numerical value in drawing units.
   - Pick two points in the drawing window.

2. ***The prompt reads*** First point:

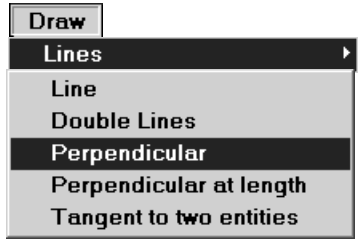
   To specify the starting point of the double line, pick a point or type numeric coordinates. The starting point remains fixed, even if you change handles (see next step).

3. ***The prompt reads*** Next point [SHIFT = Change handle]:

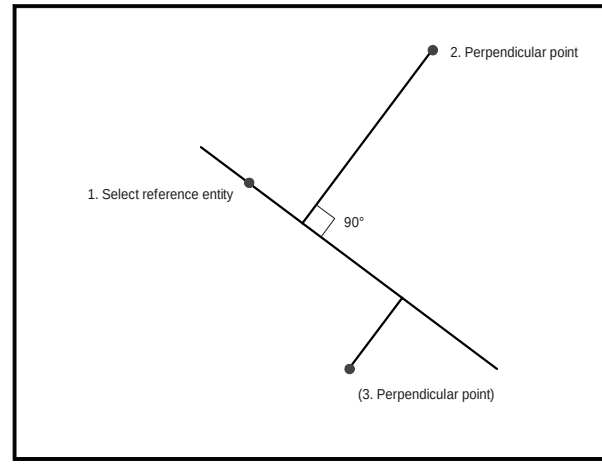
   You can change the line to which the cursor is anchored by pressing the SHIFT key, moving the mouse nearest the line you want to be the anchor, then releasing the SHIFT key. Specify the next point, using the new anchor as the "rubber-band cursor". Right-click to complete or end the command.

Text equivalent: ***DBLN***

Perpendicular Line icon



Lines > Perpendicular



LINES > PERPENDICULAR ***adds a line that is perpendicular to a selected reference entity.***

To draw a **perpendicular line**, select **[Draw > Lines > Perpendicular]**:

1. ***The prompt reads*** Reference entity:

   Pick a reference entity in your drawing. Lines you draw will be perpendicular to this reference entity; the reference entity is not changed in any way.

   If you pick a nonlinear entity such as an arc or ellipse, your lines will be tangentially perpendicular to the curve.

2. ***The prompt reads*** Perpendicular point:

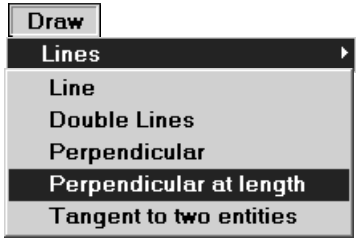
   This is the endpoint of the perpendicular line. Pick a point in the drawing window, using the dynamic cursors as an aid.

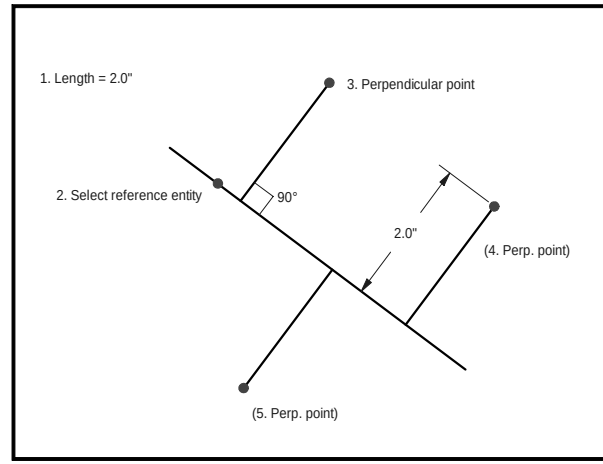
   FastCAD draws a ***perpendicular line*** entity from the ***endpoint*** to the ***reference entity*** (attaching at the nearest point). If the specified endpoint generates a perpendicular line that does not attach to reference entity, it is still drawn (it attaches to a projection of the reference entity).

   If you wish, continue to specify points that describe other perpendicular lines. Right-click to end the command.

Text equivalent: ***LNPRP***

Lines > Perpendicular at Length





LINES > PERPENDICULAR AT LENGTH ***draws a line of specified length perpendicular to a selected reference entity. To draw a perpendicular line without a pre-specified length, use the*** **LINE > PERPENDICULAR** ***command***.

To draw a **line** (**perpendicular at length**), select **[Draw > Lines > Perpendicular at length]**:

1. ***The prompt reads*** Perpendicular length [1.00000]:

   This value is the length of the line(s) that will be drawn during this command sequence. To specify the line length, you may:

   - Type a new length in drawing units, and press **ENTER**.
   - Pick two points on the screen to specify a distance value.
   - Right-click to accept the default or prior length.

2. ***The prompt reads*** Reference entity:

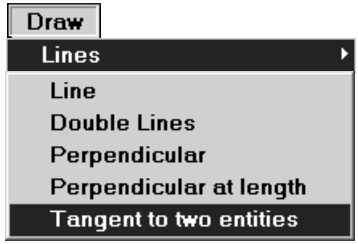
   Pick a reference entity. Lines you draw will be perpendicular to this reference entity; the reference entity is not changed in any way.

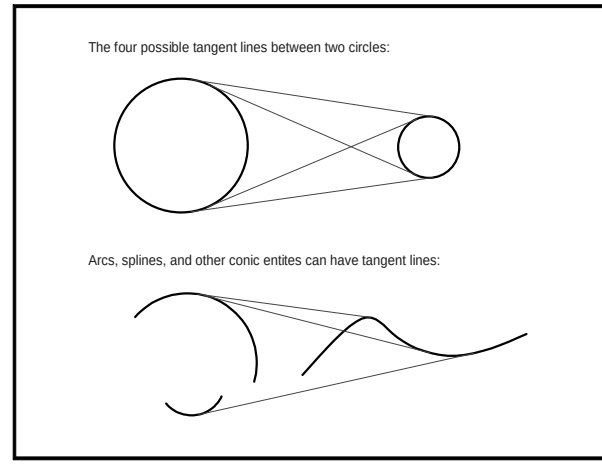
3. ***The prompt reads*** Perpendicular point:

   Using the dynamic cursors as a visual aid, pick a point anywhere along the line you are about to draw. The line is automatically drawn with the prescribed length, perpendicular to the reference entity. Continue selecting points, if desired. Right-click to end the command.

Text equivalent: ***LNPRPL***

Lines > Tangent to Two Entities





LINES > TANGENT TO TWO ENTITIES ***draws a line tangent to other entities. You can even draw a line tangent to two points on the same entity*** (**like a spline**).

NOTE: This command is not the same as the **TANGENT TO** modifier, which draws a line from any point tangent to another entity.

1. ***The prompt reads*** Reference entity:

   . Use the pick cursor to select a ***circle***, ***arc***, ***spline***, or other circular entity.

2. ***The prompt reads*** Reference entity:

   Again use the pick cursor to select a ***circle***, ***arc***, ***spline***, or other circular entity.

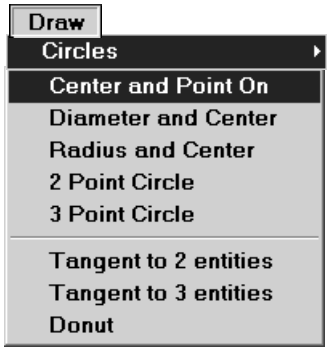
   The selection points determine how the tangent line will be drawn. For example, there are four possible tangents between any two circles or arcs: two on the outside and two that cross in the middle. **FastCAD** lets you draw all four — just select the circles or arcs on the sides you want to use for the tangent.

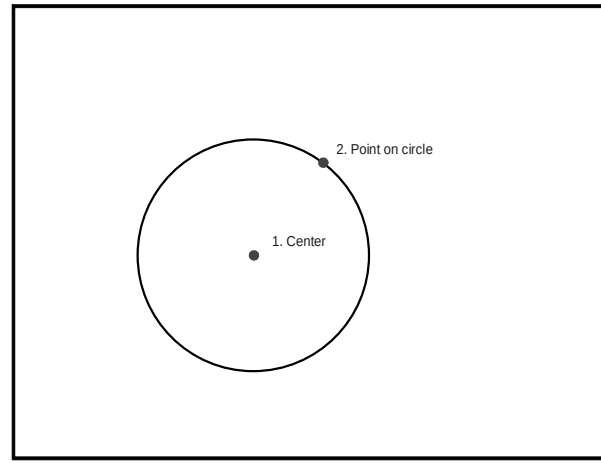
FastCAD draws the ***tangent line***.

Text equivalent: ***TANGENT***

Circle icon

Circles > Center and Point On





Circles sub-menu

When you select [**Draw > Circles**], a sub-menu appears with eight ways to draw circles. Use the cursor and left button to select the method you want. Use the auto-repeat feature to repeat a command.

FastCAD remembers the center and radius of each circle or arc you draw and offers them as default (prior) values the next time you draw a circle during the same work session. This feature lets you draw *concentric* circles. Right-click or press ENTER to accept the default, or enter a different value as explained under the individual circle commands.

CIRCLES > CENTER AND POINT ON ***draws a circle using a specified center point and circumference point.***

To draw a **center** and **point on circle**, select **[Draw > Circles > Center and Point On]**:

1. ***The prompt reads*** Center point[2.55882,5.94118]:

   To specify the center point:

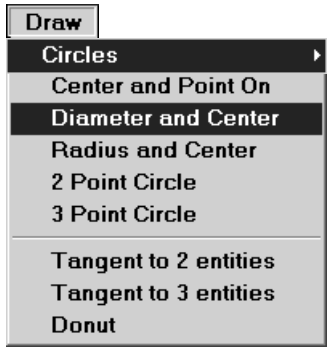
   - Right-click or press **ENTER** to accept the ***default center***;
   - Pick a new center point, or;
   - Type **2d** numeric coordinates and press **ENTER**.

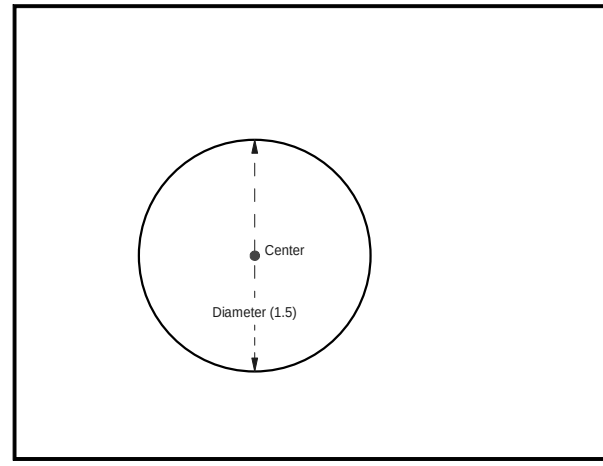
2. ***The prompt reads*** Point on circle:

   Use the dynamic cursor as a visual aid to pick a point on the screen. You can optionally combine the pick point with modifiers. When you pick a point, **FastCAD** draws the **circle** and the command terminates.

Text equivalent: ***CIRP***

Circles > Diameter and Center





Editing Circles

Each circle can be filled with any type of fill pattern.

To select a fill pattern for new circles, use FILL STYLES [Specs > Fill Styles].

Keep in mind that if you want a filled circle, it must have a LINE WIDTH of 0.0.

To change the pattern of existing circles, use CHANGE-FILL STYLE or EDIT in the *Edit menu.*

To change a circle's radius, use EDIT, DYNAMIC EDIT, or STRETCH in the *Edit menu.*

To turn a circle into an arc, use BREAK [Edit > Break].

CIRCLES > DIAMETER AND CENTER ***draws a circle using a specified diameter measurement and center point.***

To draw a **diameter** and **center circle**, select **[Draw > Circles > Diameter and Center**]:

1. ***The prompt reads*** Diameter [2.00000]:

   To specify the diameter:

   - Right-click or press **ENTER** to accept the default diameter;
   - Pick two points on the screen to specify a distance, or;
   - Type a new value for the diameter and press ENTER. For example, to draw a circle with a diameter of 4 units, type "4" and press **ENTER**.

   FastCAD displays a circular cursor with the specified diameter.

2. ***The prompt reads*** Center:

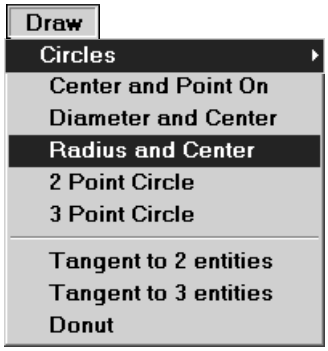
   Pick the desired center point or type numeric coordinates. **FastCAD** draws the ***circle*** with the specified ***diameter***.

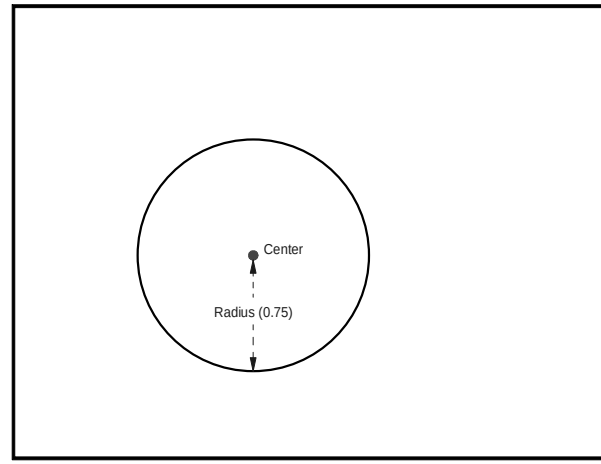
3. ***The prompt reads*** Center:

   Continue drawing ***circles*** with the same diameter by specifying new center points, or right-click to end the command.

Text equivalent: ***CIRD***

Circles > Radius and Center





CIRCLES > RADIUS AND CENTER ***draws a circle using a specified radius measurement and center point.***

To draw a **radius** and **center circle**, select **[Draw > Circles > Radius and Center]**:

1. ***The prompt reads*** Radius [1.00000]:

   To specify the radius:

   - Right-click or press **ENTER** to accept the ***default radius***;
   - Pick two points on the screen to specify a distance, or;
   - Type a new value for the radius and press ENTER. For example, draw a circle with a radius of 2 units, type "2" and press **ENTER**.

   FastCAD displays a circular cursor with the specified radius.

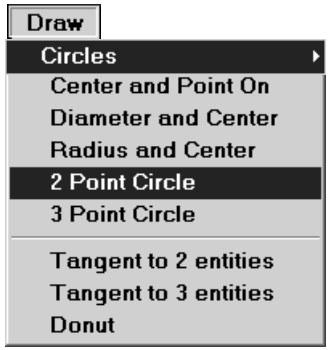
2. ***The prompt reads*** Center:

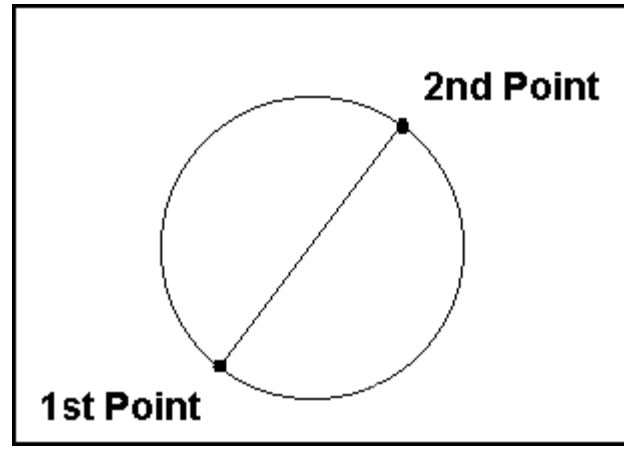
   Pick the desired center point or type numeric coordinates. **FastCAD** draws the circle with the specified radius.

3. You are again prompted for "**Center:**". Continue drawing circles with the same radius by picking new center points, or right-click to end the command.

Text equivalent: ***CIRR***

Circles > 2 Point Circle





CIRCLES > 2 POINT CIRCLE ***draws a circle using two specified points that define its diameter.***

To draw a **2 point circle** select **[Draw > Circles > 2 Point Circles]**:

1. ***The prompt reads*** 

   Pick a point or type numeric coordinates.

   FastCAD displays a stretching circular cursor, anchored at the 1st point you selected.

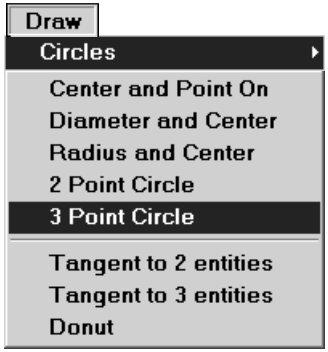
2. ***The prompt reads*** 

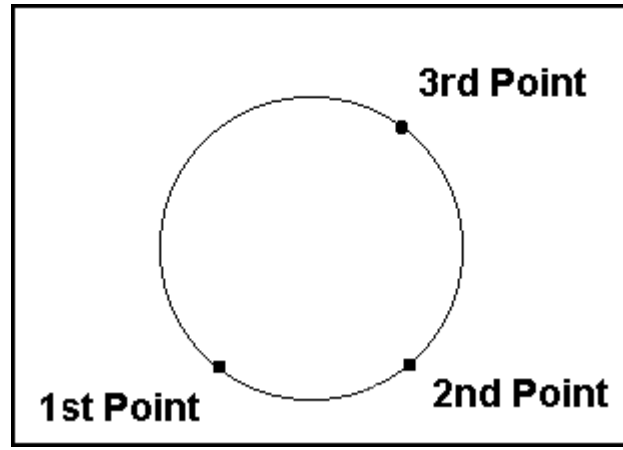
   To specify the opposite point, pick a point or type numeric coordinates. The two points specify the diameter of the ***circle***, centered about the midpoint distance between the two points.

FastCAD draws the **Circle** and terminates the command.

Text equivalent: ***CIR2***

Circles > 3 Point Circle





CIRCLES > 3 POINT CIRCLE ***draws a circle using three specified points on its circumference.***

To draw a **3 point circle**, select **[Draw > Circles > 3 Point Circle]**:

1. ***The prompt reads*** 1st point:

   Pick a point or type **2d** numeric coordinates.

2. ***The prompt reads*** 2nd point:

   Specify the ***2nd point*** as you did the 1st point.

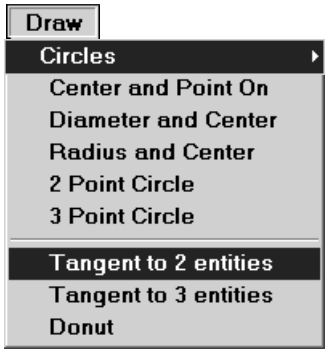
3. ***The prompt reads*** 3rd point:

   Specify the ***3rd point.***

FastCAD draws a **Circle** through the three points and terminates the command.

Note: If you specify three points in a straight line, a circle would be mathematically impossible, so **FastCAD** displays a message box: "**Can't create entity because: Three points are in a straight line.**" Click **OK** or press ENTER to clear the message.

Text equivalent: ***CIR3***



Circles > Tangent to 2 entities

TANGENT TO 2 ENTITIES draws a circle tangent to two other entities.

To draw a circle tangent to 2, select **[Draw > Circles > Tangent to 2 entities]**:

1. ***The prompt reads*** Tangent to entity:

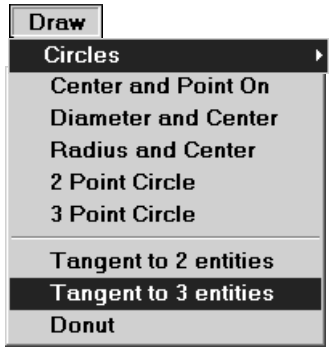
   Pick the target section (where you want the tangent circle to appear) of a circle or arc.

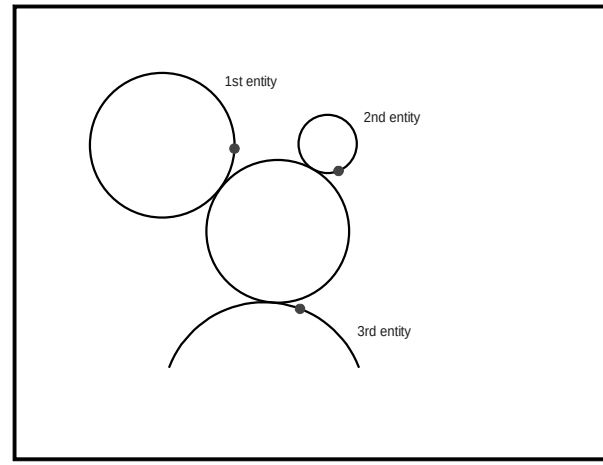
2.. ***The prompt reads*** Tangent to entity:

   Select another non-linear entity. FastCAD draws a circle in between the entities you picked.

Text equivalent: ***CIRTAN2***

Circles > Tangent to 3 entities





TANGENT TO 3 ENTITIES ***draws a circle tangent to three circles or arcs***.

To draw a **circle tangent to 3,** select **[Draw > Circles > Tangent to 3 entities]**:

1. ***The prompt reads*** Tangent to entity:

   Pick the 1st entity with the mouse.

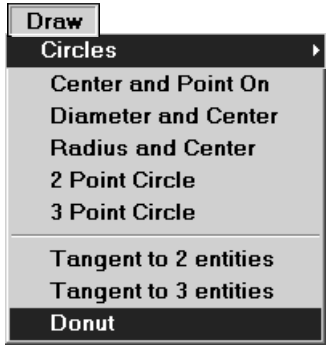
2. ***The prompt reads*** Tangent to entity:

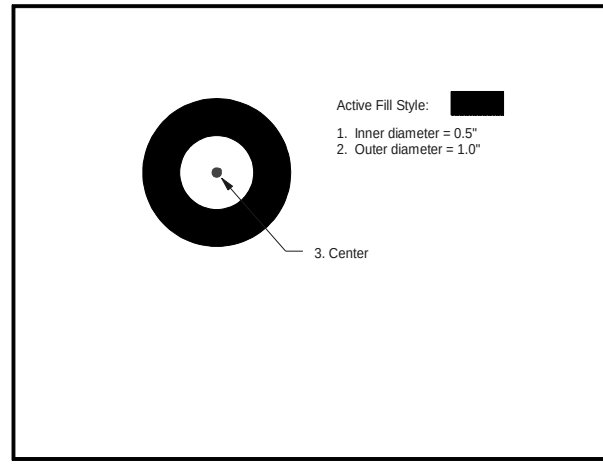
   Select the 2nd and 3rd entities and **FastCAD** draws a ***circle tangent*** to the three entities.

Often, multiple circles could be drawn, given three entities. You can help **FastCAD** draw the one you want by selecting the entities near their expected points of tangency.

Text equivalent: ***CIRTAN3***

Circle*s* > Donut





CIRCLES > DONUT a***utomatically builds a multipoly consisting of two concentric circles. The multipoly automatically fills with the current fill style, creating a donut-type shape.***

To draw a **donut** select **[Draw > Circles > Donut]**:

1. ***The prompt reads*** Inner diameter [0.50000]:

   To specify the ***diameter*** of the ***inner circle*** (**donut hole**), right-click to accept the default, or type a numeric value and press **ENTER**.

2. ***The prompt reads*** Outer diameter [1.00000]:

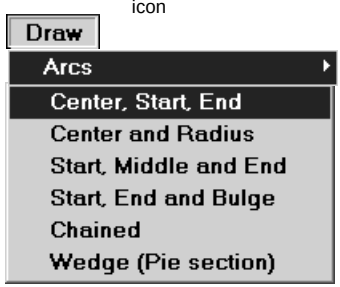
   To specify the ***diameter*** of the ***outside circle*** of the donut, right-click to accept the default, or type a numeric value and press **ENTER**.

3. The prompt reads "**Donut center:**". To specify the donut center, pick a point or type numeric coordinates.

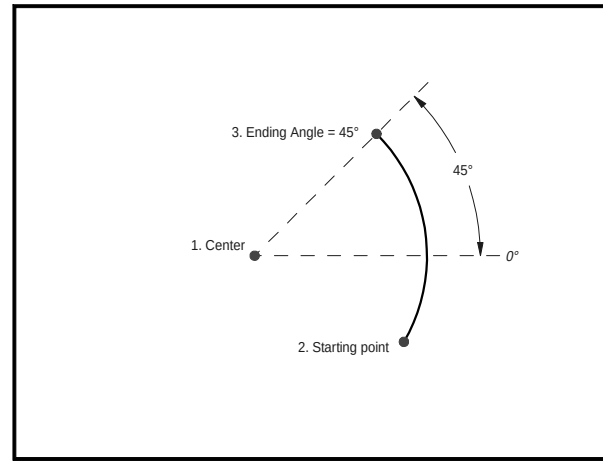
FastCAD draws the **Donut**t, hatching it with the ***current fill style***, and then terminates the command.

Text equivalent: ***DONUT***

Arc,Center Start & End icon



Arcs > Center, Start, End



Arcs sub-menu

When you select Arcs, a sub-menu appears with six ways to draw arcs. Use the cursor and left button to select the method you want.

FastCAD remembers the center and radius of the last circle or arc you drew and offers them as default or prior (suggested) values the next time you draw an arc during the same work session. This feature lets you draw *concentric* arcs. Right-click to accept the default or prior value or enter a different value as explained under the individual arc commands.

ARCS > CENTER, START, END ***draws an arc from a specified center point, start point*** (**which defines both the radius and the starting angle**), ***and end point***.

To draw a **center**, **start**, **end arc**, select **[Draw > Arcs > Center, Start, End]**:

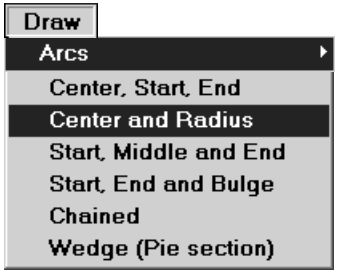
1. The prompt reads "**Center point [x,y]:**", where **x,y** is the prior or ***default center*** coordinate. To specify the ***arc center***:
   - Right-click or press **ENTER** to accept the default center;
   - Pick a new center point , or;
   - Type **2D** numeric coordinates and press **ENTER**.

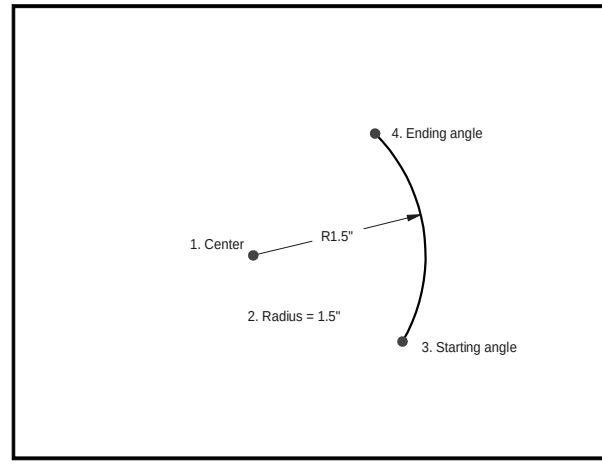
   FastCAD displays a ***circular*** dynamic cursor.
2. The prompt reads "**Arc starting point:**". The starting point tells FastCAD both the arc radius and its starting angle. To specify the starting point:
   - Pick a point using the stretching circular cursor, or;
   - Type numeric coordinates and press **ENTER**.
3. The prompt reads "**Arc ending point:**". To specify the ending angle, pick a point along the radial dynamic cursor, using the stretching arc cursor as a visual aid.

FastCAD draws the arc counterclockwise from the starting point to the ending point, and the command terminates.

Text equivalent: ***ARCS***

Arcs > Center and Radius





ARCS > CENTER AND RADIUS ***draws an arc from a center, radius measurement, starting angle, and ending angle***. Use this command when you want precise control over ***arc angles*** with a fixed ***radius***. For example, you could enter a radius of 1.5" and start an ***arc*** at exactly -30°, and end it at 45°.

To draw a **center** and **radius arc**, select **[Draw > Arcs > Center and Radius]**:

1. The prompt reads "**Center point [x,y]:**", where **x,y** is the prior or ***default center*** coordinate. To define the ***arc center***:

- Right-click or press **ENTER** to accept the default center;
- Pick a new center point, or;
- Type **2d** numeric coordinates and press **ENTER**.

FastCAD displays a circular dynamic cursor.

2. The prompt reads "**Radius [n]:**", where n is the prior or default radius distance. To specify the arc radius:

- Right-click or press ENTER to accept the default radius;
- Pick two points to specify a new radius distance, or;
- Type a new value for the radius and press ENTER. For example, draw a circle with a radius of 2 units, type "2" and press ENTER.

FastCAD displays a rubber-band cursor anchored at the center point.

3. The prompt reads "**Starting Angle [x°]:**", where **x** is the prior or ***default starting angle***. The s***tarting angle*** determines the first point of the arc. To specify the ***starting angle***:
   - Right-click or press **ENTER** to accept the default angle;
   - Pick a point, or;
   - Type a new value for the angle and press **ENTER**.

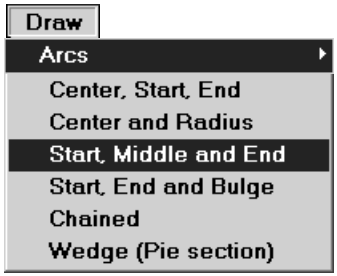
   FastCAD displays a dynamic ***arc*** cursor, anchored at the ***starting point***.

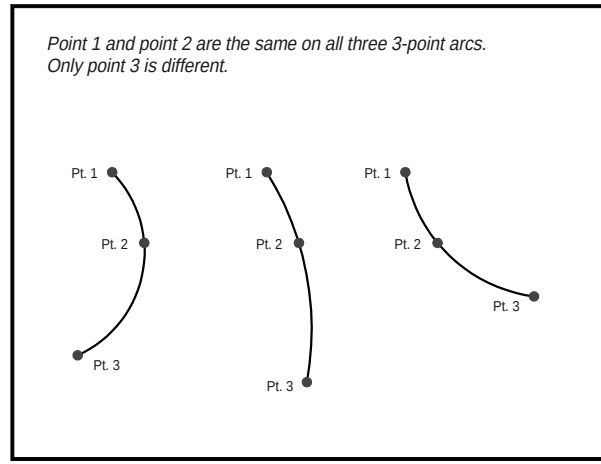
4. The prompt reads "**Ending angle [x°]**", where x is the prior or default ending angle. The ending angle tells **FastCAD** where the arc ends. Note that you need to specify the angle bearing, relative to the arc center, not the arc width. To specify the ending angle:
   - Right-click or press **ENTER** to accept the default angle;
   - Pick a point, or;
   - Type a new value for the angle and press **ENTER**.

FastCAD ***draws the arc counterclockwise from the starting angle to the ending angle***, and the command terminates.

Text equivalent: ***ARCR***

Arcs > Start, Middle and End





ARCS > START, MIDDLE AND END ***draws an arc from a specified starting point, point on the arc, and ending point.***

To draw a **start**, **middle** and **end arc**, select **[Draw > Arcs > Start, Middle and End]**:

1. The prompt reads "**1st point:**". This point locks down the starting endpoint. To specify the first point, pick a point or by type numeric coordinates.
2. The prompt reads "**2nd point:**". This point locks down an intermediate point on the arc. To specify the second point, pick a point or by type numeric coordinates.

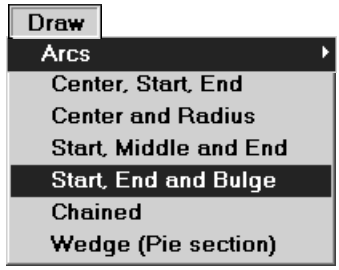
   The stretching arc cursor now passes through the second point, anchored at the first.
3. The prompt reads "**3rd point:**". This point locks down the endpoint the arc, and determines the arc size and shape. The stretching cursor previews the final arc form. To specify the third point, pick a point or type numeric coordinates.

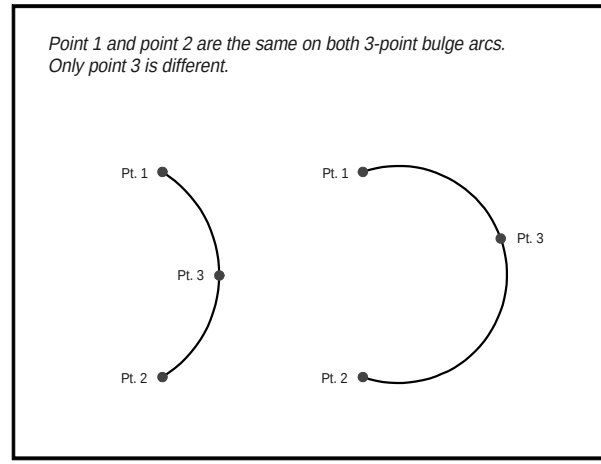
FastCAD draws the final arc from the first point, through the second, and to the third point.

Note: If you select three points in a straight line, an arc would be mathematically impossible, so **FastCAD** displays a message box: "Can't create entity because: Three points are in a straight line." Click **OK** or press ENTER to clear the message.

Text equivalent: ***ARC3***

Arcs > Start, End and Bulge





ARCS START, END AND BULGE works very much like **ARC3** except that you specify the ***arc's start*** and ***end points first***, and then ***a point on the arc***.

To draw a **start**, **end**, and **bulge arc**, select **[Draw > Arcs > Start, End and Bulge]**:

1. The prompt reads "**1st point:**". This point locks down the starting endpoint. To specify the first point, pick a point or type numeric coordinates.
2. The prompt reads "**2nd point:**". This point locks down the opposite arc endpoint. To specify the second point, pick a point or type numeric coordinates.

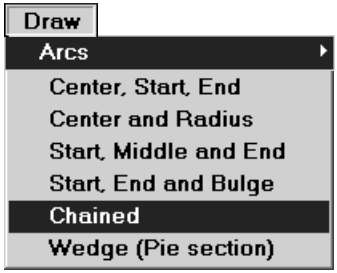
   The stretching arc cursor is anchored at the first and second points.
3. The prompt reads "**3rd point:**". This point locks down an intermediate point on the arc, and determines the arc size and shape (the "**bulge**"). The stretching cursor previews the final arc form. To specify the third point, pick a point or type numeric coordinates.

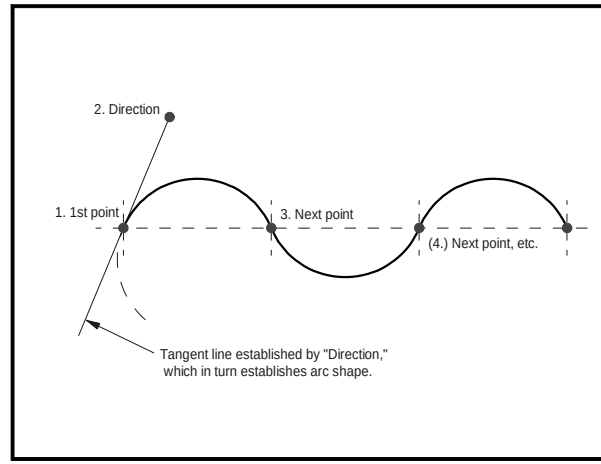
FastCAD draws the final **Arc** from the first point, through the second, and to the third point.

Please note: If you select three points in a straight line, an arc would be mathematically impossible, so **FastCAD** displays a message box: "**Can't create entity because: Three points are in a straight line.**" Click **OK** or press ENTER to clear the message.

Text equivalent: ***ARCB***

Arcs > Chained Arcs





ARCS > CHAINED ARCS ***draws a series of connected arcs***. The first point begins the ***first arc***. The second point determines a direction that is the tangent of the first point on the ***arc***. The next point sets the end of the first ***arc*** and defines the tangent and starting point of the next ***arc***.

To draw a series of **chained arcs**, select **[Draw > Arcs > Chained]**:

1. The prompt reads "**First arc start point:**". To specify the 1st point, pick a point or type numeric coordinates. **FastCAD** draws a rubber-band cursor anchored at the point you selected.
2. The prompt still reads "**First arc start point:**". The line defined by the 1st point and next point determines the direction (or tangent vector) for the first arc. To specify the nextpoint, pick a point or type numeric coordinates.

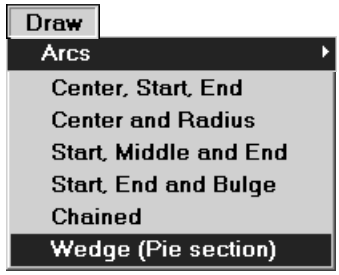
   FastCAD displays a dynamic ***arc*** cursor, anchored at the 1st point.
3. The prompt reads "**Next arc end point:**". This point determines the end of the first arc and the starting point of the next ***arc***. To specify this point, pick a point or type numeric coordinates.

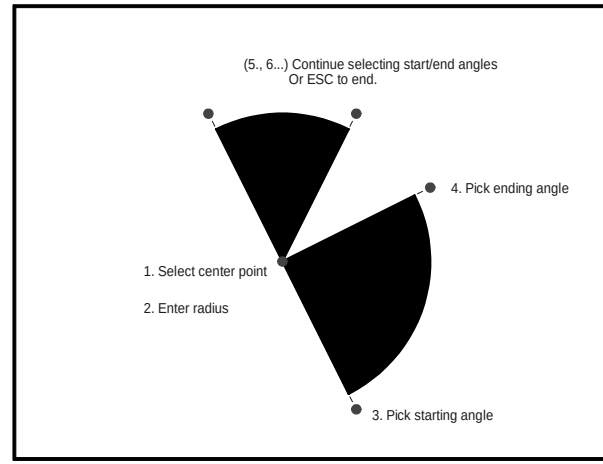
   FastCAD draws the first arc from the 1st point, constrained by the tangent vector, to the point just selected.
4. The prompt reads "**Next arc end point:**". You may continue drawing chained two-point arcs. Right-click to end the command.

Please note: If you select two arc endpoints whose bearing angle matches the direction vector, an arc would be mathematically impossible, so **FastCAD** draws a line between the two endpoints.

Text equivalent: ***CARCD***

Arcs > Wedge (Pie Section)





ARCS > WEDGE (PIE SECTION) ***is used to facilitate pie chart creation***. It does this by drawing ***filled arcs*** similarly to the **ARCR** command and providing means to quickly complete the ***circle***.

To draw a **Wedge**, select **[Draw > Arcs > Wedge (Pie section)]**:

1. The prompt reads "**Center point [x,y]:**", where **x,y** is the ***prior*** or ***default center*** coordinate. To define the ***arc center***:
   - Right-click or press **ENTER** to accept the default center;
   - Pick a new center point, or;
   - Type **2D** numeric coordinates and press **ENTER**.

   FastCAD displays a circular dynamic cursor.

2. The prompt reads "**Radius [n]:**", where n is the prior or ***default radius*** distance. To specify the ***arc radius***:
   - Right-click or press **ENTER** to accept the ***default radius***;
   - Pick two points to specify a ***new radius*** distance, or;
   - Type a new value for the ***radius*** and press **ENTER**. For example, draw a ***circle*** with a ***radius*** of 2 units, type "2" and press **ENTER**.

   FastCAD displays a rubber-band cursor anchored at the ***center point***.

3. The prompt reads "**Starting angle: [prior end, ESC=quit]**". Type an angle or select a point at the desired ***angle*** from the ***center***. If you right-click, the last ***ending***

angle of the preceding arc will be used. Pressing the **ESC** key will cancel the command.

FastCAD displays a stretching arc-shaped cursor, anchored at the starting point.

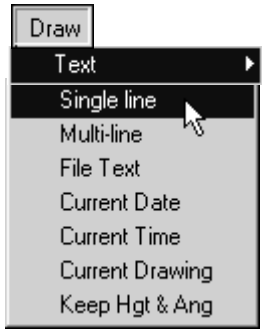
4. The prompt reads "**Ending angle [1st start, ESC=quit]:**". The ***ending angle*** determines the ***arc length of the wedge***. Type an ***angle*** or select a point when the arc looks the way you want. If you right-click, the ***wedge*** will be drawn in order to close the ***circle***. Do not accept this default on the first ***wedge*** unless you intend to draw a ***circle***.

   FastCAD ***draws the wedge counterclockwise from the starting angle to the ending angle***

5. **FastCAD** allows you to continue ***adding wedges***, prompting again for "**Starting angle:**" and "**Ending angle:**". Continue to draw a series of wedges with the same center and ***radius***. Note that the right button defaults allow you to easily draw adjacent ***wedges***. Press the **ESC** key to end the command.

Text equivalent: ***ARCW***

Single LineText icon



Single Line Text

SINGLE LINE TEXT ***adds a single line of lettering*** (**for labels, notes, etc.**) t***o your drawing.*** The text draws using the ***current font*** *and* ***text metrics*** *as specified in* ***Text Properties*** *dialog box****.***

To add **Single Line Text**, select **[Draw > Text > Single Line]**:

1. The prompt reads "**(CTRL=scale CTRL+SHIFT =rotate) Text:**". Start typing your text. The ***text*** appears both on the ***command line*** and on-screen at the ***cursor location***. As you type you can move the mouse to reposition the ***text***.

   This method of text input is intended for immediate text placement so it is unnecessary to press the ENTER key when you are done typing the line of text..

2. To insert your text, pick a point on the screen with a left-click

   In this mode you may **NOT** ***use the Zoom icons or dynamic zoom or pan to reposition the current view prior to*** **TEXT** ***placement***..

   If you need to zoom in or out befoure you place the line of **TEXT**, ***press*** **ENTER**. ***Now use the Zoom icons or dynamic zoom or pan to reposition the current view then place the*** **TEXT** ***with a left-click***.

   Typing a second single line of text and pressing **ENTER** ***does*** **NOT** ***automatically drop it directly below the previous line of text as earlier versions of*** **FastCAD** ***did***. Use the **MULTI-LINE TEXT** method to do this.

Note You may ***stretch*** or ***shrink*** the ***text prior to placemen*** *by* pressing the **CTRL** key and ***move the mouse slowly up or down***. This feature makes it very easy to ***fit text to existing objects without setting text height in advance***.

Stretching text this way does not reset **FastCAD**'s height value for new text - the next ***text*** entity you draw will still have the ***original current height***.

This ***dynamic cursor*** also allows you to ***rotate the text prior to placement***. Simultaneously hold down the **CTRL+SHIFT** keys and slowly move the mouse. ***The text rotates with your mouse movements***. This feature makes it very easy to ***align text without setting the text angle in advance***.

After placing the line of **TEXT**, you may use the **DYNEDIT** command by selecting **[Edit > Dynamic Edit]** and pick the **TEXT** ***at its baseline*** to ***reposition it, dynamicaly rotate***, ***stretch*** or ***shrink*** it.

You can draw **Single Line Text** at any ***height*** *and* ***angle***.

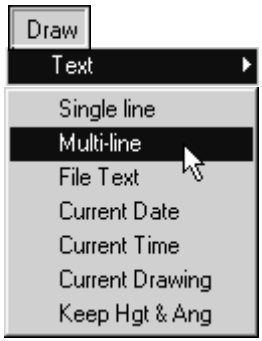
Text equivalent: ***TEXT***

Typing Special Characters

There may be occasions when you need to insert non-alphanumeric characters. In most cases, you can enter these by holding down the **ALT** key and typing the appropriate three- or four-digit code.

In most font sets, the **degree sign** (°) can be entered by typing **ALT+248**. The **Greek phi** (**Ø**) , often used to denote diameters, can be entered by typing **ALT+0216**. **Be sure to use the keyboard's number pad, not the top row number keys.**

You can determine the **ALT** key code combination for any character in a font set by using the Character Map program that's bundled with every copy of Windows.



Text > Multi Line Text

MULTILINE TEXT ***lets you add multiple lines of text at one time***. Because multiple lines will not fit on the command line (as in the **SINGLE LINE TEXT** command), **MLTI-LINE TEXT** require you to enter text in the ***Edit Text*** *dialog*.

To add **Multi-line Text** select **[Draw > Text > Multi Line]**:

1. **FastCAD** displays the ***Edit Text*** *dialog*. Enter your ***text*** in the ***Eext Text*** *window*. To start a new line, press the **ENTER** key.

   While typing, you can:

   - Use the keyboard ***arrow keys***, **HOME**, and **END** to move the cursor within the editing window;
   - Press **DELETE** ***to delete the character under the cursor***;
   - Press **BACKSPACE** ***to delete characters to the left of the cursor;***
   - Type special characters using the **ALT** key and the appropriate numbers from the key pad as explained in the side bar in this and the previous page.

   Click the ***Properties*** *button* if you wish to change any of the ***standard text properties***, such as ***height***, ***font***, or ***justification***.

2. When you are finished entering text, click **OK**.
3. The prompt reads "**Text origin:**". Your ***typed text*** is displayed on screen with a dynamic cursor. To position the text, pick a point on the screen, or type in numeric coordinates and press **ENTER**.

Note You may ***stretch*** or ***shrink*** the ***text prior to placemen*** *by* pressing the **CTRL** key and ***move the mouse slowly up or down***. This feature makes it very easy to ***fit text to existing objects without setting text height in advance***.

Stretching text this way does not reset **FastCAD**'s height value for new text - the next ***text*** entity you draw will still have the ***original current height***.

This ***dynamic cursor*** also allows you to ***rotate the text prior to placement***. Simultaneously hold down the **CTRL+SHIFT** keys and slowly move the mouse. ***The text rotates with your mouse movements***. This feature makes it very easy to ***align text without setting the text angle in advance***.

After placing the line of **TEXT**, you may use the **DYNEDIT** command by selecting **[Edit > Dynamic Edit]** and pick the **TEXT** ***at its baseline*** to ***reposition it, dynamicaly rotate***, ***stretch*** or ***shrink*** it.

Text equivalent: ***MLTEXT***

Typing Special Characters

There may be occasions when you need to insert non-alphanumeric characters. In most cases, you can enter these by holding down the **ALT** key and typing the appropriate three- or four-digit code.

In most font sets, the **degree sign** (°) can be entered by typing **ALT+248**. The **Greek phi** (**Ø**) , often used to denote diameters, can be entered by typing **ALT+0216**. **Be sure to use the keyboard's number pad, not the top row number keys.**

You can determine the **ALT** key code combination for any character in a font set by using the Character Map program that's bundled with every copy of Windows.

Edit Text dialog

The ***Edit Text*** *dialog* is displayed when **FastCAD** requests you to ***enter*** or ***edit text***.

Editing Text

Commands for editing text can be found in the chapter dealing with the ***Edit*** *menu*. But here's a summary of commands applicable to text:

To set properties for new text, use **TEXT PROPERTIES** [**Specs > Text Properties**].

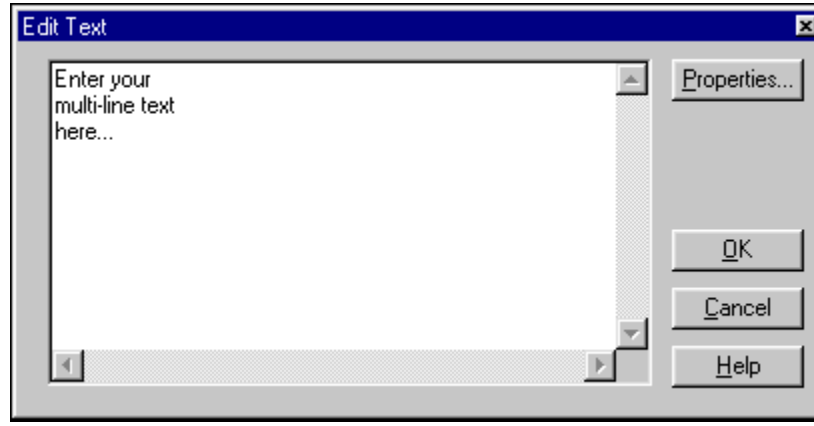
To change the properties of existing text, use **EDIT** or **CHANGE > TEXT PROPERTIES** in the ***Edit*** *menu*.

To insert text from an **ANSI** text file, use the **FILE TEXT** command [**Draw > Text > File Text**].

To modify the text value of a single text entity, use the **EDIT** command [**Edit > Edit**].

To change multiple, existing text entities to the same value, use the **CHANGE > TEXT** command [**Edit > Change > Text**].

To reposition existing text, use the **DYNAMIC EDIT** command [**Edit > Dynamic Edit**]. You can also use the dynamic cursor capabilities to **re-size** and **rotate** the text.



FastCAD displays ***Edit Text*** *window* into which you can type up to 256 characters on one line.

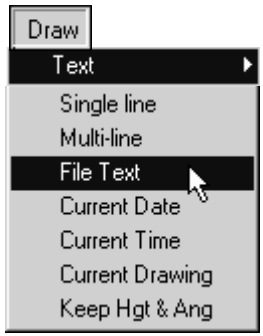
Edit window Type your ***text*** here. To create a new line, press the **ENTER** key.

While typing, you can:

- Use the keyboard ***arrow keys***, **HOME**, and **END** to move the cursor within the editing window;
- Press **DELETE** to ***delete the character under the cursor***;
- Press **BACKSPACE** ***to delete characters to the left of the cursor***;
- Type special characters using the **ALT** key.

Click **OK** when you are done entering text. **FastCAD** returns you to the drawing screen and continues with the current command.

Properties button Opens the ***Text Properties*** *dialog*, where you can change any of the standard ***text properties***, such as ***height***, ***font***, or ***justification***. Depending on the command calling this dialog, this button may not always appear.



Text > File Text

FILE TEXT inserts an **ANSI text file** into the ***active drawing***, using the ***current Text Properties.*** When you start this command, **FastCAD** displays the ***Text from file*** *dialog box*. This dialog is a variant of the standard ***Load File*** *dialog*, filtered to display only **.TXT** files. Though only **..TXT** files appear in the ***file list window***, the source file is not required to have a .txt extension.

To insert ***text from a text file***, select **[Draw > Text > File Text]**:

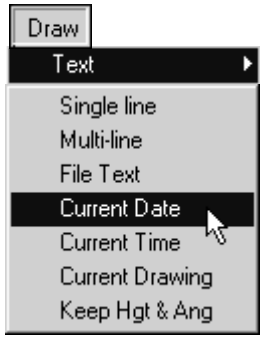
1. **FastCAD** displays the ***Text from file:*** *dialog*. Navigate to the appropriate folder containing the desired ***text file***, and then select the ***text file***. You can either double-click on the ***file name***, or select the ***file name*** and click **OK**.

   FastCAD displays a dynamic text cursor.

2. The prompt reads "**Text origin:**". To specify the ***insertion point***, pick a point on the drawing window, or type numeric coordinates and press **ENTER**.

FastCAD draws the **Text**.

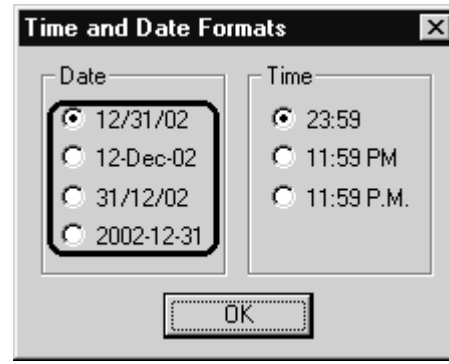
Text equivalent: ***TFILE***



Text > Current Date

The **[Draw > Text > Current Date]** command retrieves the **CURRENT DATE** from your computers system **CALENDAR** and allows you to place it in the current drawing as a piece of **TEXT**. It is a special entity called a **TEXT ITEM**. Title blocks are the usual place for placing the **CURRENT DATE** but it may be placed anywhere in the current drawing.

Setting the Date Format



To set the desired **DATE FORMAT**, select the **TIME AND DATE** command from the **SPECS** menu. You may also type **TDFMT** to open this dialog box. Click the button of the desired **DATE** and **TIME** format you want to use.

The Current Date Automatically Updates

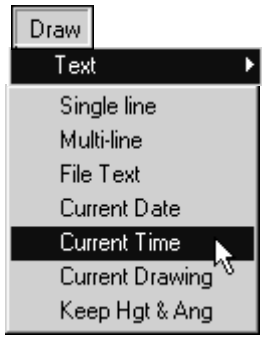
Each time you open the drawing the **CURRENT DATE** will automatically be updated. If you use a command such **DRAG** or **MOVE** to change the location of the **CURRENT DATE** text it will also be updated.

When you **PRINT** your drawing the **CURRENT DATE** text item will be updated so you may easily look at a printed drawing and know the **DATE IT WAS PRINTED**.

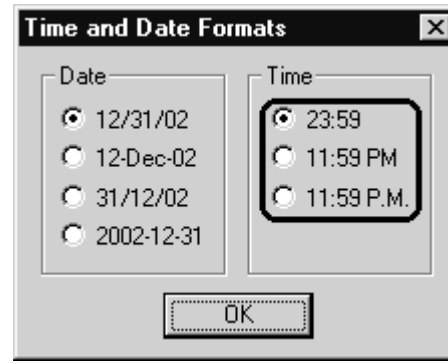
NOTE: The **CURRENT DATE** text item **MAY BE EXPLODED** into a normal piece of **TEXT**. If you use the **[Edit>Explode]** command and select the **CURRENT DATE** text item and **EXPLODE** it, it **WILL NOT UPDATE.** Use the **[Draw > Text > Current Date]** command again to place it as an updating **TEXT ITEM** in the drawing

Text equivalent: ***TDATE***

Text > Current Time



The **[Draw > Text > Current Time]** command retrieves the **CURRENT TIME** from your computers system **CLOCK** and allows you to place it in the current drawing as a piece of **TEXT**. It is a special entity called a **TEXT ITEM**. Title blocks are the usual place for placing the **CURRENT TIME** but it may be placed anywhere in the current drawing.



To set the desired **TIME FORMAT**, select the **TIME AND DATE** command from the **SPECS** menu. You may also type **TDFMT** to open this dialog box. Click the button of the desired **DATE** and **TIME** format you want to use.

The Current Time Automatically Updates

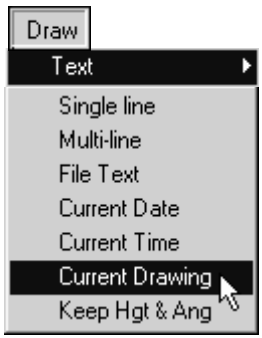
Each time you open the drawing the **CURRENT TIME** will automatically be updated. If you use a command such **DRAG** or **MOVE** to change the location of the **CURRENT TIME** text it will also be updated.

When you **PRINT** your drawing the **CURRENT TIME** text item will be updated so you may easily look at a printed drawing and know the **TIME IT WAS PRINTED**.

NOTE: The **CURRENT TIME** text item **MAY BE EXPLODED** into a normal piece of TEXT. If you use the **[Edit > Explode]** command and select the **CURRENT TIME** text item and **EXPLODE** it, it **WILL NOT UPDATE**. Use the **[Draw > Text > Current Time]** command again to place it as an updating **TEXT ITEM** in the drawing.

The **CURRENT DATE, CURRENT TIME** and **CURRENT DRAWING** text items will be exploded to standard text when the drawing is exported to **AutoCADâ .DWG** or .**DXF** file format.

Text equivalent: ***TTIME***



Text > Current Drawing

The **[Draw > Text > Current Drawing]** command retrieves the **CURRENT DRAWING NAME** complete with the **DRIVE** and **DIRECTORY PATH** where it is located and allows you to place it in the current drawing as a piece of **TEXT**. Title blocks are the usual place for placing the ***current drawing name*** but it may be placed anywhere in the current drawing.

NOTE: If you use the **[File > Save As]** command and save it with a different filename on a different drive, the **TEXT** will update to that new **DRIVE, DIRECTORY PATH** and **FILENAME** automatically as seen in the example below.

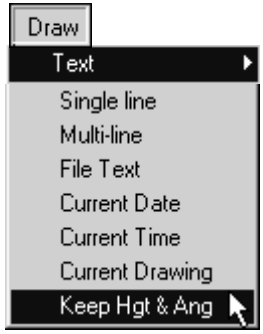
C:\FastCAD7\My_Drawing.FC7

Drive, Directory Path and Filename Automatically Update

If you use the **[File > Save As]** command and save it with a different filename on a different drive, the **TEXT** will update to that new **DRIVE, DIRECTORY PATH** and **FILENAME** automatically.

NOTE: The **CURRENT DRAWING** text item **MAY BE EXPLODED** into a normal piece of **TEXT**. If you use the **[Edit > Explode]** command and select the **CURRENT DRAWING** text item and **EXPLODE** it, it **WILL NOT UPDATE** when you save it to another drive or folder. Use the **[Draw > Text > Current Drawing]** command again to place it as an updating **TEXT ITEM**.

Text equivalent: ***TFILENAME***



Text > Keep Text Height and Angle

Use **KEEP TEXT HIEGHT AND ANGLE** ***to copy the height and angle specifications of the last text entered into the current text specs. You do not need to select the text-it automatically uses the most recently created text entity.*** New text you draw will use those settings until you change them again. This works even if the last inserted text was manipulated using dynamic cursors.

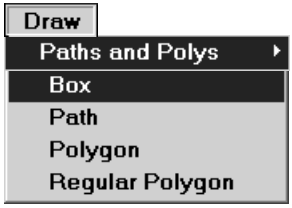
For instance, if you invoke the **TEXT** command and place the ***text*** using the **CTRL** and **CTRL+SHIFT** keys to ***dynamically scale*** and ***rotate*** the text, using the **KEEPTHA** command would ***copy the text's height and angle into the current settings***. You can verify that by invoking the **TSPEC** command and looking at the ***Text Properties*** *dialog* box by clicking the ***Text Properties*** *icon*.

KEEPTHA works similarly to the **KEEP LIKE** command, except that the **KEEP LIKE** command ***copies all text and entity specifications to the current settings***. **KEEPTHA** ***copies only the text height and angle***, leaving other properties such as layer, color, font, and justification unchanged.

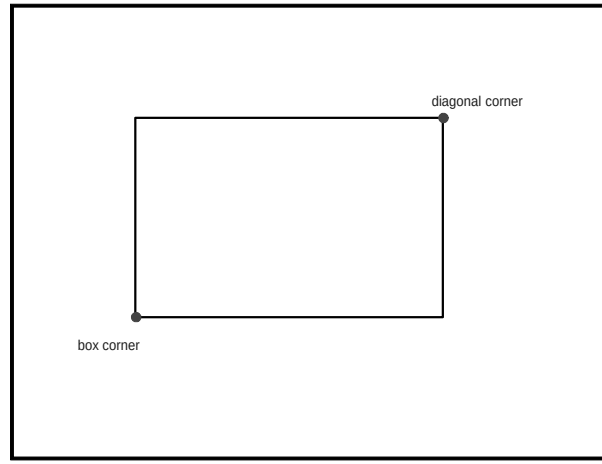
The macro commands **TSPECH** and **TSPECA** let you explicitly set the text height and angle, respectively.

Text equivalent: ***KEEPTHA***

BOX icon



Paths and Polys > Box



BOX ***is a shortcut command that enters a rectangular polygon entity***. **FastCAD** ***constructs the box using two corner points you specify.***

When you use the **BOX** command, the order in which you specify the corner points is disregarded. Certain ***modifiers***, such as **EPT**, **MID**, and **%,** utilize overall entity lengths. For **BOX** ***polygons***, the ***starting node is always the lower-left point***, and the ***midpoint is always the upper-right point***. For instance, the **EPT** (**endpoint**) of a non-rotated **BOX** ***polygon is always the lower-left node***.

To draw a **Box**, select **[Draw > Paths and Polys > Box]:**

1. The prompt reads "**First corner:**" To specify the ***first corner*** of the box, pick a point, or type numeric coordinates.

   The cursor changes to a stretching rectangle anchored at the ***first corner***.

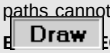
2. The prompt reads "**Opposite corner:**". To specify the ***opposite corner***, stretch the cursor and pick a point when the rectangle is the size and shape you want, or you type numeric coordinates.

FastCAD draws a **Box**-shaped ***polygon*** entity.

Text equivalent: ***BOX***

Paths and Polys sub-menu

Paths and **Polygons** share the common characteristics of being singular entities that are composed of multiple connected segments. **Polygons** have closure and **Paths** do not. **Polygons** can be filled. Boundaries created with simple paths cannot be filled.

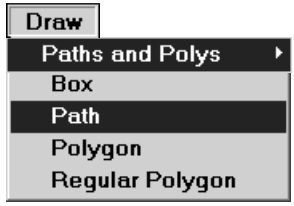


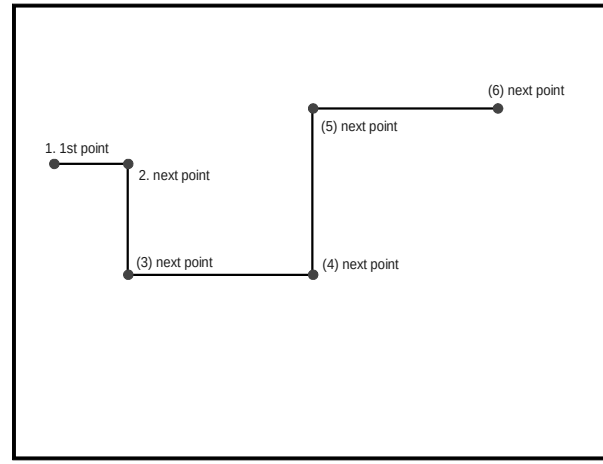
EGULAR POLYGON make it easy to draw these types of **Polygons**.

FastCAD stores the same information for all entities of this type. **Paths**, **Polygons**, **Smooth Polygons**, **Splines**, **Arrows** and **Double Arrows** are all defined by a series of nodes and can be edited in similar ways—for example, with **STRETCH** or **DYNAMIC EDIT** in the ***Edit*** *menu.*

PATH icon

Paths and Polys > Path





PATH ***draws a routed connection between two end points.*** A ***path*** looks like a s***eries of connected lines*** but ***is more like an open-ended polygon with at least 2 node points***. ***Paths*** are useful in mapping and other applications where connections are important.

To draw a **Path**, select **[Draw > Paths and Polys > Path]**:

1. The prompt reads "**1st point:**". Pick a point, or type numeric coordinates.

   FastCAD displays a rubber-band cursor anchored at the ***first point***.

2. The prompt reads "**Next point [done]:**". Pick the next node point on the path.

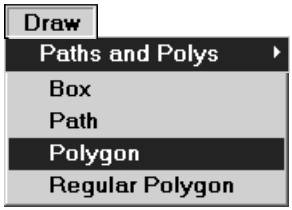
   FastCAD draws one segment of the ***path*** (in gray).

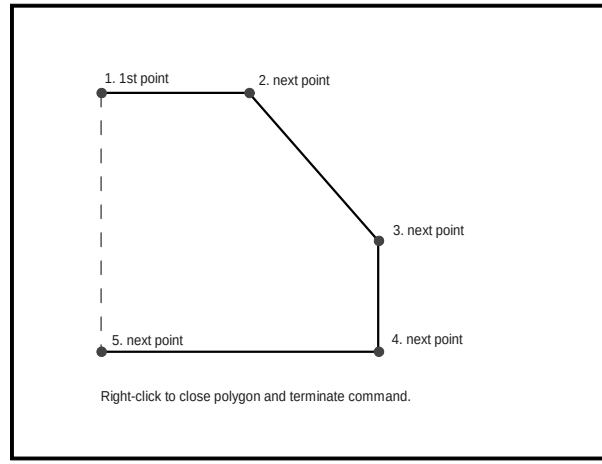
3. The prompt reads "**Next point [done]:**". If desired, continue to specify nodes. After placing the ending point of the ***path***, right-click to end the command.

Text equivalent: ***PATH***

POLYGON icon

Paths and Polys > Polygon





POLYGON ***draws a closed shape, optionally filled or crosshatched, with at least 3 sides.***

To draw a **polygon**, select **[Draw > Paths and Polys > Polygon]**:

1. The prompt reads "**1st point:**". When drawing ***polygons***, the 1st point is again used at the end of the command as the ***closing point***. To specify the ***first node*** (**corner**) of the ***polygon***, pick a point, or type numeric coordinates.

   FastCAD displays a rubber-band cursor, anchored at the ***1st point***.

2. The prompt reads "**Next point [done]:**". Specify the ***next poin***t.

   FastCAD draws one side of the ***polygon*** (in gray). A new rubber-cursor is displayed, anchored at the point just specified.

3. The prompt reads "**Next point [done]:**". Continue specifying nodes (each additional node forms another polygon side). If you are done adding nodes (**you must have at least three nodes**), ***right-click to finish the polygon***.

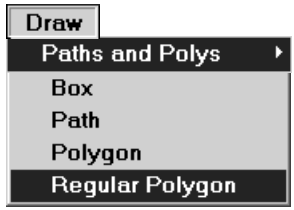
FastCAD ***draws the final line segment from the last point back to the first point, automatically closing the polygon***. The command terminates.

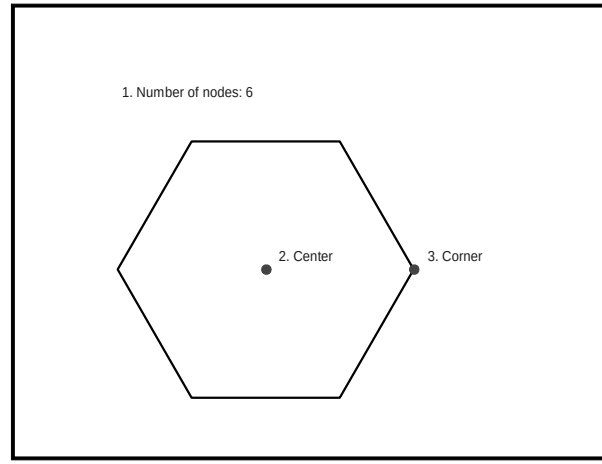
To draw a **box** (**rectangle**) **polygon**: **BOX**

To draw a **Regular Polygon**, such as a ***hexagon*** or ***octagon*** use **RPOLY**

Text equivalent: ***POLY***

Paths and Polys > Regular Polygon





REGULAR POLYGON is a shortcut command for entering ***regular polygon*** entities like ***equilateral triangles***, ***squares***, or ***hexagons***. You define the ***center*** and ***a corner of the polygon*** (the corner is a point on an imaginary circle that encloses the polygon).

To draw a **regular polygon**, select **[Draw > Paths and Polys > Regular Polygon]**:

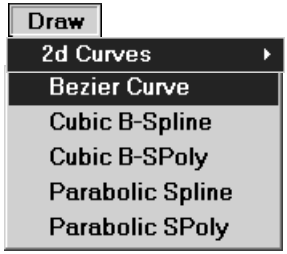
1. The prompt reads "**Number of nodes [n]:**". This determines the ***number of corners on the polygon***. Right-click to accept the default or ***prior number of sides***, or type a ***whole number value*** (for example, type "**4**" and press **ENTER** to draw a ***square***).
2. The prompt reads "**Center [x,y]:**". To determine the center point for the polygon:
   - Right-click or press **ENTER** to accept the default or prior center;
   - Pick a new center point, or;
   - Type numeric coordinates and press **ENTER**.

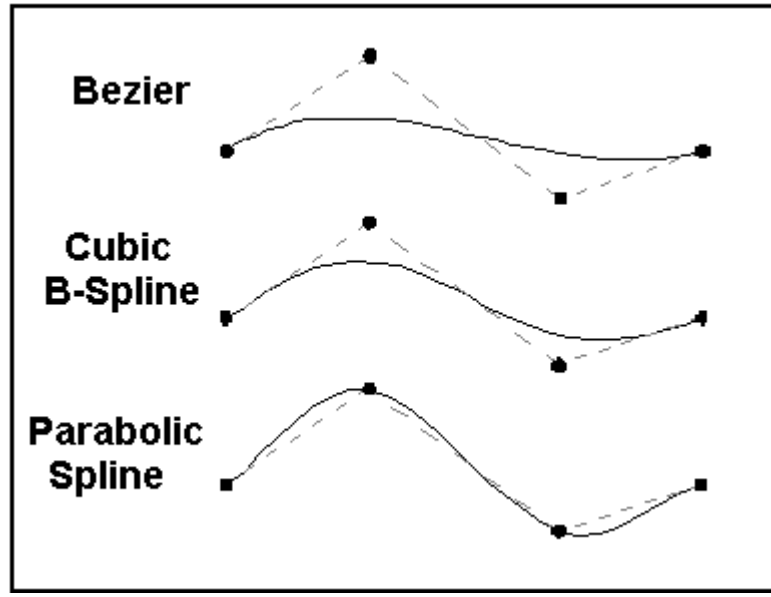
   FastCAD displays a dynamic cursor anchored to the ***center*** of the entity. A radial cursor marks the 1st corner.
3. The prompt reads "**Starting corner:**". This determines the ***final position of the regular polygon***. To specify the location of the 1st corner, pick a point, or by type numeric coordinates.

FastCAD draws the **Regular Polygon** and terminates the command.

Text equivalent: ***RPOLY***

2d Curves > Bezier Curve





2d Curves sub-menu

Splines are smooth curves, kind of like the French curve template in **FastCAD**'s toolbox. Bezier Curve, Cubic B-, and Parabolic type curves can be drawn. Smooth polygons (SPolys) are closed entities that can be hatched with a desired fill style.

The supporting mathematics behind the construction of these curves is well beyond the scope of this manual. However, quick experimentation will help determine which method works best for the type of complex curve you wish to create.

2D CURVES > BEZIER CURVE draws an ***open smooth curve***, determined by a ***first point***, a ***last point***, and ***two control points*** called a ***frame***. The curve touches the first and last points and follows an average path among the other two nodes. ***A Bezier Curve must have exactly 4 nodes.***

To draw a **Bezier curve**, select **[Draw > 2d Curvess > Bezier Curve]:**

1. The prompt reads "**1st point:**". The **Bezier** starts exactly at this point. To specify the starting point, pick a point, or type numeric coordinates.

   FastCAD displays a rubber-band cursor, anchored at the ***first point***.

2. The prompt reads "**Last Point:**". The ***Bezier*** ends exactly at this point. Specify the last point of the Bezier using the same manner as the 1st point.

   FastCAD displays a dynamic cursor showing the ***Bezier Curve*** and the frame (both in gray).

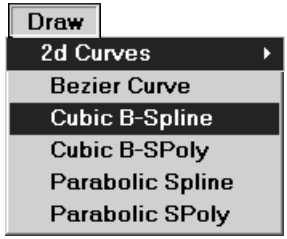
3. Specify the two control nodes ("**1st middle point:**" and "**2nd middle point [optional]:**") to finish the command. The **Bezier** does not pass through these points, but these points do control the shape of the ***spline***. You may right-click at the "**2nd middle point [optional]:**" propmt or pick a point.

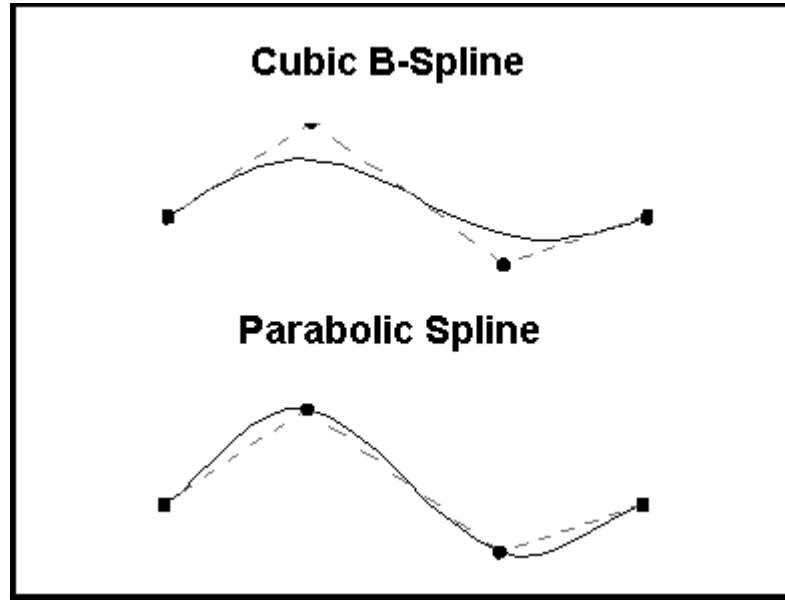
FastCAD draws the **Bezier Curve** and terminates the command.

Text equivalent: ***BEZ***

2d Curves > Cubic B-Spline

SPLINE icon





2d CURVES > CUBIC B-SPLINE (smooth polyline) draws an open smooth curve defined by a series of node points called a frame. The curve touches the first and last points and follows an average path among the other nodes. A spline must have at least 3 nodes.

To draw a **Cubic B-Spline**, select **[Draw > 2d Curves > Cubic B-Spline]**:

1. The prompt reads "**1st point:**". To specify the first spline node, pick a point, or type numeric coordinates.

   FastCAD displays a rubber-band cursor, anchored at the 1st point.

2. The prompt reads "**Next point [done]:**". Specify the second node in the spline.

   FastCAD displays a dynamic cursor showing both the curve and the frame.

3. The prompt reads "**Next point [done]:**". Continue to specify nodes as desired. Right-click to end node selection.

FastCAD draws the **Cubic B-Spline** and terminates the command.

Text equivalent: ***SPLINE***

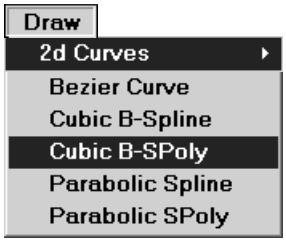
Frames

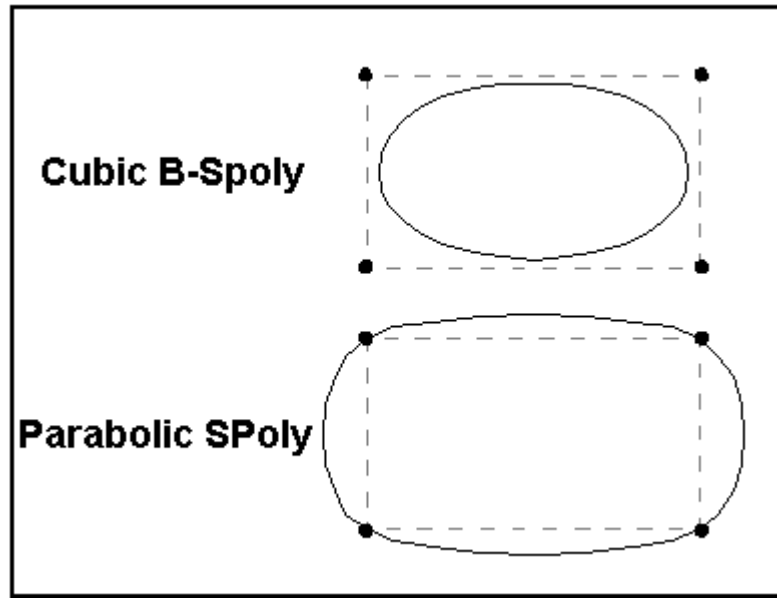
In FastCAD, frames apply exclusively to the supporting node network of a spline or SPoly. The frame is shown as connected line segments.

You can toggle frame display on or off using the TOGGLE FRAMES command [Specs > Toggle Frames] or by typing CTRL+F.

Frames also display when you use DYNAMIC EDIT to move, insert, or delete nodes from a **SPLINE** or **SPOLY**.

2d Curves > Cubic B-SPoly





CURVES > CUBIC B-SPOLY (**smooth polygon**) ***draws a closed, curved shape defined by a series of node points called a frame***. The curve doesn't touch the nodes. A smooth polygon must have at least 3 nodes. Each **SPoly** can be filled or crosshatched.

To draw a **Cubic B-SPoly**, select **[Draw > 2d Curves > Cubic B-SPoly]**:

1. The prompt reads "**1st point:**". To specify the first point, pick a point or type numeric coordinates.

   FastCAD displays a rubber-band cursor, anchored at the 1st point.

2. The prompt reads "**Next point [done]:**". Specify the next node in the spoly.

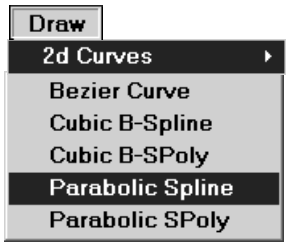
   FastCAD displays a dynamic cursor showing both the curve and the frame.

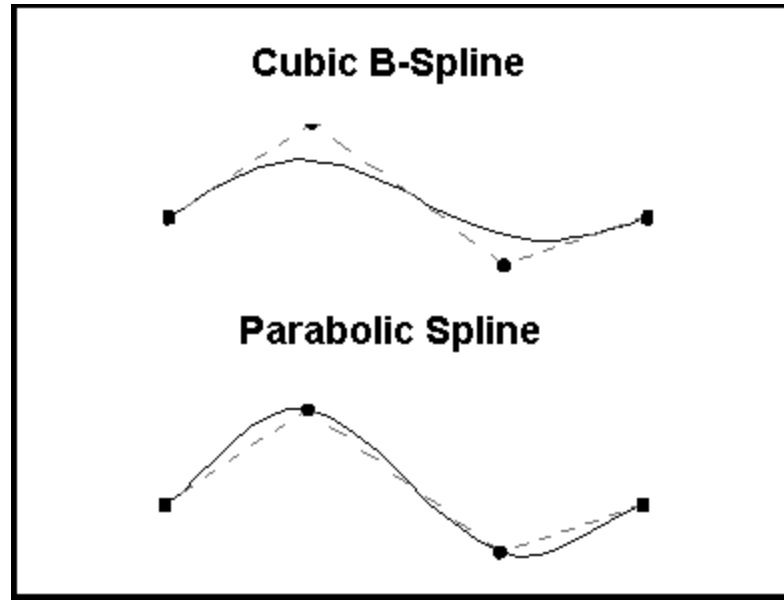
3. The prompt reads "**Next point [done]:**". Continue specifying nodes as desired. Right-click to end node selection.

FastCAD draws the **SPoly** by constructing the final frame segment from the last point back to the first point, automatically closing the smooth polygon. The command terminates.

Text equivalent: ***SPOLY***

2d Curves > Parabolic Spline





PARABOLIC SPLINE (smooth polyline) draws an open, curved shape defined by a series of node points called a frame. The curve draws through all the nodes. A parabolic spline must have at least 3 nodes.

To draw a **Parabolic Spline**, select **[Draw > 2d Curves > Parabolic Spline]**:

1. The prompt reads "**1st point:**". To specify the first spline node, pick a point or type numeric coordinates.

   FastCAD displays a rubber-band cursor, anchored at the 1st point.

2. The prompt reads "**Next point [done]:**". Specify the second node in the spline.

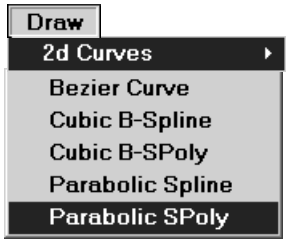
   FastCAD displays a dynamic cursor showing both the curve and the frame.

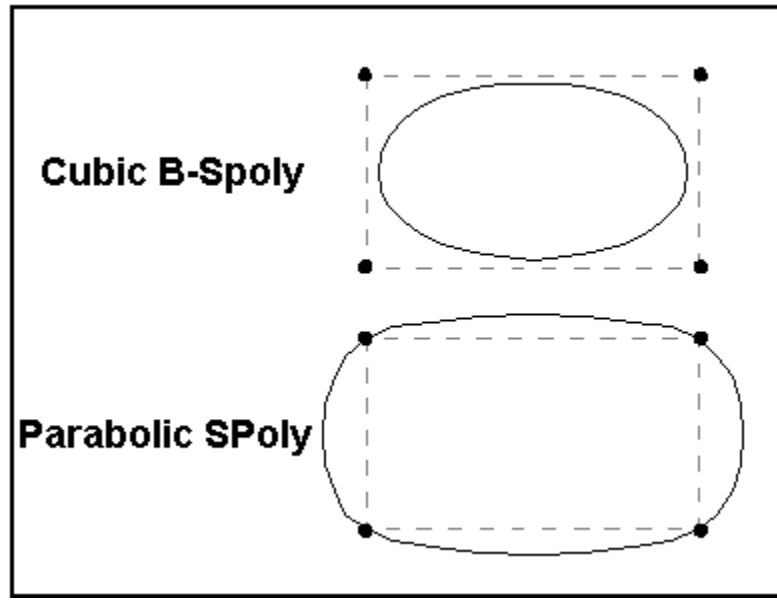
3. The prompt reads "**Next point [done]:**". Continue to specify nodes as desired. Right-click to end node selection.

FastCAD draws the **Parabolic Spline** and terminates the command.

Text equivalent: ***PSPLINE***

2d Curves > Parabolic S Poly





2D CURVES > PARABOLIC SPOLY (**smooth polygon**) ***draws a closed, curved shape defined by a series of node points called a frame***. The curve draws through all the nodes, using the same mathematical definition as a parabolic spline. A ***smooth polygon*** must have at least 3 nodes. Each smooth polygon can be solid filled or crosshatched.

To draw a **Parabolic SPoly**, select **[Draw > 2d Curves > Parabolic SPoly]**:

1. The prompt reads "**1st point:**". To specify the first point, pick a point or type numeric coordinates.

   FastCAD displays a rubber-band cursor, anchored at the 1st point.

2. The prompt reads "**Next point [done]:**". Specify the next node in the **SPoly**.

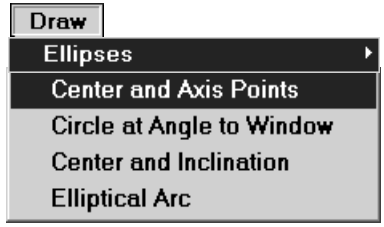
   FastCAD displays a dynamic cursor showing both the curve and the frame.

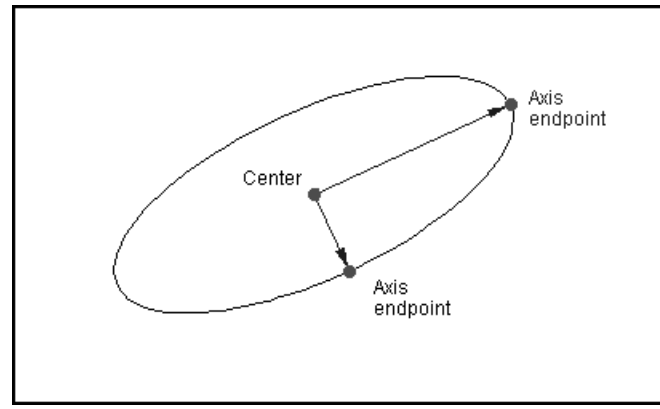
3. The prompt reads "**Next point [done]:**". Continue specifying nodes as desired. Right-click to end node selection.

FastCAD draws the **Parabolic SPoly** by constructing the final frame segment from the last point back to the first point, ***automatically closing*** the ***smooth polygon***. The command terminates.

Text equivalent: ***PSPOLY***

Ellipses > Center and Axis Points





CENTER AND AXIS POINTS ***draws an ellipse using a specified center point and two special points on its circumference – the ends of its minor and major axes. Of all possible points on the circumference, the ends of the major axis are farthest from the center, while the ends of the minor axis are closest.***

Ellipses sub-menu

When you select **ELLIPSES**, a sub-menu appears with four ways to draw ellipses and elliptical arcs. Select the desired method.

An ellipse has a **major axis** and a **minor axis**. The **major axis** is the longer diameter of an **ellipse**. The minor axis is the shorter diameter.

You can draw ellipses at any angle. The angle of the **major axis** is called the inclination of the **ellipse**.

To draw a **center** and **axis point ellipse**, select **[Draw > Ellipses > Center and Axis Points]:**

1. ***The prompt reads*** Center point[2.55882,5.94118]:

   To specify the ellipse center:

   - Right-click or press ENTER to accept the default center;
   - Pick a point , or;
   - Type numeric coordinates and press ENTER.

   FastCAD displays a rubber-band cursor, anchored at the center.

2. ***The prompt reads*** Major axis endpoint:

   Specify a point that defines an axis endpoint. The point you select can describe either the major or minor axis.

   FastCAD displays a stretching ellipse cursor.

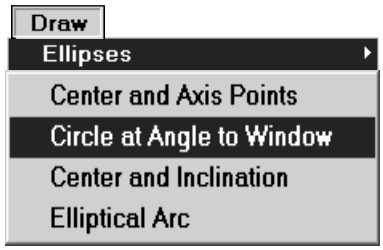
3. ***The prompt reads*** Minor axis point:

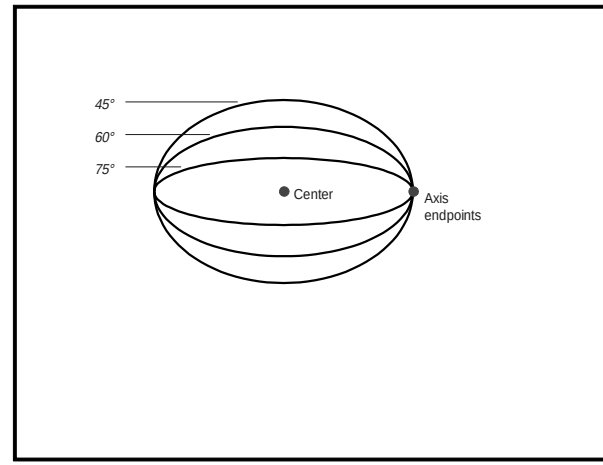
   Pick a point when the dynamic cursor displays the size and shape you want.

FastCAD draws the ***ellipse*** and terminates the command.

Text equivalent: ***ELIPP***

Ellipses > Circle at Angle to Window





ELLIPSES > CIRCLE AT ANGLE TO WINDOW d***raws an ellipse as if it were a circle at an angle to the plane of your screen***. The greater the angle, the more squashed the ***ellipse***. For example, an ***ellipse*** at 0° to the screen looks exactly like a ***circle***, and its major and minor axes are the same. An ***ellipse*** at 90° looks like a straight line. In an ***ellipse*** angled 60°, the major axis is exactly twice as long as the ***minor axis***.

Note: Don't confuse angle to the screen with the inclination of the ***ellipse***.

To draw a **circle at angle to window**, select **[Draw > Ellipses > Circle at Angle to Window]:**

1. The prompt reads "**Center point [x,y]:**". To specify the center of the ellipse:

   - Right-click or press **ENTER** to accept the default or prior center;
   - Pick a point, or;
   - Type numeric coordinates and press **ENTER**.

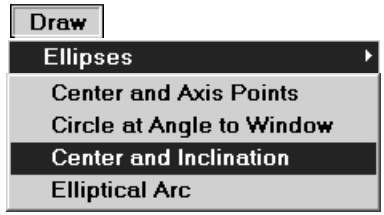
   FastCAD displays a dynamic circular cursor, anchored at the center.

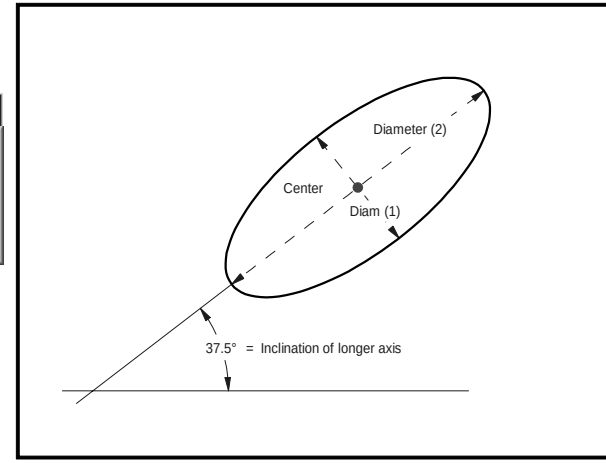
2. The prompt reads "**Major axis endpoint:**". The axis endpoint determines the ellipse's inclination. Specify a point that defines an axis endpoint.
3. The prompt reads "**Angle to screen:**". Specify the "***angle to window***" by typing an angle from 0 to 90 degrees, then press ENTER.

FastCAD draws the **ellipse** and terminates the command.

Text equivalent: ***ELIPC***

Ellipses > Center and Inclination





ELLIPSES > CENTER AND INCLINATION ***draws an ellipse using numeric values for its inclination and its major and minor diameters.***

To draw a **center** and i**nclination ellipse**, select **[Draw > Ellipses > Center and Inclination]:**

1. The prompt reads "**Center point [x,y]:**". To specify the center of the ***ellipse***:
   - Right-click or press **ENTER** to accept the default or prior center;
   - Pick a point, or;
   - Type numeric coordinates and press **ENTER**.

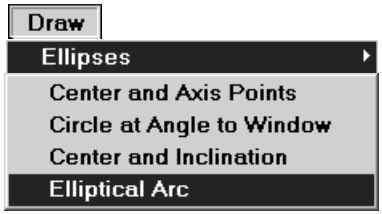
   FastCAD displays a rubber-band cursor, anchored at the center.

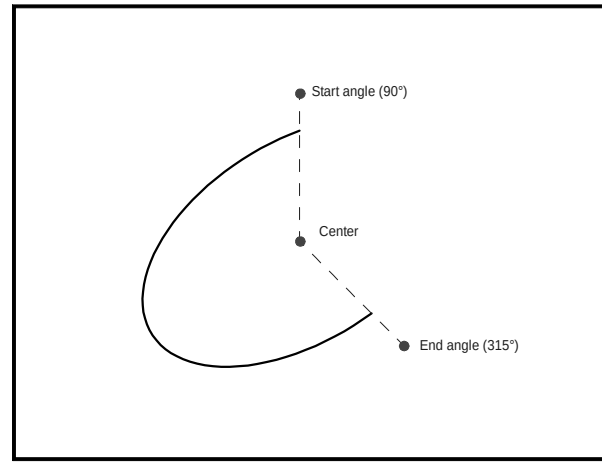
2. The prompt reads "**Inclination angle/to point :**". Inclination determines the angle of the major axis of the ellipse. Using the rubber-band cursor, pick a point that specifies a bearing.
3. The prompt reads "**First axis diameter:**" This can be the length for either the major or minor axis diameters. The order doesn't matter, as **FastCAD** will automatically utilize the larger value for the major axis. To specify the axis length, either type a value and press ENTER, or visually specify the diameter by picking points in the drawing window.
4. The prompt reads "**Second axis diameter:**". This is the length of the other axis diameter. To specify the axis length, either type a value and press ENTER, or visually specify the diameter by picking points in the drawing window.

FastCAD draws the **ellipse** and terminates the command.

Text equivalent: ***ELIPI***

Ellipses > Elliptical Arc





ELLIPSES > ELLIPTICAL ARC ***draws an elliptical arc. The procedure for drawing an elliptical arc is much like the*** **ELLIPSE:CENTER AND AXIS POINTS** ***command, except that the starting and ending angle of the arc segment need to be defined.***

To draw an **elliptical arc**, select **[Draw > Ellipses > Elliptical Arc]**:

1. ***The prompt reads*** Center point[2.55882,5.94118]:

   To specify the ***ellipse center***:

   - Right-click or press **ENTER** to accept the default or ***prior center;***
   - Pick a point, or;
   - Type numeric coordinates and press **ENTER**.

   FastCAD displays a rubber-band cursor, anchored at the center.

2. ***The prompt reads*** Major axis endpoint:

   Pick a point that defines an ***axis endpoint***. The point you select can describe either the major or minor axis.

   FastCAD displays a stretching ***ellipse cursor***.

3. ***The prompt reads*** Minor axis point:

   Pick a point when the dynamic cursor displays the size and shape you want.

   FastCAD draws a ghost ***ellipse*** and a rubber-band cursor anchored at the ***ellipse***'s center.

4. ***The prompt reads*** Starting angle/point:

   This point defines the ***arc*** segment based on the defined ***ellipse***. Pick a point at the desired angle from the center, or type a ***bearing value*** and press **ENTER**.

FastCAD displays a stretching ***elliptical arc*** that follows the rubber-band cursor.

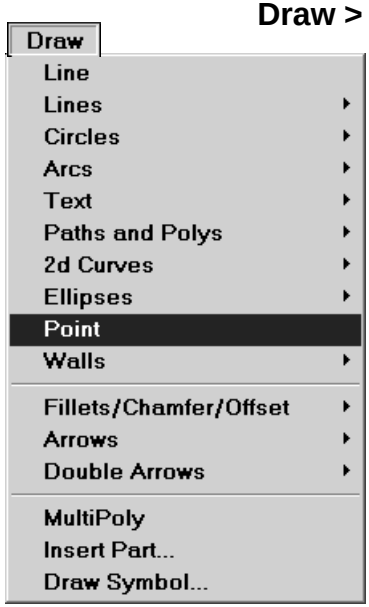
5. ***The prompt reads*** Ending angle/point:

   Pick a point when the ***arc*** looks the way you want, or type an ***bearing value*** and press **ENTER.**

FastCAD draws the ***elliptical arc*** and terminates the command.

Text equivalent: ***ELIPA***

Draw > Point



POINT ***draws a single dot*** (**one pixe**l). ***Points*** a***re usually used as temporary reference markers that you hide or remove when your drawing is complete.***

To draw a **point**, select **[Draw > Point]**:

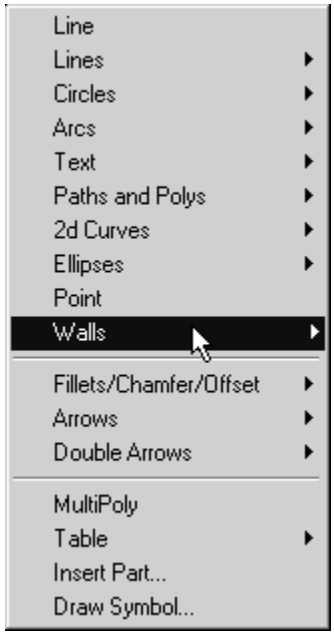
1. ***The prompt reads*** 

   To specify the position of each point, pick points in the drawing window, or type numeric coordinates.

2. After drawing a point, **FastCAD** lets you draw a series of points without re-selecting the command. Right-click to stop drawing points.

Note: On some high-resolution displays, a single pixel is very small and may be difficult to see using certain background color and entity color combinations.

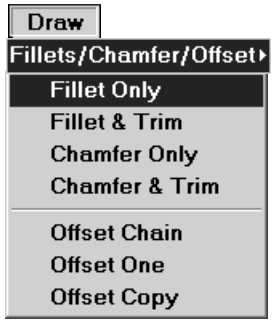
Text equivalent: ***POINT***

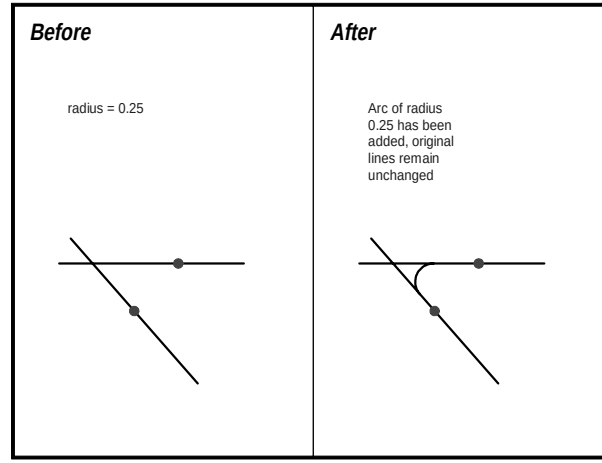


Draw > Walls

See **chapter 9** for details on drawing **WALL** entities.

Fillets/Chamfer/Offset > Fillet Only





FILLETS / CHAMFER / OFFSET > FILLET ONLY ***draws an arc entity at the intersection of lines, circles, arcs, ellipses, and splines for a smoothly rounded corner***. It does not change the selected entities in any way; it only draws the ***fillet*** between them. If you want the selected entities to be trimmed to the ***fillet***, use the **FILLET & TRIM** command instead.

Note: ***do not specify fillets of zero radius***. ***A zero-radius fillet cannot exist***. Use the **TRIM TO INTERSECTION [Edit > Trims > Trim to intersection]** command instead.

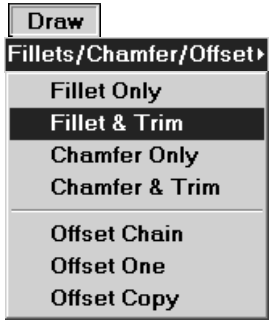
To draw a **fillet**, select **[Draw > Fillets/Chamfer/Offset > Fillet Only]**:

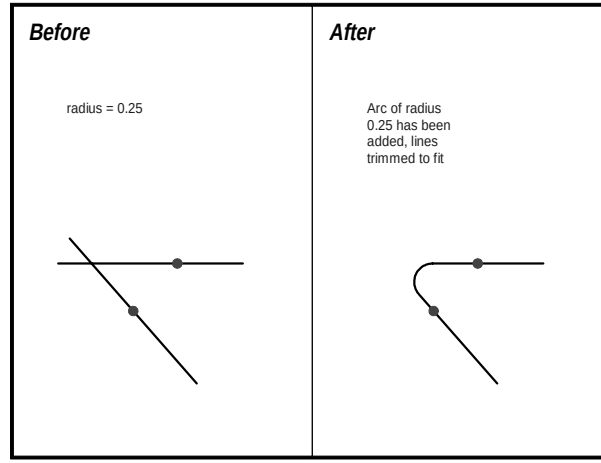
1. The prompt reads "**1st entity:**". Pick a valid entity. **FastCAD** highlights the selected entity.
2. The prompt reads "2**nd entity:**". Pick another valid entity (it must be neither parallel nor concentric to the first entity). **FastCAD** highlights the selected entity.
3. The prompt reads "**Fillet radius [x]:**", where x is the prior or ***default radius*** value. To specify a ***radius***, right-click to accept the default value, or type a new value and press **ENTER**.

FastCAD draws the **fillet arc** without modifying the picked entities.

Text equivalent: ***FIL*** *or* ***FILLET***

Fillets/Chamfer/Offset > Fillet & Trim





FILLETS / CHAMFER / OFFSET > FILLET & TRIM ***draws an arc entity at the intersection of lines, circles, arcs, ellipses and splines for a smoothly rounded corner***. This command will trim or extend the selected lines or arcs to meet the ends of the ***fillet arc***. If you want to add a ***fillet*** but do not want to change the selected entities in any way, use the **FILLET ONLY** command instead.

Note: do not specify fillets of zero radius. ***A zero-radius fillet can not exist***. Use **TRIM TO INTERSECTION [Edit > Trims > Trim to intersection]** instead.

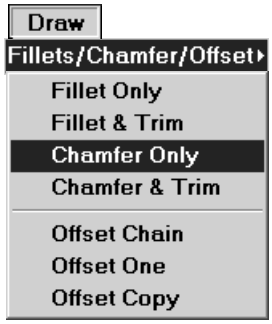
To draw a **fillet**, select **[Draw > Fillets/Chamfer/Offset > Fillet & Trim]**:

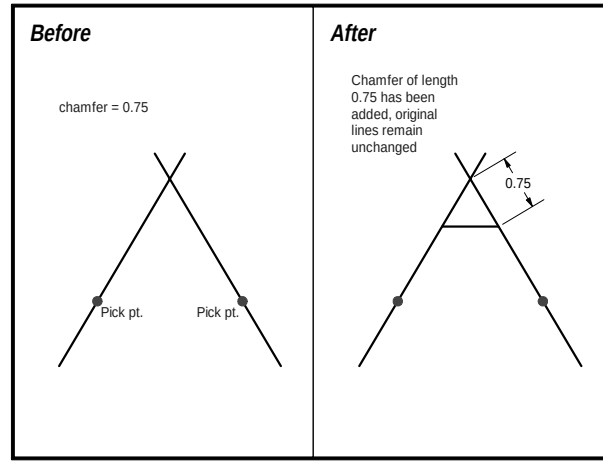
1. The prompt reads "**1st entity:**". Pick a valid entity. **FastCAD** highlights the selected entity.
2. The prompt reads "**2nd entity:**". Pick another valid entity (it must be neither parallel nor concentric to the first entity). **FastCAD** highlights the selected entity.
3. The prompt reads "**Fillet radius [n]:**". To specify a ***radius***, right-click to accept the default or prior value, or type a new value and press **ENTER**.

FastCAD draws the **fillet arc** and automatically trims or extends the selected entities to meet the ends of the ***fillet arc***.

Text equivalent: ***TFIL***

Fillets/Chamfer/Offset > Chamfer Only





FILLETS / CHAMFER / OFFSET > CHAMFER ONLY ***draws a line entity near the intersection of two entities for a beveled corner.*** You can add a ***chamfer*** to entities that cross, entities that are already connected at their endpoints, or entities that do not meet (as long as they are not parallel). **CHAMFER** does not change the selected entities in any way; it only draws the ***chamfer*** between them. If you want the selected entities to be trimmed to the chamfer, use the **TCHAM** command instead.

NOTE: ***Do not specify chamfers of zero length***. There will be no apparent change in the drawing, but a zero-length chamfer line entity will be drawn at the intersection point. Instead, use the **TRIM TO INTERSECTION** command **[Edit > Trims > Trim to intersection].**

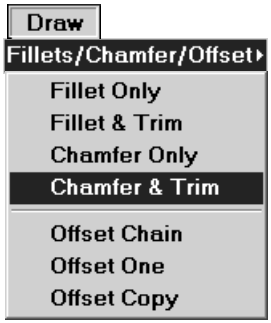
To draw a **non-trimming chamfer**, select **[Draw > Fillets/Chamfer/Offset > Chamfer Only]:**

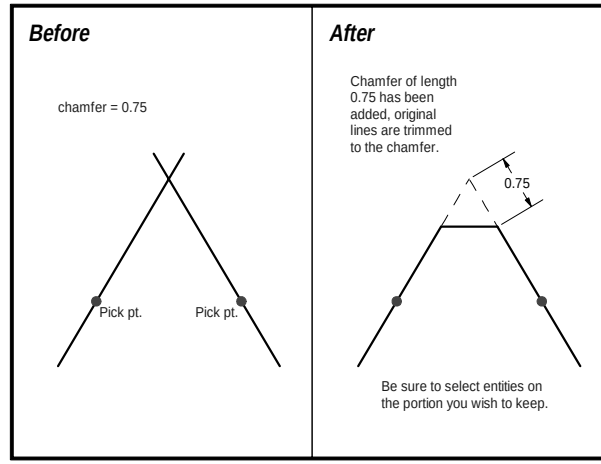
1. The prompt reads "**1st entity:**". Pick a valid entity. **FastCAD** highlights the selected entity.
2. The prompt reads "**2nd entity:**". Pick another valid entity (it must be not be parallel to the first entity). **FastCAD** highlights the selected entity.
3. The prompt reads "**Chamfer distance [n]:**". The chamfer distance specifies how far from the intersection of the selected entities to put the chamfer line (not the length of the chamfer line). To specify the chamfer distance, right-click to accept the default or prior value, or type a new value and press **ENTER**.

FastCAD draws the **chamfer line** without modifying the picked entities.

Text equivalent: ***CHAM*** *or* ***CHAMFER***

Fillets/Chamfer/Offset > Chamfer & Trim





FILLETS / CHAMFER / OFFSETS > CAMFER & TRIM ***draws a line entity near the intersection of two entities for a beveled corner***. You can add a ***chamfer*** to entities that cross, entities that are already connected at their endpoints, or entities that do not meet (**as long as they are not paralle**l). If you do not want **FastCAD** to trim the entities, use the command **CHAMFER**.

Note: do not specify chamfers of zero length. If you specify a ***chamfer*** distance of 0 (**zero**), **FastCAD** will extend or trim the two lines to their intersection point. However, a zero-length line entity will be created at the intersection. Instead, use the **TRIM TO INTERSECTION** command **[Edit > Trims > Trim to intersection]**.

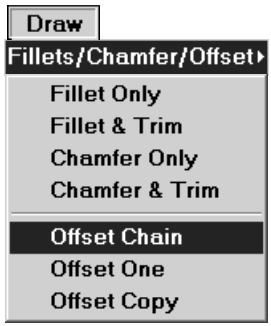
To draw a **trimmed chamfer,** select **[Draw > Fillets/Chamfer/Offset > Chamfer & Trim]**:

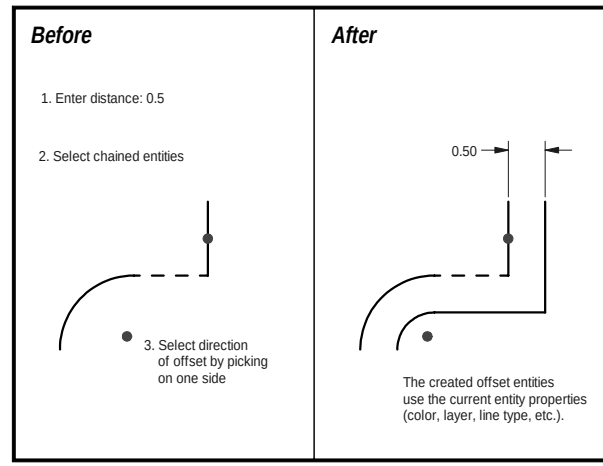
1. The prompt reads "**1st entity:**". Pick a valid entity. **FastCAD** highlights the selected entity.
2. The prompt reads "**2nd entity:**". Pick another valid entity (it must be not be parallel to the first entity). **FastCAD** highlights the selected entity.
3. The prompt reads "**Chamfer distance [n]:**". The chamfer distance specifies how far from the intersection of the selected entities to put the **chamfer line** (not the length of the **chamfer line**). To specify the chamfer distance, right-click to accept the default or prior value, or type a new value and press **ENTER**.

FastCAD draws the **chamfer line**, and automatically trims or extends the picked entities to meet the ends of the **chamfer line**.

Text equivalent: ***TCHAM***

Fillets/Chamfer/Offset > Offset Chain





FILLETS / CHAMFER / OFFSETS > OFFSET CHAIN ***creates a parallel copy of a selected entity chain at a specified offset distance***. **OFFSET** works with "**connected**" ***lines***, ***circles***, ***arcs***, ***ellipses***, ***elliptical arc***s, ***paths***, ***polygons***, and ***splines***. ***Entities must share endpoints to continue the sequence or*** "**chain**". If more than two entities share the same endpoint, **FastCAD** makes a best guess as to what direction the chain is to take.

The newly created offset entities acquire the current entity properties (**color**, **layer**, **line width**, **line style**, **fill style**) as specified on the ***Status*** *bar*. I***f you wish the offset entities to have the same properties as the originals, use the*** **OFFSETCPY** command **[Draw > Fillets/Chamfer/Offset > Offset Copy].**

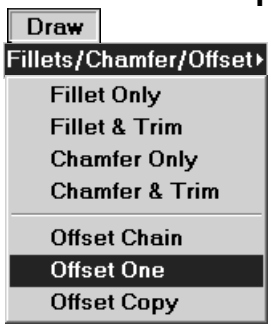
Offset copies ***of splines and ellipses are path entities parallel to the originals.***
You may use **OFFSET** on entities belonging to a **GROUP**. The new offset entities will not be included in the group.
To create an **offset chain,** select **[Draw > Fillets/Chamfer/Offset > Offset Chain]**:

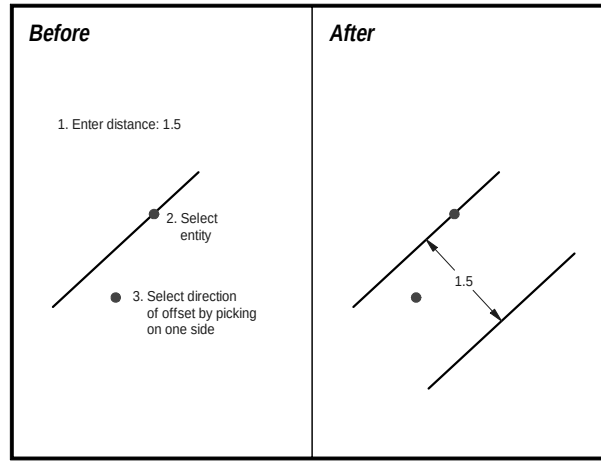
1. The prompt reads "**Distance [n]:**". To specify an offset distance:
   - Right-click or press ENTER to accept the default or prior distance;
   - Visually pick two points the desired distance apart, or;
   - Type a new value for the offset distance and press ENTER.
2. The prompt reads "**Entity:**". Pick an entity whose endpoint is shared with at least one other entity.
3. The prompt reads "**Pick a side:**". To indicate which side of the entity chain you wish the offset to be drawn, pick a point.

   FastCAD draws the **offset copy** and terminates the command.

Text equivalent: ***OFFSET***

Fillets/Chamfer/Offset > Offset One





FILLETS / CHAMFER / OFFSETS > OFFSET ONE ***creates a parallel copy of a selected entity at a specified offset distance.***

T***he newly created offset entities acquire the current entity properties*** (**color**, **layer**, **line width**, **line style**, **fill style**) as specified on the ***Status*** *bar*. If you wish the ***offset*** entities to have the same properties as the originals, use the **OFFSETCPY** command **[Draw > Fillets/Chamfer/Offset > Offset Copy].**

OFFSET1 works with ***lines***, ***circles***, ***arcs***, ***ellipses***, ***elliptical arcs***, ***paths***, ***polygons***, and ***splines***. ***Offset copies of splines and ellipses are paths parallel to the originals***.

You may use **OFFSET1** on entities belonging to a **GROUP**. ***The new offset entities will not be included in the group.***

To create an **offset chain**, select **[Draw > Fillets/Chamfer/Offset > Offset One]**:

1. The prompt reads "**Distance [n]:**". To specify an offset distance:

- Right-click or press **ENTER** to accept the default or prior distance;
- Visually pick two points the desired distance apart, or;
- Type a new value for the offset distance and press **ENTER**.

2. The prompt reads "**Entity:**". Pick the entity you wish to create an ***offset copy***.

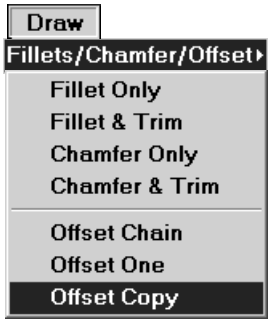
3. The prompt reads "**Pick a side:**". To indicate which side of the entity ***chain*** you wish the offset to be drawn, pick a point.

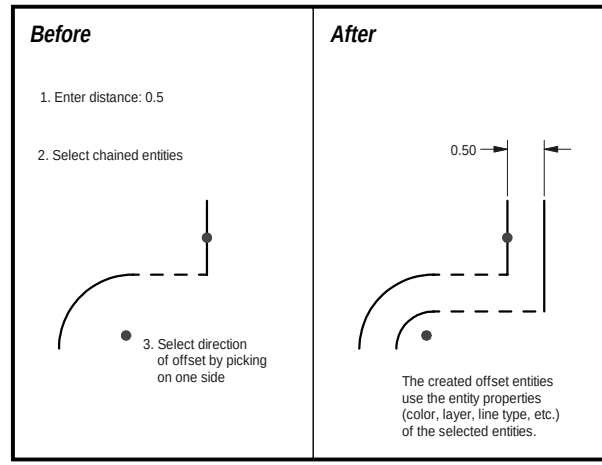
FastCAD draws the **offset copy**.

4. The prompt reads "**Entity:**". You may continue to select entities and ***offset*** directions at the distance already specified. Right-click to end the command.

Text equivalent: ***OFFSET1***

Fillets/Chamfer/Offset > Offset Copy





FILLETS / CHAMFER / OFFSETS > OFFSET COPY c***reates a parallel copy of a selected entity chain at a specified offset distance***. It works just like the **OFFSET** command ***except that the new offset copies retain the same entity properties*** (such as **color**, **layer**, **line style**, **line width**, etc.) ***as the originals***.

OFFSET COPY works with "**connected**" ***lines***, ***circles***, ***arcs***, ***ellipses***, ***elliptical arcs***, ***paths***, ***polygons***, and ***splines***. Entities must share endpoints to continue the sequence or "**chain**". If more than two entities share the same endpoint, **FastCAD** must make a best guess as to which direction the chain is to take. Otherwise, you may use **OFFSET COPY** with single unconnected entities.

Offset copies of ***splines*** and ***ellipses*** are ***path*** entities that are parallel to the originals.

You may use **OFFSET COPY** on entities belonging to a **GROUP**. The new ***offset*** entities will not be included in the group.

To create an ***offset chain***, select **[Draw > Fillets/Chamfer/Offset > Offset Copy]**:

1. The prompt reads "**Distance [n]:**". To specify an offset distance:

- Right-click or press ENTER to accept the default or prior distance;
- Visually pick two points the desired distance apart, or;
- Type a new value for the offset distance and press ENTER.

2. The prompt reads "**Entity:**". Pick an entity whose endpoint is shared with at least one other entity. If an unconnected entity is chosen, only that entity will be selected.

3. The prompt reads "**Pick a side:**". To indicate on which side of the entity chain you wish the offset to be drawn, pick a point.

FastCAD draws the **offset copy** and terminates the command.

Text equivalent: ***OFFSETCPY***

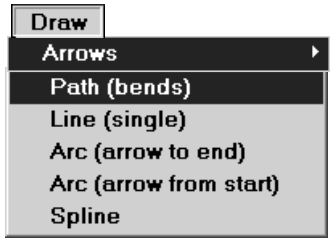
Arrows

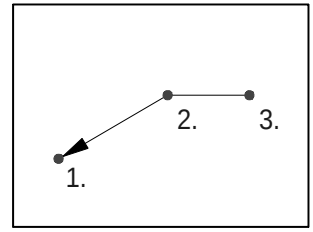
The **ARROWS** commands ***draw entities*** (**paths**, **arcs**, or **splines**) ***with an arrowhead at the first node point.*** Please keep in mind the following:

- The arrowhead shape and size are determined by **ARROWHEAD STYLE [Specs > Arrowhead Style**];
- The ***arrowhead style*** is independent from ***arrowheads used in dimensions***.
- You can create notes by combining **ARROW** entities with text entities.

PATH ARROW icon 

Path Arrow (bends)



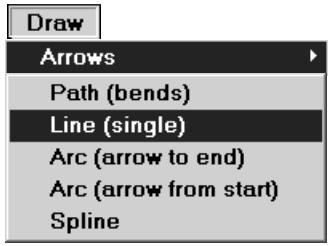


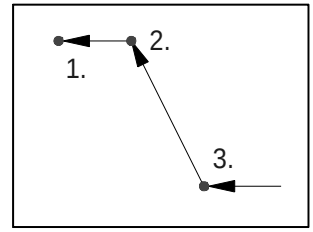
PATHA draws connected line segments with an arrowhead placed at the first point.

1. Select the first point. Use the crosshairs and left button, or type numeric coordinates. The arrow will point here.
2. Select the next node point on the arrow. Continue selecting nodes until you press the right button to end the line.

Text equivalent: ***PATHA***

Line Arrow (single)



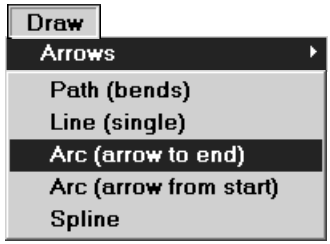


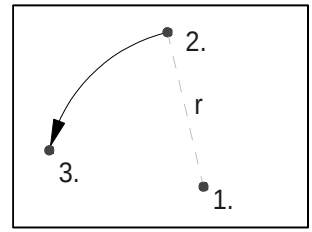
LINEA draws lines, each with arrowheads at the first generating point. If connected lines are drawn, each segment will begin with an arrowhead.

1. Once you've selected the first point, use the crosshairs or type numeric coordinates.
2. Select the next point to draw the line. You can continue to draw a series of connected line arrows until you press the right button to end the line.

Text equivalent: ***LINEA***

Arc Arrows (arrow to end)



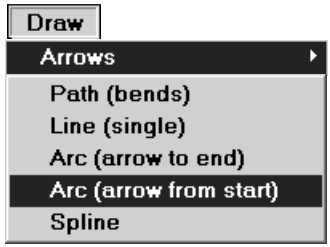


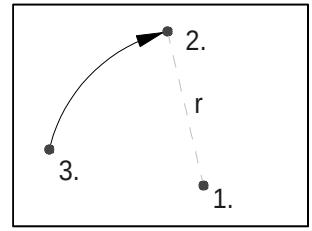
ARCA draws an arrow to the end of the arc segment you define. The ***arc radius*** (r) is determined by the distance between the center (1.) and the arc begin point (2.).

1. Select the ***center point***. Pick the point, or type numeric coordinates.
2. Select the 2nd and 3rd points of the ***arc***. The arrow will appear on the 3rd point, at the end of the arc.

Text equivalent: ***ARCA***

Arc Arrow (arrow from start)



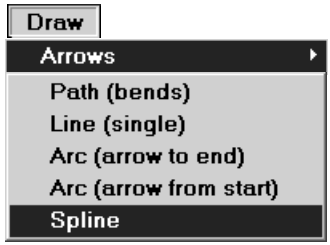


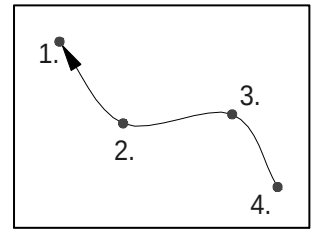
ARCAS draws an arrow pointing to the start of the arc segment you define. The arc radius (r) is determined by the distance between the center (1.) and the arc begin point (2.).

1. Select the ***center point***. Pick the point, or type numeric coordinates.
2. Select the 2nd and 3rd points of the arc in the same manner as you selected the 1st point. The arrow will appear on the 2nd point, at the start of the arc.

Text equivalent: ***ARCAS***

Spline Arrow





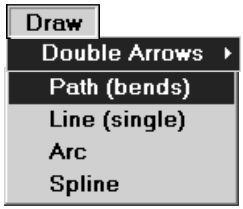
SPLINEA draws a parabolic spline and places an arrowhead at the starting point.

1. Select the first point. Pick the point, or type numeric coordinates.
2. Select the next point. Continue selecting nodes until you right-click to end the arrow.

Text equivalent: ***SPLINEA***

Double Arrows

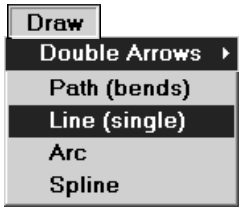
DOUBLE ARROW commands draw entities just like the **ARROW** commands, except with arrowheads on both ends of each entity. The arrowhead shape and size are determined by **DIMSTYLE [Specs > Dimension Styles]**. You can create notes by combining **DOUBLE ARROW** entities with text entities. If you use **BREAK** to insert a gap in the double arrow, **FastCAD** replaces it with two single arrows.



Path Double Arrow (bends)

1. Select the first point. Pick the point, or type numeric coordinates.
2. Select the next node point on the arrow. Continue selecting nodes, or right-click to end the arrow.

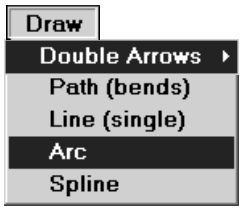
Text equivalent: ***PATHDA***



Line Double Arrow (single)

1. Select the first point. Pick the point, or type numeric coordinates.
2. Select the next point to draw the line. You can continue to draw a series of connected line arrows, or right-click to end the arrow.

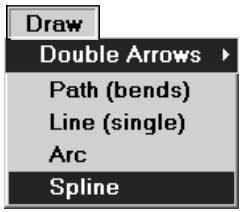
Text equivalent: ***LINEDA***



Arc Double Arrow

1. Select the first point. Pick the point, or type numeric coordinates.
2. Select the 2nd and 3rd points of the arc in the same manner as you selected the 1st point.

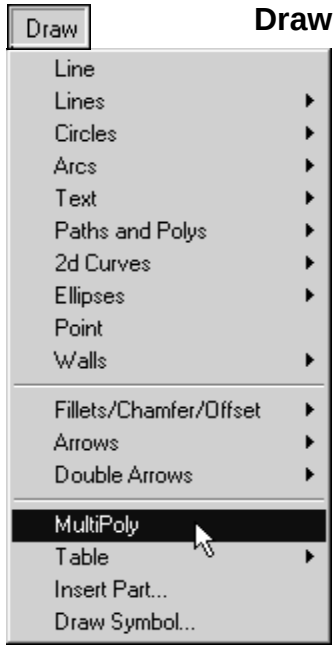
Text equivalent: ***ARCDA***



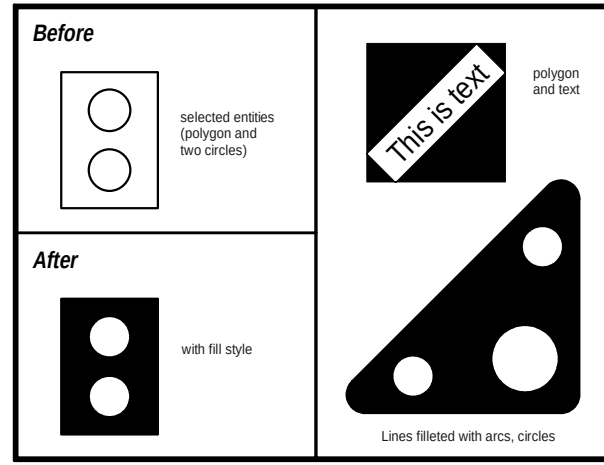
Spline Double Arrow

1. Select the first point. Pick the point, or type numeric coordinates.
2. Select the next point. Continue selecting nodes, or right-click to end the arrow.

Text equivalent: ***SPLINEDA***



Draw > Multipoly



MULTIPOLY (**Multiple Polygons**) ***collects existing entities and turns them into a new single entity. This entity can then be hatched with a fill pattern***. For instance, a triangle you may have drawn with lines instead of with the polygon command can be "**multipolied**" and then hatched with a ***fill style***.

An enclosed entity inside a filled area will leave a hole, or unfilled area. The entities that can be included in a ***multipoly*** are ***lines***, ***arcs***, ***circles***, ***ellipses***, ***paths***, ***polygons***, ***splines***, ***text***, and ***existing multipolys***.

MULTIPOLY works differently from most others because it requires that the entities you want to ***multipoly*** must already exist before selecting the **MULTIPOLY** command.

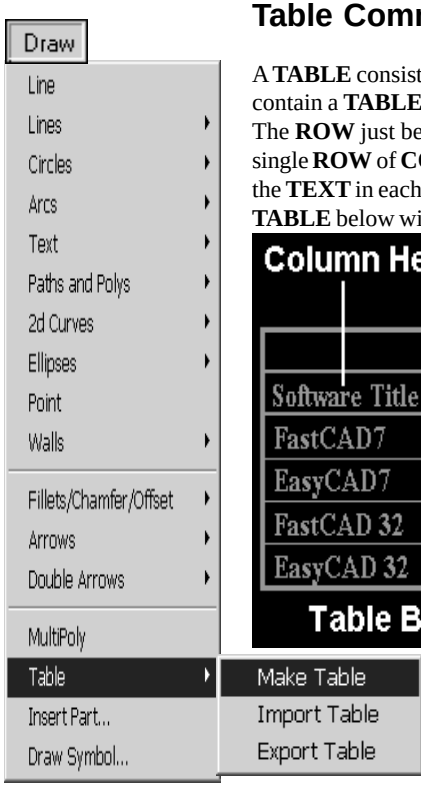
In order for an applied fill style to be contained, the bounding entities must share endpoints. Otherwise, the fill pattern may leak out. When a ***multipoly*** is created, all affected entities are changed to the current color, layer, and style. **DONUT** and **WEDGE** are examples of ***multipolys*** that **FastCAD** builds "**on-the-fly**". ***Multipolys*** can also be used to group entities together, although it does affect the ***layer***, ***color***, and ***style*** of those entities.

To use, select the desired entities. **FastCAD** replaces the selected entities with a ***multipoly*** entity, giving it the ***current layer***, ***color***, ***line style***, and ***fill style***.

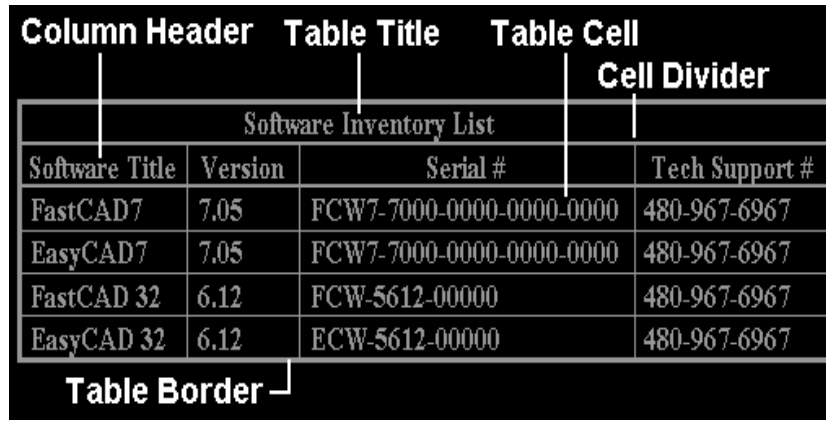
To "**un-multipoly**", use the **EXPLODE** command. An exploded ***multipoly*** reduces to its component entities (as if they were never multipolied). Any ***fill styles*** applied to an expldoed ***multipoly*** are lost.

Text equivalent: ***MPOLY2***

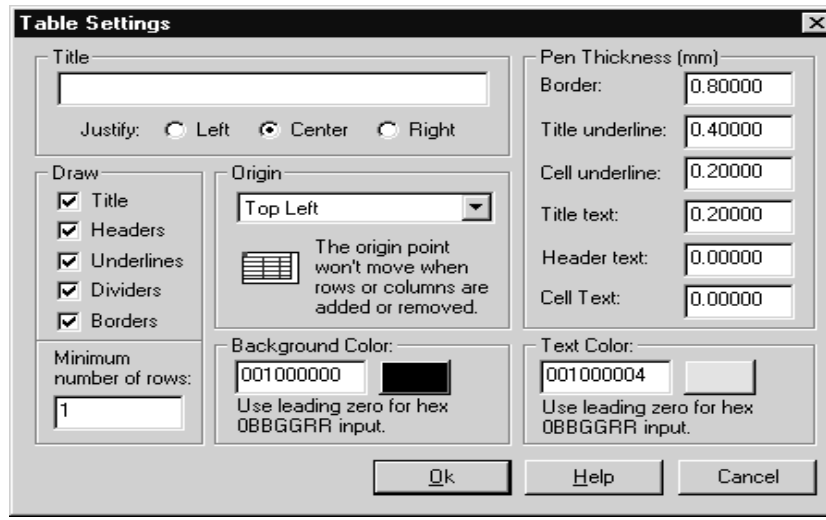
Table Commands



A **TABLE** consists of a **RECTANGULAR ARRAY** of **ROWS** and **COLUMNS** that contain a **TABLE TITLE** at the top extending across the entire width of the **TABLE**. The **ROW** just below the TITLE contains the **COLUMN HEADERS**, which is a single **ROW** of **COLUMN LABELS**. The **TITLE** and **HEADER LABELS**, as well as the **TEXT** in each cell, may be **RIGHT**, **CENTER** or **LEFT** justified. See the sample **TABLE** below with its various areas labeled.



When you select the **[Draw > Table > Make Table]** command, the **TABLE SETTINGS** dialog box opens as seen below.



Making a New Table

Enter the **TABLE TITLE**, and click the **LEFT, CENTER** or **RIGHT** radio button to justify it. You may leave the **TITLE** blank and enter it later if you like.

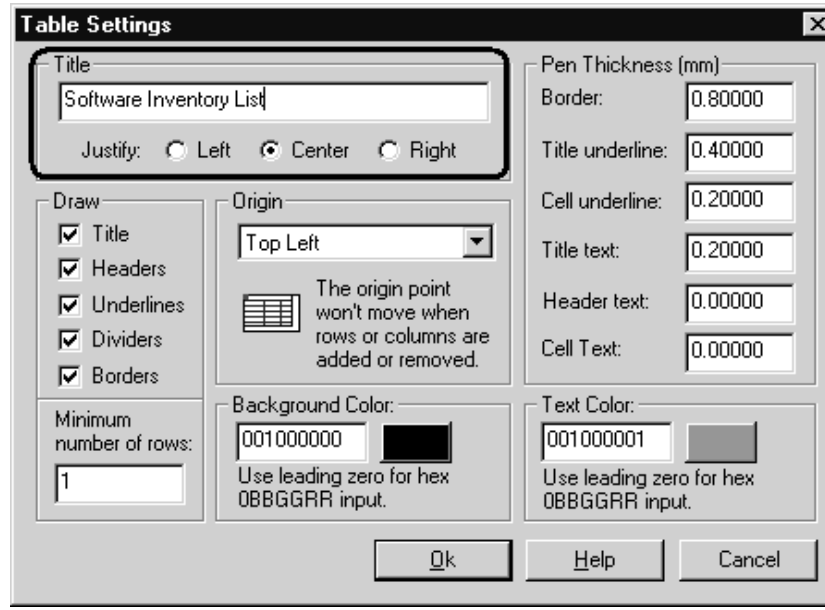
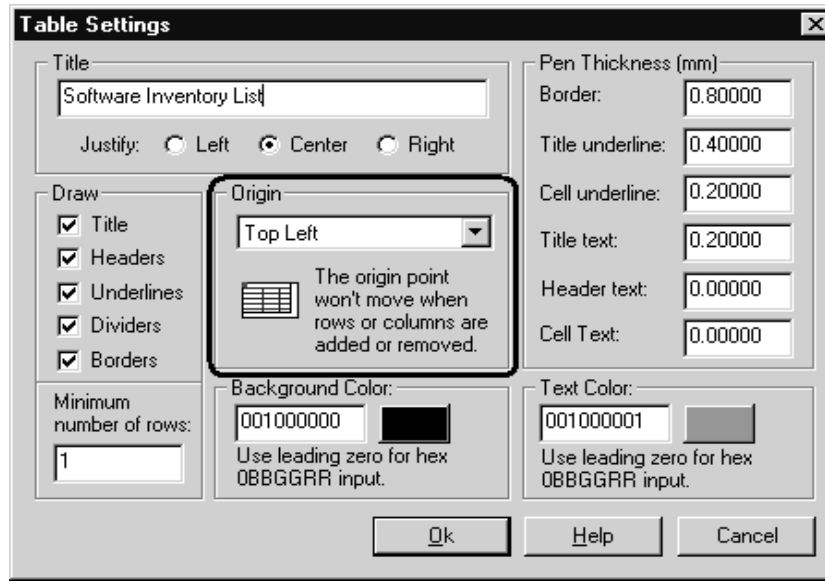
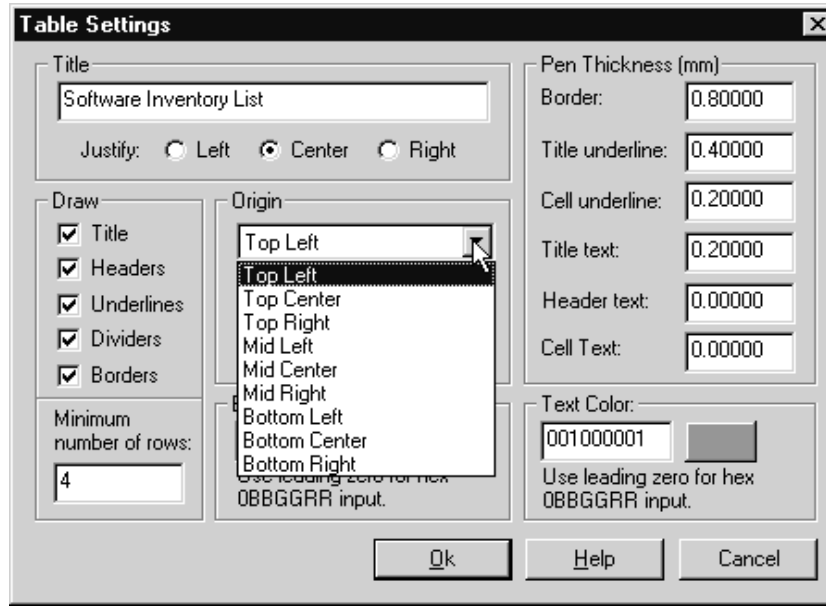


Table Origin

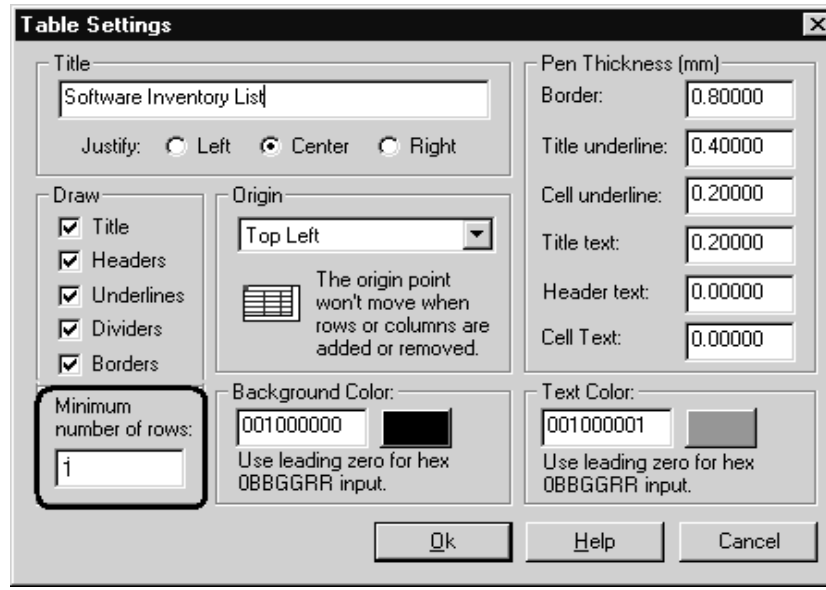
The **TABLE ORIGIN** is the point used to place the **TABLE** in the drawing. If **ROWS** are ***added*** or ***deleted***, the **ORIGIN** point will remain fixed in place, so you can safely **ADD** to or **REMOVE** elements from a **TABLE** and have it remain lined up in title blocks, etc.



The TABLE ORIGIN May be set for the folloeing nine positions.

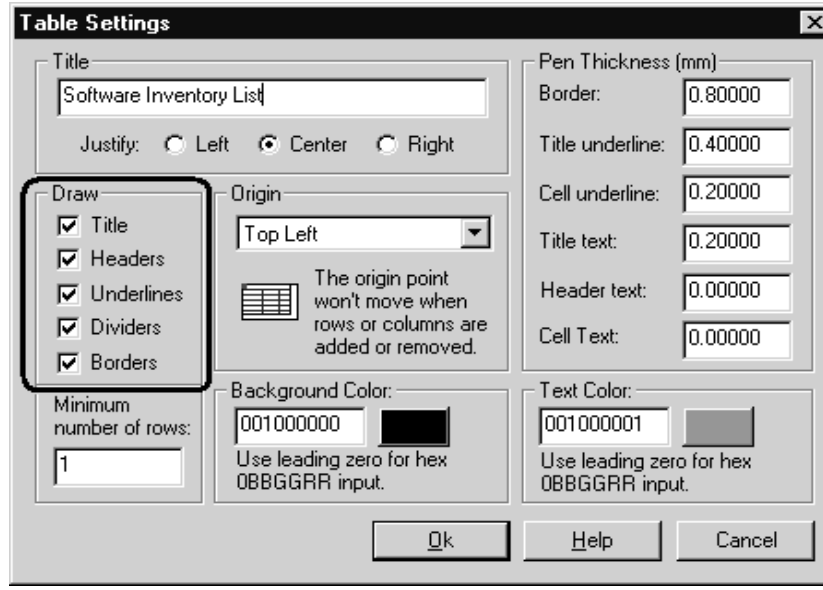


Minimum number of ROWS setting



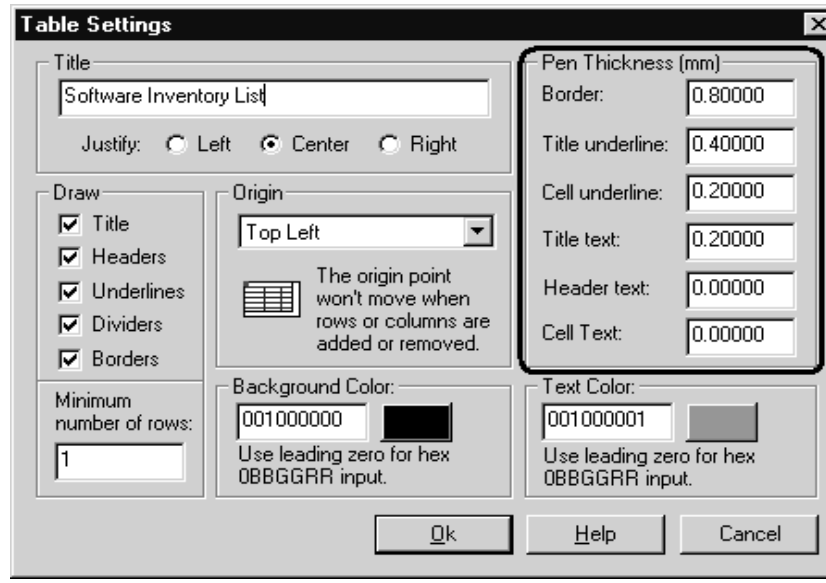
The **TABLE** must have at least **ONE ROW** so **ONE** is the default **MUNIMUM NUMBER OF ROWS** in a new **TABLE**. Set this number to the number of **ROWS** that you want in your **TABLE**. Remember you may use the **[Edit > Dynamic Edit]** command to dynamically **ADD** or **REMOVE ROWS** after the TABLE is placed in the drawing.

DRAW area



These five check boxes **ENABLE** or **DISABLE** the display of the **TITLE**, **HEADERS**, **UNDERLINE**, **DIVIDERS** and **BORDERS** in the **TABLE**.

PEN THICKNESS [mm] area

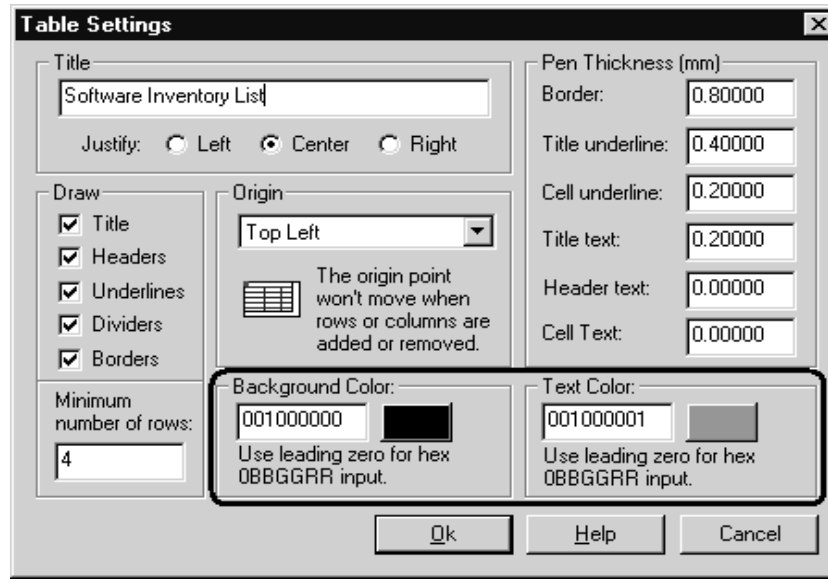


Individule **PEN THICKNESSES** may be set for the **BRODER, TITLE UNDERLINE, CELL UNDERLINE, TITLE TEXT, HEADER TEXT and CELL TEXT. PEN THICKNESS** is only applied to **VECTOR** fonts as described below.

NOTE: Windows ALWAYS considers the screen as **[96 PIXELS PER INCH]**, regardless of resolution or monitor size. So if you specify a 0.2mm line, then it will draw as one pixel, but will print heavier than a 0.1mm line, because the printer has better resolution. (**.265mm = 1 pixel. Numbers are rounded, so anything less than 0.39mm will draw as a single pixel)**.

Text Pen Thickness settings only apply to vector fonts such as the **FastCAD DOS .FNT** we provide with **FastCAD v7** or the **AutoCADâ .SHX** fonts if you own them.

TABLE Background and TEXT Color



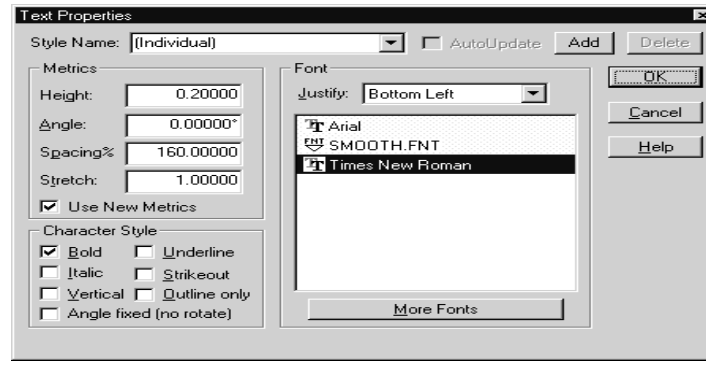
The **CURRENT DRAWING BACKGROUND COLOR** is the **INITIAL TABLE BACKGROUND COLOR**.

You can use the color sample button to change this in this dialog, or the **[Edit > Change > Fill (2nd) Color]** command later to specify a ***different background color.***

The **CURRENT DRAWING COLOR** is the **DEFAULT COLOR** for the **TABLE TEXT**.

You can also specify a different color here. This is initially the **TABLE'S ENTITY COLOR**. Lines are drawn using the current entity color, so these colors can be changed with the **[Edit > Change > Color]** command after the **TABLE** is placed in the drawing.

TABLE TEXT Specifications



The CURRENT TEXT SPECS are set for the TABLE initially.

They can be changed with the **[Edit > Change Text properties]** command later. These values have some special interactions with **TABLES**:
The **TEXT ANGLE** is **NOT USED -** the **ANGLE** is determined by the **TABLE'S** current angle. The justification for everything is set to **LEFT**, **CENTER**, or **RIGHT** by this dialog, and by the **COLUMN** dialogs that follow.

About Text Height

TABLE TEXT PROPERTIES, ***font***, ***height***, ***italic***, ***bold*** etc., apply to **ALL TABLE TEXT.**
You may NOT HAVE two different font styles or some bold and some italic **TEXT** in the TABLE.. **TABLE TEXT COLOR** may be set independently from the **TABLE SETTINGS** and **COLUMN** dialog boxes by selecting the **[Edit > Edit]** command and clicking on the edge of the **TABLE**.

Most importantly, the **HEIGHT** of a **ROW** is the **TABLES TEXT HEIGHT** and the **SPACING %**. Later, each column will be assigned a **COLUMN WIDTH**, and this is independent of the **TEXT HEIGHT** .

If TEXT doesn't fit in a COLUMN

If **TEXT** does not fit in a **COLUMN** it will **OVERUN** the **COLUMN**. Use the **[Edit > Edit]** command and select the **TABLE BORDER** and increase the **COLUMN WIDTH** in the appropriate **COLUMN** dialog box.
You may also use the **[Edit > Dynamic Edit]** command and select the **TABLE BORDER** and place the cursor target box over a **COLUMNS RIGHT VERTICAL DIVIDER** bar and left-click-hold and **DRAG** it **LEFT** or **RIGHT** to **INCREASE** or **DECREASE** the **COLUMN WIDTH**. The **TABLE** will automatically redraw and adjust to the new **COLUMN WIDTH** change.

Entering or Editing the Table Text

The **[Edit > Edit]** command will let you ***edit*** all **TABLE SETTINGS**, so you can change **COLUMN WIDTHS** later if **COLUMNS** need more room.

To **EDIT** a **TABLE** select **[Edit > Edit]** and click on the edge of the **TABLE BORDER** and the **TABLE SETTINGS** dialog will appear ready for you to make changes.

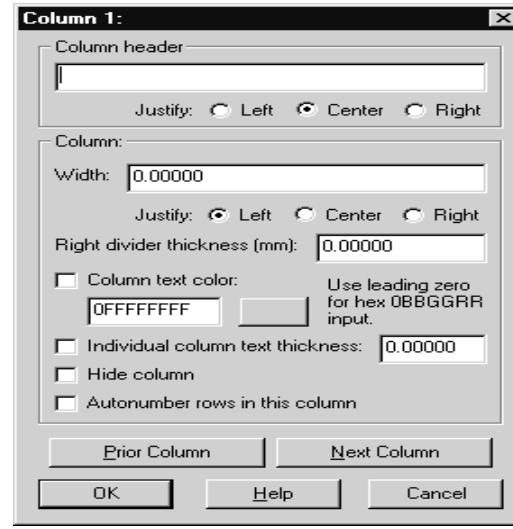
Generally it is a good idea to **INCREASE** the **TEXT SPACING PERCENT** to **160** to allow room for **ROW LINES**.

When all settings are complete, click **OK** to start entering **COLUMN** information.

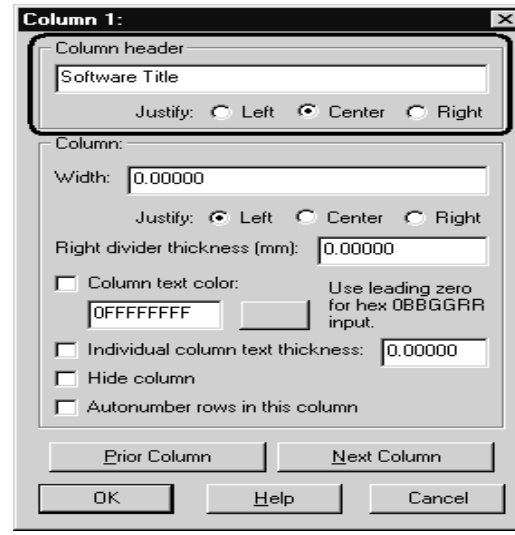
To enter **TEXT** ***into a blank cell*** or **EDIT TEXT** ***in a cell*** select the **[Edit > Dynamic Edit]** command and click on the **TABLE BORDER** to select it. Click directly in a **CELL** and it will highlight. Modify its contents and press **ENTER.** You may use the four **ARROW keys** on the keyboard to move to an **ADJECENT CELL** or just left-click on another **CELL**. When you are done editing the **TABLE** right-click to end the command.

TABLE COLUMN dialog box - COLUMN Header

When you are done entering options into the **TABLE SETTINGS** dialog box click its **OK** button and the **COLUMN 1** dialog box will appear.

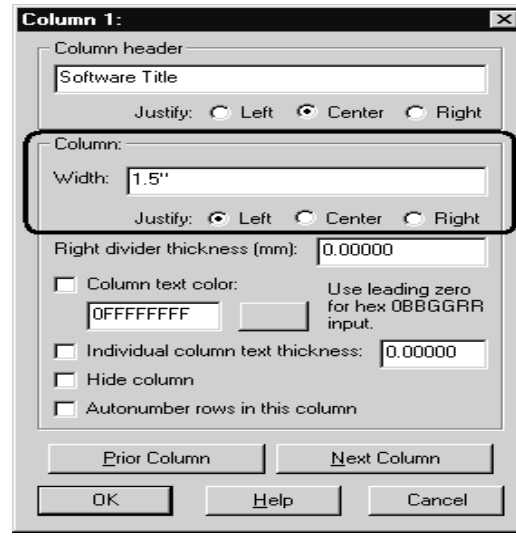


Enter a **COLUMN HEADER NAME** and click the radio button for **LEFT**, **CENTER** or **RIGHT** to justify it in the **HEADER** cell over the **COLUMN**.



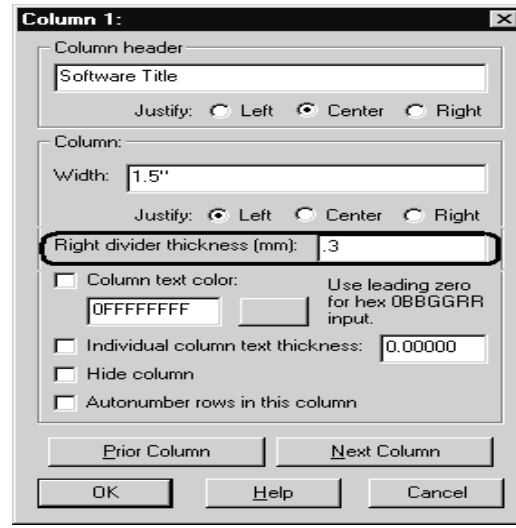
COLUMN WIDTH setting

You can **TAB** to the **COLUMN WIDTH** window, which must be greater than 0. After entering the **COLUMN WIDTH** and click the **LEFT, CENTER** or **RIGHT** radio button to select the desired **JUSTIFICATION** for the **COLUMNS TEXT.** A **TAB** takes you to the nex option window in the **COLUMN 1** dialog box. If you are done with this **COLUMN** dialog box click the **'Next Column'** button, or you can press '**N**' (or '**P**' to edit earlier columns) without needing to use the mouse.



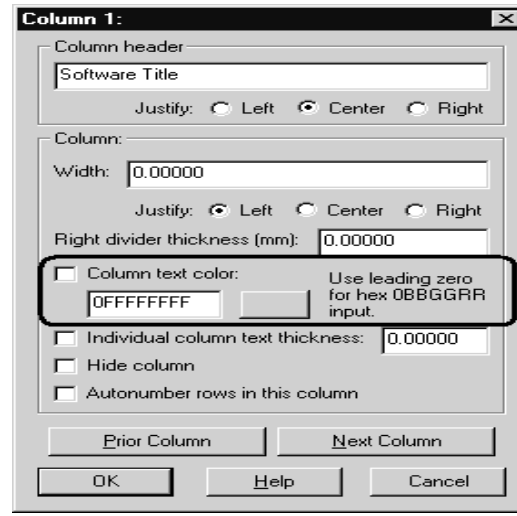
COLUMN Divider PEN THICKNESS setting

The **VERTICAL DIVIDER** line of the **COLUMN** may have a **PEN THICKNESS** in **MILIMETERS** set ***independently*** of the **TABLE BORDER** in this window.



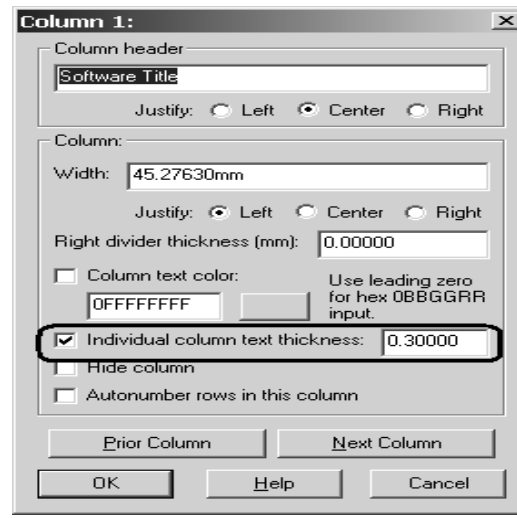
Individule COLUMN TEXT Color setting

You can also set **INDIVIDUAL COLUMN TEXT COLOR** by clicking the **COLOR BLOCK** selecting from the color dislog.



Individule COLUMN TEXT Pen Thickness setting

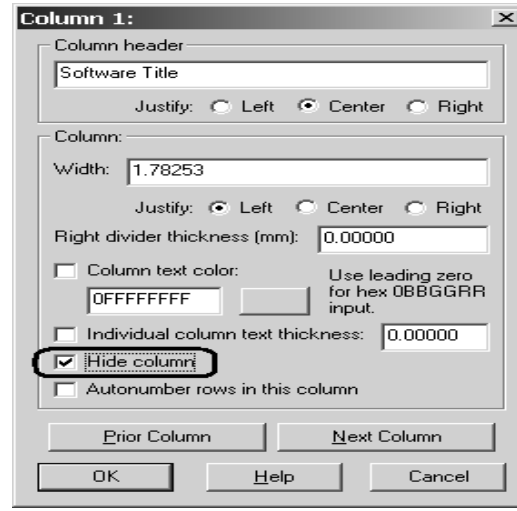
If you use **VECTOR FONTS** such as the provided **.FNT DOS** fonts or if you own **AutoCAD .SHX** fonts, you may give them a Pen Thickness. Setting a **Pen Thickness** here has **NO EFFECT** on Windowsâ **TrueType** fonts..



Hidding COLUMNS

You can check "**Hide Column**" box if you want the **TABLE** to be drawn without this **COLUMN**.

This is useful when you import a **.CSV** file **[Comma Separated Values]** from Excelâ or an address book from Netscape and do not wish to see everything. This can be changed later with the **[EDIT > Edit]** command. When all **COLUMNS HEADERS** have been entered, press **OK**.



Auto Numbering a COLUMN

Checking this box will **AUTO-NUMBER** each **ROW** starting at the **TOP ROW** with **#1** for as many **ROWS** as the **TABLE** has. This NUMBER cell may NOT be **EDITED. NO** other text can be placed in this ***cell***. The **CELL WIDTH** should be fairly narrow and you can leave the **COLUMN HEADER** blank or put a # as the **HEADER**.

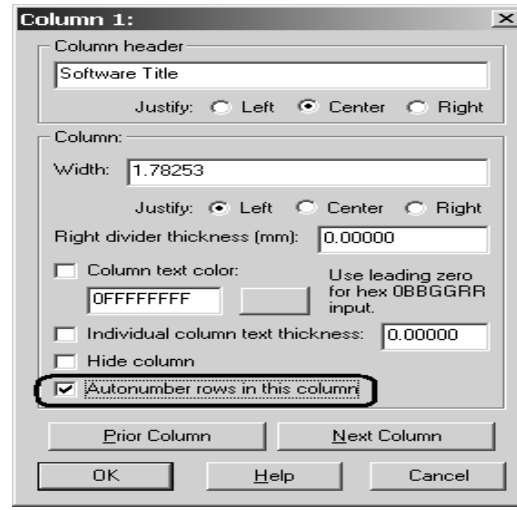
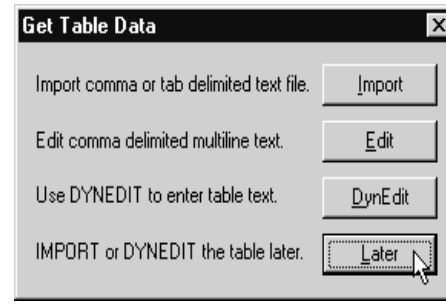


TABLE COLUMN Data Entry



After entering the last **COLUMN DATA** click **OK.** Now you are prompted for the source of the **TABLE'S TEXT DATA**. If you click **IMPORT**, the **TABLE TEXT** will be read from a ***comma or tab-delimited text file*** with either a **.CSV** or a **.TXT** file name extension.

Each **ROW** is ***entered on one line***, with ***columns separated by*** **TABS** or **COMMAS**. **BLANK COLUMNS** are entered by just specifying a **COMMA** with **NO SPACES** for the data. Extra fields, or not enough fields are corrected automatically. The file name will be saved and used as the default on a later **[Draw > Tables > Import Table]** or **[Draw > Tables > Export Table]** command for this **TABLE**.

You can also use the **Multiline text editor** to enter .**CSV data** directly (**but you must use commas, as Window's dialogs exit on a ENTER key**).
You can elect to enter the **TABLE'S DATA LATER** with either **[Edit > Dynamic Edit]** or the **[Draw > Tables > Import Table]** command, or you can the **[Edit > Dynamic Edit]** command now to enter **TABLE'S TEXT DATA**.

Editing a TABLE using the [Edit > Edit] command

After you place a **TABLE** in the drawing, its **PROPERTIES** may quickly and easily be changed. Click the **[Edit > Edit]** command and ***click on the edge of the*** **TABLE BORDER** and the **TABLE SETTINGS** dialog will appear. Make any changes you want and click the **OK** button.

The **COLUMN 1** dialog box will appear ready for you to make changes to its **PROPERTIES**. You can click the **NEXT COLUMN** or **PRIOR COLUMN** buttons or simply type the **P** or **N** shortcut keys to move between **COLUMN** dialog boxes.

Editing a TABLE using the [Edit > Dynamic Edit] command

This is normally how you will add **TEXT** to a newly created **TABLE**. After you place a **TABLE** in the drawing you may enter **TEXT** into a blank cell or edit the **TEXT** already in a ***cell***.

You select the **[Edit > Dynamic Edit]** command and ***click on the edge of the*** **TABLE BORDER** and it will highlight signifying that it has been selected for editing.

Currently, **DYNAMIC EDIT** only supports ***non-rotated tables***. It currently highlights the **BORDER** of the **TABLE**when it is selected for **DYNAMIC EDIT**. Nested commands work. Clicking on a ***cell*** opens an "**in place**" edit box with correct sized left-justified text and a ***red position cursor***.

Key input, **BACKSPACE**, **DEL**, **LEFT** and **RIGHT** arrow keys work, and a left-click will position the cursor. Right button, **ENTER**, or **TAB** end edit and keep the changes. **ESC** ***will discard the changes to the cell.*** **Zoom/Pan** commands may be used during ***cell*** editing.

When ***no cell*** is selected, the ***right button*** or **ENTER** will ***end*** **DYNAMIC EDIT** keeping all changes, and **ESC** will **CANCEL**, ***discarding all changes***. Clicking on a different ***cell*** while editing one will keep any changes for that ***cell*** and start editing the new ***cell***.

Using the UP, DOWN, LEFT or RIGHT arrow keys to move to an adjacent cell

After pressing **ENTER** to terminate editing a cell, you may use the **UP**, **DOWN**, **LEFT** or **RIGHT** ***arrow keys*** to move in all four directions to a ***cell adjacent to the cell you just edited*** and immediately start editing that ***cell***.

This means you don't need to click the mouse unless you need to move to a ***non-adjacent cell*** for editing.

If you moved to the ***wrong cell*** using the **UP**, **DOWN**, **LEFT** or **RIGHT** ***arrow keys***, press **ENTER** to terminate editing on that ***cell*** and ***move to a cell adjacent to it*** with the **UP**, **DOWN**, **LEFT** or **RIGHT** ***arrow keys*** or click on another ***cell*** with the mouse.

Adjusting a COLUMNs Width Dynamically

When no cell is active for editing, you can left-click-hold and DRAG on the **RIGHT VERTICAL DIVIDER BAR** of a **COLUMN** and a **VERTICAL LINE CURSOR** will appear that can be moved to reposition it, and let the button up to set a new **COLUMN WIDTH**). When you release the mouse button, the table will redraw in the new size.

This new **COLUMN WIDTH** will update and appear in its '**COLUMN #**' dialog box if you use the **[Edit>Edit]** command on the **TABLE** later.

Deleting a Column

To **DELETE** a **COLUMN**, move the cursor over the **HEADER CELL** for that **COLUMN** and press **DEL**. To **DELETE** a **ROW**, move the cursor over **ANY CELL IN THE ROW** to be ***deleted*** and press the **DEL** key.

NOTE: The '**MINIMUM NUMBER OF ROWS**' setting in the **TABLE SETTINGS** dialog box will determine how deleting **ROWS** will behavior. You may **NOT DELETE** less than the number set in this option.

Inserting a Column

To **INSERT** a **COLUMN**, move the cursor over the **VERTICAL DIVIDER** and press **INS**. To **INSERT** a **ROW**, move the cursor over the **HORIZONTAL LINE** and press **INS**.

VERTICAL DIVIDERS take precedence over **HORIZONTAL LINES**– placing the cursor over an intersection picks the **VERTICAL DIVIDER**.

Entering Text into a Cell Using Dynamic Edit

This is normally how you will add **TEXT** to a newly created **TABLE**. After you place a **TABLE** in the drawing you may enter data into a ***blank cell*** or ***edit*** the data in a ***cell***. You select the **[Edit > Dynamic Edit]** command and click on the edge of the **TABLE BORDER** and it will highlight signifying that it has been selected for editing.

The illustration below shows a **TABLE** with the **TEXT** "**FastCAD7**" having just been typed with the red cursor bar next to the "7". The cell is highlighted in white when it is in **EDIT** mode. After **TYPING** or **EDITING** the **TEXT** press **ENTER**

You may use the **UP**, **DOWN**, **LEFT** or **RIGHT** ***arrow keys to move to an adjacent cell next the one last edited*** but ***you can only go one cell in any direction***. Use the mouse to ***click on another cell*** to **ENTER** or **EDIT** its **TEXT**. This is the usual method for entering **TEXT** into you **TABLE** after you create it.

Software Inventory List			
Software Title	Version	Serial #	Tech Support #
FastCAD7			

Table - Dynamically Scale or Drag

Use **SCALE**, or **DRAG's** ***dynamic*** **SCALE**, to dynamically grow the **TABLE SIZE**, including ***text height***, and the ***column widths***, all in correct proportion.

Use the **[Edit > Scale > Scale XY]** command to grow **width/height independently**.

Tables Explode when Exporting to DWG/DXF

Tables will **EXPLODE**, losing some **PEN THICKNESS** and **COLOR PROPERTIES** when they are **EXPORTED** to **AutoCAD** .**DWG** or .**DXF** files.

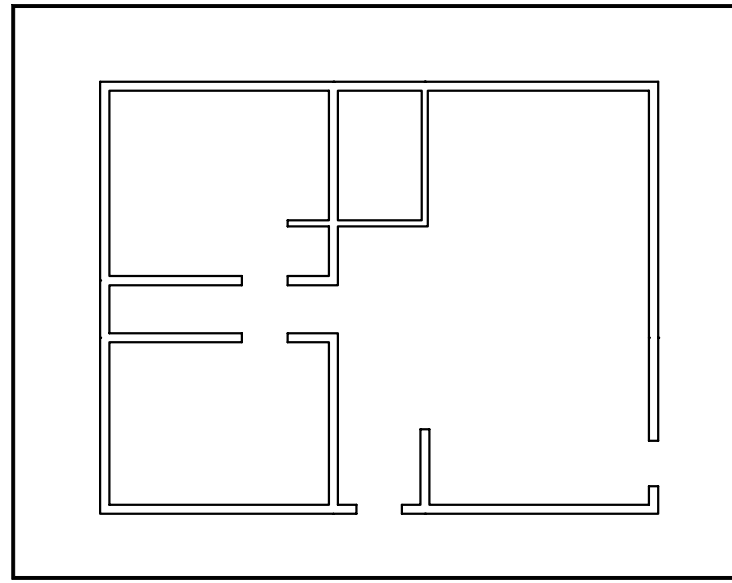
Using the Stretch command to Resize a TABLE

Because of complex interdependencies, **TABLE STRETCH** simply moves the **ENTIRE TABLE** if it's **ORIGIN** is included in the **STRETCH WINDOW** otherwise it leaves it unchanged. Use the **[Edit > Scale > Scale Value]** command, **[Edit > Scale > Scale XY]** command, or the **[Edit > Drag]** command to **RESIZE TABLES**.

Chapter 9

Drawing with Wall Networks

Architectural Drawings made easy



WALLS ***are a type of*** **FastCAD** ***entity incorporated as a node network, in which each wall span has an intelligent relationship with other related walls. They also interact intelligently with symbols that have been designed to work with walls.***

These abilities can make the construction of architectural drawings extremely fast and easy.

Understanding Wall Networks

In some ways, a ***wall*** is like a ***smart path***. A ***path*** is a series of ***connected nodes***, and so is a ***wall***. But whereas a ***path*** is a sequence of otherwise unrelated nodes, a ***wall*** is a ***network of related points***. ***All spans in a wall have awareness of the others***. Additional ***wall spans*** can be added to a ***wall network*** in almost any ***direction*** or ***length***, and attached at almost any point. ***Openings or gaps*** can be ***cut*** or ***removed*** from any ***wall span***.

The great advantage of walls is that intersections are automatically cleaned up. When you create corners, ***T-intersections***, or ***four-way intersections***, ***no tedious or repetitive breaking or trimming is required***. **FastCAD**'s brain does it all for you.

Note too that ***walls***, of course, ***have width***. The ***width*** is assigned to each ***span at each of its two nodes***. This allows ***wall segments within the same network to have different thickness***. ***Also, a wall segment can have a different thickness at each end of its span.***

Since a ***wall network is a homogenous entity***, it can be assigned ***only one line style*** and ***fill style***. Other entity properties, such as ***pen thickness for its edges***, ***color***, ***layer***, etc., follow suit.

Drawing Walls

Wall networks contain only one type of entity: ***walls***. There are three basic methods of creating ***spans*** within the ***network***:

1. **New Walls**, **using the WALL command**

   Each use of the ***wall*** command creates a ***new wall network***. You can draw ***spans*** individually, or in a connected series.

2. **Added Walls, using the ADD TO command**

   Use the **ADD TO** command to ***add new spans*** to an existing ***wall network***. Added ***spans*** will associate themselves to the selected ***wall network***, ***so all intersection and corner cleanups will occur automatically.***

3. **Rooms, using the ROOM command**

   Rooms are a special constructive subset of ***walls***. When you select a ***room*** command, you use a command similar to the **BOX** command to ***create a rectangular, ortho-locked, 4-node closed wall with two diagonal corner picks***.. ***Rooms*** can be added to an existing ***wall network***, or drawn separately.

Editing Walls

An ***existing wall network*** can be manipulated using the following actions:

- **Add new spans**
- **Remove spans**
- **Cut gaps in spans**
- **Change the width of a span—either for the entire network, a single span, or any one end of any span.**

Nodes

Each span in a wall network is defined by two nodes, or endpoints. Nodes exist at every endpoint, corner, or intersection. When you work with ***walls***, you never directly manipulate nodes. However, everything is indirectly performed at this level. Understanding the ***node network*** will help your understanding of ***walls***.

You can makes nodes visible by switching on frames. To switch on frames, type **CTRL+F** or selectBso a checkmark appears by the menu entry.

With frames on, each node appears as a gray "**x**" where present. You'll see that nodes are automatically created when you cut a gap into a span, or when you intersect spans.

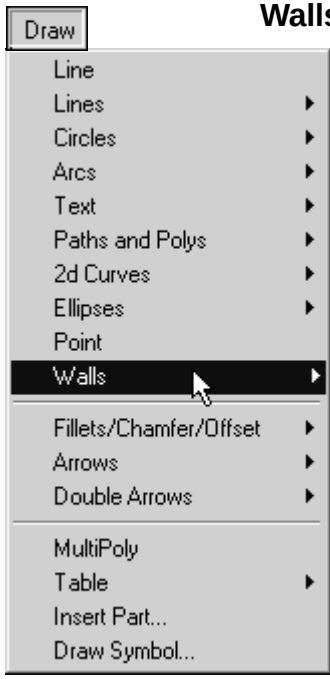
Working with Smart Symbols

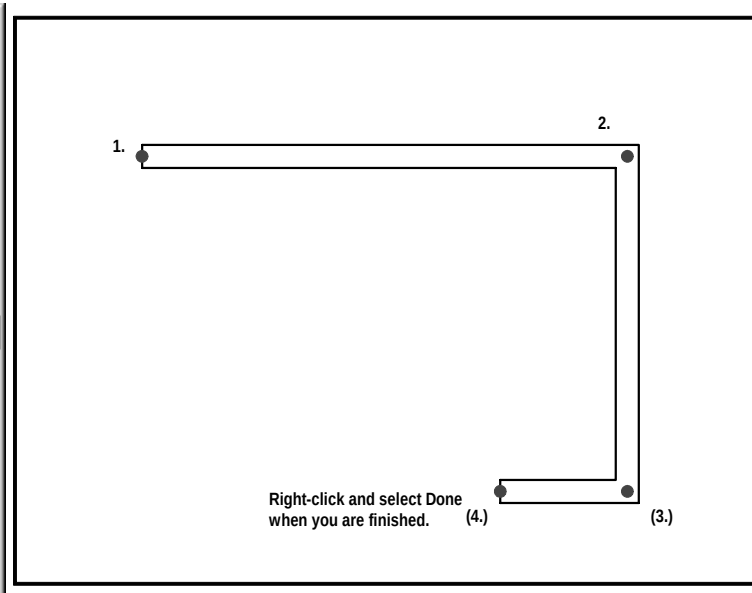
Smart symbols add a level of automated intelligence to symbol placement. If you are drawing with wall networks, you certainly want to consider using smart symbols.

Smart symbols, when inserted, automatically align themselves with the span into which they are inserted. Also, they can automatically "**cut**" and trim off the wall upon insertion.

Smart symbols can be easily created with the simple addition of control points to the symbol definition. Not only does this permit easy symbol creation, but existing symbols can also be converted rapidly into ready-to-use smart symbols.

Walls > Wall





Use the* WALL *command to create a new wall network. Walls you draw will use the width specified with the **SET WIDTH** command.

To draw a new wall network, type **WALL** at the command prompt, or select **[Draw > Walls > Wall]** from the menu:

1. The prompt reads "**1st corner:**". Select the first point using the crosshairs, or type numeric coordinates.

   FastCAD displays a rubber-band cursor anchored at the first point.

2. The prompt reads "**Next corner:**". Select a node point on the wall to create a span.

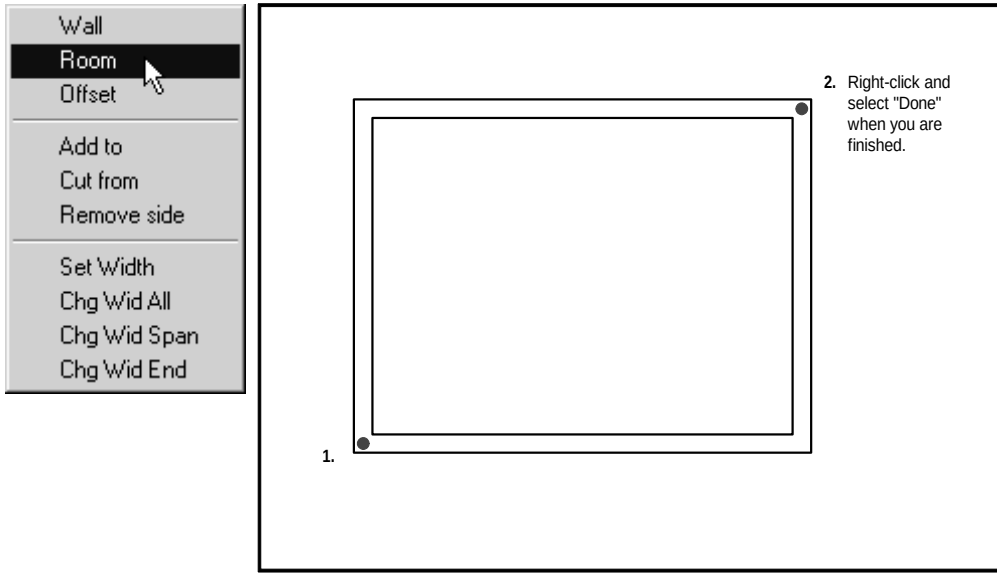
   FastCAD draws one span, using the current wall width. Note how the end of the wall is automatically capped. If you continue adding wall segments, the intersections are automatically cleaned up, even if you eventually return to the first point.

3. The prompt reads "**Next corner:**". If desired, continue to select nodes. After placing the last point of the path, right-click and choose **Done** from the ***Walls*** *popup menu*.

   Notice that anytime during the wall drawing process, a right-click will display the ***Walls*** *popup menu*. This control lets you quickly select the next action to take.

Text equivalent: ***WALL***

Walls > Room



The* ROOM *command lets you add new rooms to your drawing. A ***room*** is exactly the same construct as a ***regular wall*** entity, except that it is created differently. In the same way that the **BOX** command draws a ***rectangular polygon*** entity, **ROOM** draws a ***rectangular wall*** shape by ***picking two diagonal corners***.

Rooms can be added to an ***existing wall network***, or as a ***new separate wall network***. When you add a ***room*** to an existing ***wall network***, it works similarly to the **ADD TO** command. Added ***rooms*** will merge themselves to the existing ***wall network***, s***o that intersections are automatically cleaned up***. ***Walls*** added as "**new**" ***are not associated with any existing wall network.***

Once a ***room*** is drawn, it can be edited the same as any other ***wall*** entity.

To draw a ***new room***, select **[Draw > Walls > Room]** from the menu:

1. The prompt reads "**Pick existing wall [new]:**". With the pick cursor, select an existing wall network you wish to expand.

If no network currently exists, or if you wish to create a new wall network starting with a room, you may instead right-click. A right-click starts a new **ROOM** command.

2. The prompt reads "**Corner:**". Use the mouse to select one corner of the ***wall***. If you picked an existing ***wall*** in the first step, select a point on the selected network. If you "**eyeball**" the point, **FastCAD** automatically snaps to the nearest point within the pick box.

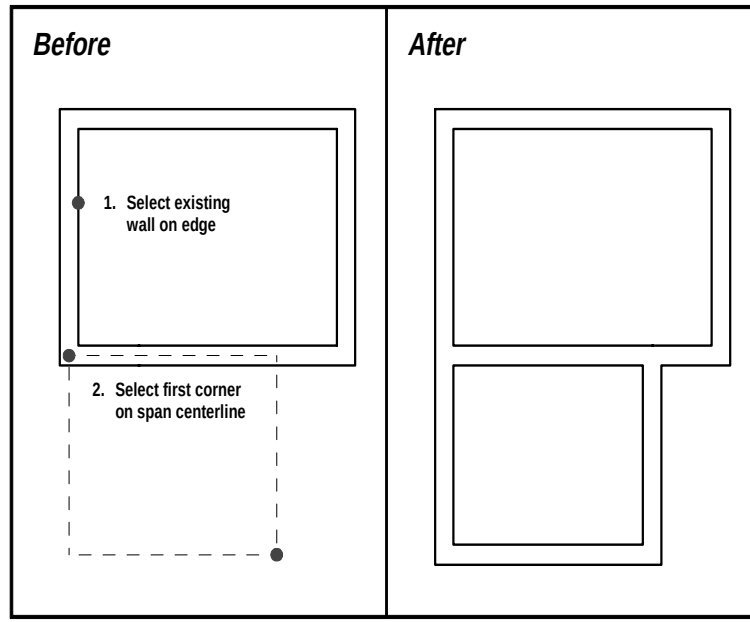
3. The prompt reads "**Opposite corner:**". Use the mouse to select the next point, completing the ***room***. ***A room added to an existing wall network will automatically clean up at all intersections.***

   EXAMPLE: For the diagonal corner of the ***room*** you may use a ***relative x,y length*** for an exact size ***room.*** If you wal to draw a **14'x12'** ***room***, place the 1st corner with a left-click and for the opposite corner type **@14',12'** and press **ENTER.** You now have a **14' horizontal** by **12' vertical** rectangular ***wall network***. You may now use the **CUT FROM** command to cut doorways and window openings. This is a quick way to lay out a floor plan.

4. The prompt reads "**Corner:**" again. You can continue drawing additional ***rooms***. ***Rooms*** added in this manner will be associated with the current wall network.

   When you are done adding ***rooms***, right-click and select **Done** from the ***Walls*** *popup menu* .

Adding rooms to existing networks



Adding ***new rooms*** to existing ***wall networks*** is easy. In the first step, select an existing ***wall network*** by picking a point on any edge. To seamlessly attach the added ***room***, be sure to pick one corner point along an ***existing wall's centerline.*** To help pick this point, you can turn on frames (**CTRL+F**) . If you pick close to a ***wall node***, **FastCAD** will automatically snap you to the precise point.

Text equivalent: ***ROOM***

Walls > Offset

WALL > OFFSET ***creates a parallel copy of the existing wall network, offset by the specified distance***. ***The offset copy determines its shape from the outside edge of the selected wall***. ***The offset distance is added to the existing wall to determine the width of the new wall***. For example, if the existing wall's width is **5.0**, and you specify an ***offset*** of **3.0**, the ***width*** of the offset ***wall network*** will be **8.0**.

The offset copy draws with the same entity properties as the original. However, the offset wall is a different network, so you can separately change its ***line style***, ***color***, etc., as you would any other entity.

Note that the standard **OFFSET** commands, found in the **Draw** menu' **FILLETS/ CHAMFERS/OFFSETS** sub menu, ***will not work on wall network entities***.

To ***offset and existing wall network***, select **[Draw > Walls > Offset]** from the menu:

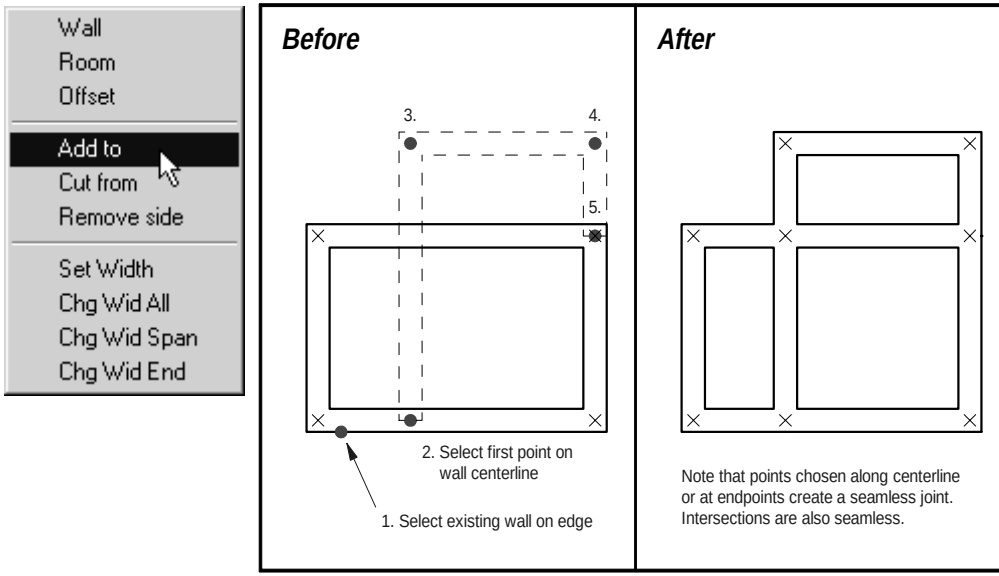
1. The prompt reads "**Wall to change:**". With the pick cursor, select the ***existing wall network*** you wish to ***offset***.
2. The prompt reads "**Offset distance:**". To specify an offset distance:
   - Use the mouse to visually pick two points the desired distance apart, or
   - Type a new value for the ***offset*** distance and press **ENTER**.

   As soon as you enter the value, the offset wall entity is created.
3. The prompt reads "**Offset distance:**". You can continue creating ***offset copies***, or right-click for the ***Walls*** *popup menu* and left-click the **Done** command.

If you continue, the value you specify will be added to the ***width*** of the last ***offset wall network***, not to the originally selected ***wall network***.

Text equivalent: ***WLOFST***

Walls > Add to



***The* ADD TO *command lets you add new spans to an existing wall network*. *Any span added to a wall network becomes part of that network*.**

To ***add a wall*** to an existing network, type **ADD TO** or select **[Draw > Walls > Add to]**:

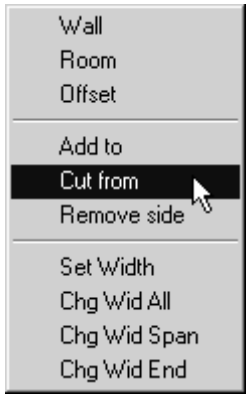
1. The prompt reads "**Pick existing wall [new]:**". With the pick cursor, select the existing wall network you wish to expand.

   If no network currently exists, or if you wish to create a new wall network, you may instead right-click. A right-click will cause the command to continue as the regular **WALL** command.

2. The prompt reads "**1st corner:**". Use the mouse to select the point on the network. If you "**eyeball**" the point, **FastCAD** snaps to the nearest node within the pick box.
3. The prompt reads "**Corner:**". Use the mouse to select the next point, completing the span. When you do, note that the ***new wall breaks*** and ***trims*** as necessary to form a clean intersection with the existing ***wall network***.
4. The prompt reads "**Corner:**". You can continue drawing additional spans. Because the ***spans*** are part of the ***existing network***, all intersections with the selected network are ***automatically cleaned up***. Whenever you are done adding walls, right-click and select **Done** from the ***Walls*** *popup menu*.

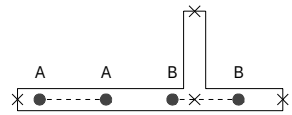
Because you are adding to an existing wall network, the ***new walls*** will disregard your current property setting and instead use those of the selected network. Such properties include ***color***, ***pen thickness***, **l*ine style***, ***fill style***, ***layer***, etc.

Text equivalent: WLADD



Walls > Cut from

The* CUT FROM *command lets you remove linear segments from existing spans. This command does not affect the resulting ***spans'*** associations with the original network. For example, if you have a network that consists of ***one span***, then use the **CUT FROM** to separate the ***span*** in the middle, the resulting two ***walls*** are still part of the same network.



The wall cut between points A-A is valid. However, the B-B cut is not valid because the two cut points cross the node present at the wall intersection.

The two points that define a ***cut*** must be made on the ***same span***. Remember that a ***span*** is the length between any two nodes. At every point where ***walls*** intersect, even at T-intersections, there is a network node. Therefore, ***cuts must be made between intersections.***

To ***cut a span segment***, tpye **CUT FROM** at the command prompt, or select **[Draw > Walls > Cut from]** from the menu:

1. The prompt reads "**Wall to change:**". Select the ***wall network*** you wish to ***cut***. You can select any point on the network; you do not have to specifically select the target ***span***.
2. The prompt reads "**First corner:**". Use the mouse to select the ***first cut point*** on the target ***span***.

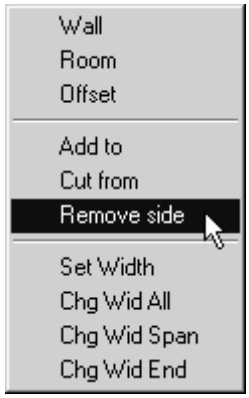
   FastCAD displays a rubber band cursor from the first point.
3. The prompt reads "**Next corner:**". Define the length of ***span to cut***. Use the mouse to select the ***second cut point*** on the ***target span***.

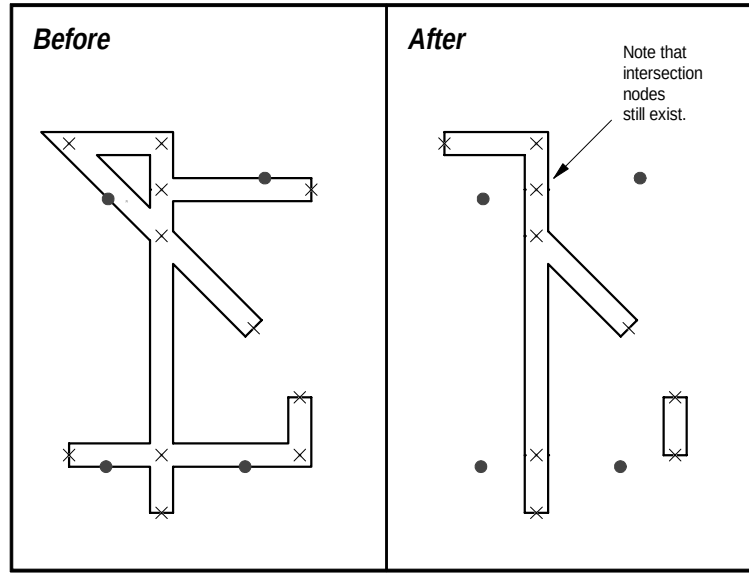
   FastCAD ***removes the defined segment from the span***.

   If the selected point is not valid, **FastCAD** displays an alert box telling you so. Re-select the point, ensuring that the selection point is within the ***wall*** boundary, and on the ***same span***.
4. The prompt reads "**First corner:**" again. If you are finished, right-click and choose **Done** from the ***Walls*** *popup menu*. You can instead choose to continue, as long as the cuts are on the ***wall network***. If you wish to continue cuts on different ***wall network***, you must rihgt-click for the ***walls*** popup menu and left-click on the **Done** command to finish the ***current cut operation*** and repeat the **CUT FROM** command with a right or left mouse click.

Text equivalent: ***WLCUT***

Walls > Remove Side





The* REMOVE SIDE *command lets you remove entire spans from an existing wall network*. *Remember that a span is the length between each node*. *Nodes occur at all endpoints and/or intersections.

To ***remove a span***, type **WLRMV** at the command prompt, or select **[Draw > Walls > Remove side]** from the menu:

1. The prompt reads "**Wall to change:**". Select the actual ***span*** you wish to ***remove***.

   FastCAD ***automatically removes the selected span and squares off the ends of the remaining segments in the network***.

2. The prompt reads "**Wall to chaange:**". You may continue selecting spans to remove, or right-click for the ***walls*** popup menu and left-click on the **Done command** to finish.

Text equivalent: ***WLRMV***

Walls > Set Width

The **SET WIDTH** command lets you set the width for all new spans drawn with the **WALL**, **ROOM**, or **ADD TO** command. After a new width is entered, the **SET WIDTH** commend defaults directly into the **ADD TO** command to allow immediate wall construction.

To set a new wall width, type **SET WIDTH** at the command prompt, or select **[Draw > Walls > Set Width]** from the menu:

1. The prompt reads "**Wall width [*n*]:**". Type in a new wall width in drawing units and press ENTER; or specify a distance on the screen, using the mouse. At this point if you do not enter a different width than the default displayed between the brackets and right-click or press **ENTER**, the command is terminated.

B The prompt reads "**Pick existing wall [*new*]:**". At this point, the command switches to the **ADD TO** command **[Draw > Walls > Add to]**. Any spans you draw now will use the new width. If you wish to end the command at any point, right-click and choose **Done** from the ***Walls*** *popup menu*.

Text equivalent: ***WLWID***

Walls > Chg Wid All

***The* CHANGE WIDTH ALL *command lets you change the width of an entire wall network*. *Every span in the network will have its width reset to the width you specify*.**

Be careful when using this command with ***wall*** systems where you have created ***spans*** of varying ***widths***. **CHANGE WIDTH ALL** will instantly reset those ***widths*** to the new value. If this happens, use the **UNDO** command to restore your drawing.

To set a ***wall network*** to specified ***new width***, select **[Draw > Walls > Chg Wid All]** from the menu:

1. The prompt reads "**Change width to [n]:**". Type in a ***new width*** in drawing units and press **ENTER**; or specify a distance on the screen using the mouse. At this point if you do not enter a different ***width*** than the default displayed between the brackets and right-click or press **ENTER**, the command is terminated.
2. The prompt reads "**Wall to change:**". Using the pick box, select any point on a ***wall network***.

   FastCAD immediately redraws the network using the specified ***width***.
3. The prompt reads "**1st corner:**". At this point, the command switches to the **ADD TO** command **[Draw > Walls > Add to]**. Any new spans you draw now use the ***width*** last specified with the **SET WIDTH** command.

   If you wish to quit at any time, right-click for the ***walls*** popup menu and left-click on the **Done** command.

Text equivalent: ***WCWIDA***

Walls > Chg Wid Span

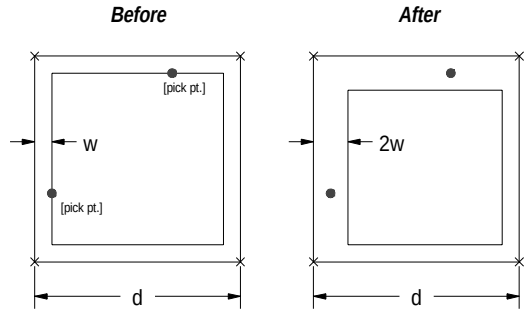
The **CHANGE WIDTH SPAN** command lets you change the width of a single span in a wall network.

To change the **width** of an existing **span**, select **[Draw > Walls > Chg Wid Span]** for the menu:

1. The prompt reads "**Change width to [n]:**". Type in a ***new width*** in drawing units and press ENTER; or specify a distance on the screen, using the mouse. At this point if you do not enter a different ***width*** than the default displayed between the brackets and right-click or press **ENTER**, the command is terminated.
2. The prompt reads "**Wall to change**:". Using the pick box, select the specific ***span*** you wish to modify.

   FastCAD immediately redraws the network, re-specifying the ***width*** of the targeted ***span***.
3. The prompt still reads "**Wall to change:**". Using the pick box, you can continue to select ***spans*** set to the ***new width***. Or to finish, right-click and select **Done** from the ***Walls*** *popup menu*.

Span widths and Left/Right edge locking



This illustration shows the effects of wall widths when walls are drawn with left or right edge locking. Note the location of the nodes.

Draw walls with edge locking by selecting Left or Right from the *Walls popup menu* (to display, right-click when drawing walls). "Left" or "right" corresponds with "inside" or "outside" depending upon orientation.

When ***walls*** are drawn with left or right edge locking, those edges remain fixed when ***new widths*** are applied to ***spans*** (or to the entire wall network). This ability lets you change the thickness of a ***wall*** without changing the inside or outside perimeter (whichever is fixed). In default center-locking mode, ***spans widen*** or ***narrow*** about the ***span*** centerline(s).

Text equivalent: ***WCWIDS***

Walls > Chg Wid End

The **CHANGE WALL WIDTH END** command lets you change the width at a single span's endpoint. A span can have different widths at each node, enabling you to create walls with a taper. The end width is the length measured at the endpoint, perpendicular to the direction of the span.

To change the end width of an existing span, select **[Draw > Walls > Chg Wid End]** from the menu:

1. The prompt reads "**Change width to [n]:**". Type in a new width in drawing units and press ENTER; or specify a distance on the screen, using the mouse. At this point if you do not enter a different width than the default displayed between the brackets and right-click or press **ENTER**, the command is terminated.
2. The prompt reads "**Wall to change:**". Using the pick box, select the specific span near the endpoint you wish to modify.

   FastCAD immediately redraws the network, re-specifying the end width of the targeted span.
3. The prompt still reads "**Wall to change:**". Using the pick box, you can continue to select spans, modifying their chosen endpoint widths. Or to finish, right-click and select **Done** from the ***Walls*** *popup menu*.

Text equivalent: ***WCWIDE***

Walls popup menu

Walls popup menu

Notice that anytime during the ***wall drawing process***, a right-click will display the ***Walls** popup menu*. This control lets you quickly select the next action to take. Most drawing operations require that you select **'Done'** from this menu to complete the command.

Next Corner Returns to the "**Next point:**" prompt, allowing you to add more wall segments.

Break Allows you to interrupt the current wall path, and draw a new wall path that is still part of the same network. Walls that are drawn using break will intersect cleanly with the active network.

add Room This is a shortcut to the **ROOM** command, except that it defaults into add mode and auto-selects the wall you are currently drawing or editing.

Done Finishes drawing the current wall network. If you wish to add to the finished network, you must use the **WALL>ADD TO** command. The regular **WALL** command starts a new network that is independent of the existing network.

Cancel Selecting this will cancel the active draw/edit operation. Note that a **Cancel** is not the same as an **UNDO**. A cancelled operation cannot be recovered.

Remove This works identically to the **REMOVE SIDE** command. To erase a span, select a span from any network. The wall segment will be deleted without affecting the remaining network, even if it has been separated into separate networks that share no common nodes.

Cut This is a shortcut to the **WLCUT** command, to work on the wall network you are currently constructing. **WLCUT** removes a specified section from a span. When invoked, **FastCAD** prompts you to enter the two endpoints of the cut. Both cut points must be selected on the same span. **FastCAD** then removes that section from the network.

Offset This is a shortcut to the **WLOFST** command, to work on the wall network you are currently constructing. **WLOFST** creates a parallel copy of the existing wall network, offset from the outer edge by the specified distance. The offset copy draws with the same entity properties as the original. However, it is a different network, so you can separately change its line style, color, etc., as you would any other entity.

Left/Center/Right Select during wall construction to set the "**locking**" edge. When **Center** is selected, the nodes defined during the draw process define the centerline of each span. When **Left** or **Right** is selected, the nodes define the wall perimeter (inside or outside edge, depending on the orientation). Setting **Left** or **Right**

is helpful when the wall-to-wall distance is known. When wall widths are modified, the left/right wall edge will remain fixed; the wall reflects the new thickness by displacing the opposite edge.

Width > Child Menu

Width > Width This is a shortcut to the **WLWID** command **[Draw > Walls > Width]**. This command interrupts the current drawing process, and allows you to specify a new uniform wall width for walls drawn from that point. At the "**Wall width [*n*]:**" prompt, you can type in a new width or specify a new distance with the mouse. **FastCAD** then prompts you to "**Pick existing wall [*new*]:**". If you pick an existing wall, new walls with the new width will be added to the selected network. If you right-click, a new network will be started.

Width > Change All This is a shortcut to the **WCWIDA** command **[Draw > Walls > Chg Wid All]**. This command suspends the current draw operation and allows you to change the width of all spans in a selected network.

Width > Change Span This is a shortcut to the **WCWIDS** command **[Draw > Walls > Chg Wid Span]**. This command suspends the current draw operation and allows you to change the width of a single selected span.

Width > Change End This is a shortcut to the **WCWIDE** command **[Draw > Walls > Chg Wid End]**. This command suspends the current draw operation and allows you to change one end width of a single selected span.

Chapter 10

3D Draw Commands

Adding 3d entities and objects to your drawing

Commands under the ***Draw3 menu*** add **3d** entities to your drawing. These entities can be three-dimensional surface and solids (objects that occupy space, as opposed to flat objects). Solids can be hollow (like a shoebox), or solid through and through (like a brick). Some **FastCAD** solids (cylinders and domes, for example) let you specify thickness for their walls.

While **FastCAD** stores solid geometry for these objects, this version is not a solids modeling package. For example, if you move the viewpoint inside a cube, **FastCAD** renders it on-screen as if it were hollow.

Before you draw ***3d entities*** you should select the **[View > Std. 3d Views]** command to place ***four view windows*** on the drawing screen. The ***upper-left window*** is the ***top view***, the ***lower-left window*** **is the** ***front view,*** the ***lower-right window*** is the ***right side view*** and the ***upper-right window*** is the ***overall view looking down on the drawing from an upper-left oblique angle.*** This will help you better visualize the entities and pick points in ***3d space***

After **FastCAD** draws the ***3d entities***, select the **[View > Std. 3d Views]** command to ***center the view in each of the four windows***.

Note: The ***fillet***, ***chamfer***, and ***tangent*** commands work only with **2d**entities. In spite of this limitation, these commands are useful for drawing complex objects to be crosshatched and converted to **3d**.

3d Line

3d Path

3d Polygon

3d Line

3D LINE ***works like the two-dimensional line*** **2d LINE** ***command, except that you can specify points anywhere in the three-dimensional space. You can pick the point, or type*** **XYZ** ***coordinates. Depth mode may be used to to place*** **3d** ***points for 3d lines.***

Text equivalent: ***LINE3***

3d Path

3d PATH ***works like the*** **2d PATH** ***command, except that you can place nodes anywhere in three-dimensional space. You can pick a node in a drawing window, or type*** **XYZ** ***coordinates. Depth mode may be used to to place*** **3d** ***points for 3d paths***

Text equivalent: ***PATH3***

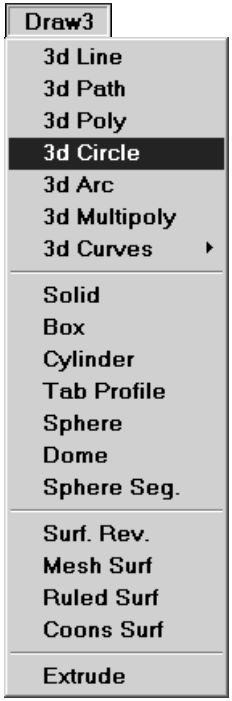
3d Poly command

3d POLY ***works like the*** **2d POLY** ***command, except that you can place nodes anywhere in three-dimensional space. You can pick a node in a drawing window, or type*** **XYZ** ***coordinates. Depth mode may be used to to place 3d points for each node od 3d polygons.***

Note: **3d** polygons work best if you place all the points in one plane (i.e., **make the polygon flat**).

Text equivalent: ***POLY3***

3d Circle



3d CiIRCLE ***works like the two-dimensional*** **CIRCLE** ***command, except that you can specify points anywhere in three-dimensional space.*** **3d CIRCLE** ***draws a circle using a specified center point and circumference point, each having an x,y,z coordinate value***.

Use the **[View > Std 3d views]** command to set up the standard ***top, front, right side and overall view*** *windows*. This makes it easier to pick points in ***3d space*** and see how the entities look from four different view angles.

To draw a **3d circle**, select **[Draw3 > 3d Circle]:**

1. ***The prompt reads*** Center point[0.00000,0.00000,0.00000]:

   To specify the **center point**:

   - Right-click or press **ENTER** to accept the default ***center;***
   - Pick a ***new center point*** or;
   - Type ***3d numeric coordinates*** and press **ENTER**.

2. ***The prompt reads*** Up direction from center [perp to last window]:

   To specify the ***center point***:

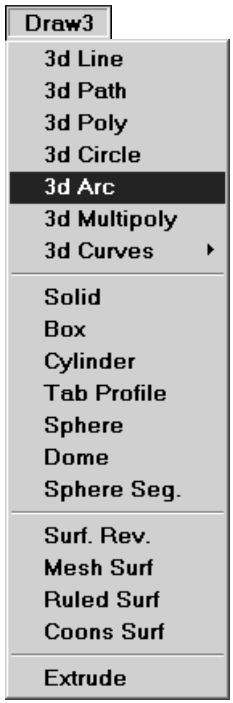
   - Right-click or press **ENTER** to accept the default ***up direction which will be perpendicular to the last active window surfave***.
   - Use the dynamic cursor as a visual aid to pick a ***direction in any window*** that will be the ***center axis of the 3d circle***. ***It will always be perpendicular to the circle***. ***You can think of this as the axel if the circle was a wheel.***
   - Type ***3d numeric coordinates*** and press **ENTER**

3. ***The prompt reads*** Point on circle:

   Use the dynamic cursor as a visual aid to pick a point on the screen. When you pick a point, **FastCAD** draws the ***3d circle*** and the command terminates.

Text equivalent: ***CIR3D***

3d Arc



3d ARC ***works like the two-dimensional*** **ARCS** ***command, except that you can specify points anywhere in three-dimensional space***. **3d ARC** ***draws an arc from a specified center point, an up direction for its axis, start point*** **(which defines both the radius and the starting angle)**, ***and end point***.

Use the **[View > Std 3d views]** command to set up the standard ***top, front, right side*** and ***left oblique overall view*** *windows*. This makes it easier to pick points in ***3d space*** and see how the entities look from four different view angles.

To draw a **3d arc**, select **[Draw3 > 3d Arc]:**

1. ***The prompt reads*** Center point[0.00000,0.00000,0.00000]:

   To specify the ***arc center***:

   - Right-click or press **ENTER** to accept the default ***center;***
   - Pick a ***new center point*** or;
   - Type ***3dnumeric coordinates*** and press **ENTER**.

2. ***The prompt reads*** Up direction from center [perp to last window]:

   To specify the ***center point***:

   - Right-click or press **ENTER** to accept the default ***up direction which will be perpendicular to the last active window***.
   - Use the dynamic cursor as a visual aid to pick an ***up direction in any window*** that will be the ***center axis of the*** **3d Arc**. ***It will always be perpendicular to the arc***. ***You can think of this as the axel if the Arc was part of a wheel.***
   - Type ***3d numeric coordinates*** and press **ENTER**

3. ***The prompt reads*** Arc starting point:

   This tells **FastCAD** both the ***Arc radius*** and ***its starting angle***. To specify the ***starting point***:

   - Pick a point or;
   - Type ***3d numeric coordinates*** and press **ENTER.**

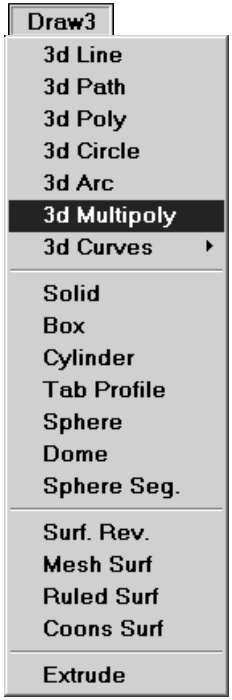
4. ***The prompt reads*** Arc ending point:

   To specify the ending angle, pick a point along the radial dynamic cursor.

FastCAD draws the ***arc counterclockwise from the starting point to the ending point***, and the command terminates.

The plane of the Arc is parallel to that of the window in which the center was digitized, passing through the center point. **The point on the circle becomes the T=0 point of the Arc.**

Text equivalent: ***ARC3D***



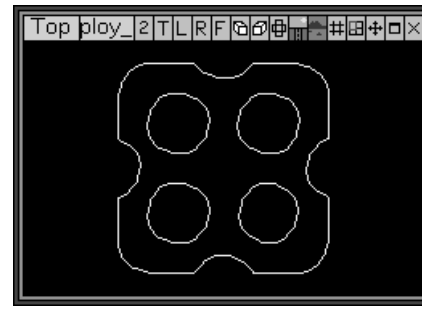
3d Multipoly

3d MULTIPOLY (**Multiple Polygons**) ***works like a*** **2d MULTIPOLY** ***but each coordinate point has*** **XYZ** ***coordinates. Defining a*** **MULTIPOLY** ***collects existing entities and turns them into a new entity.*** **3d MULTIPOLYs** ***can be filled with fill style although when a*** **MULTIPOLY** ***is*** **EXTRUDED** ***the fill becomes*** **SOLID.**

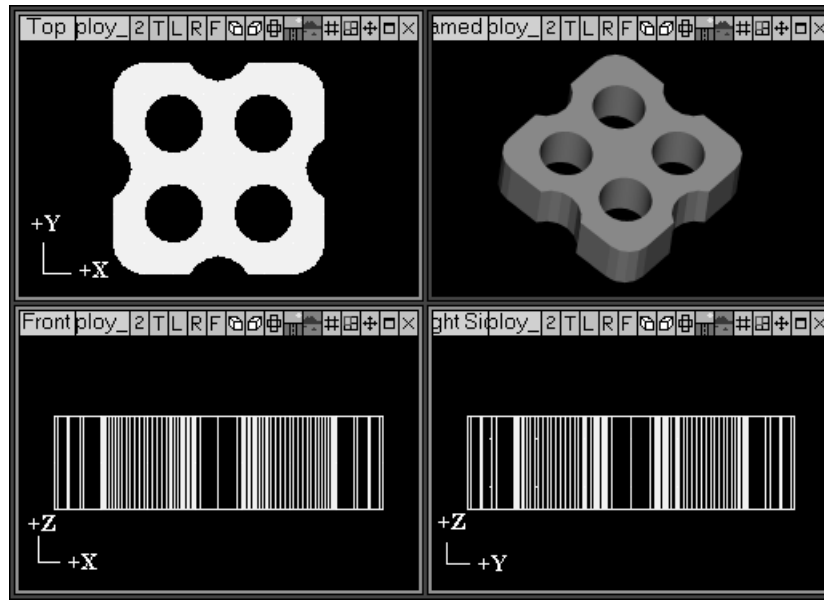
The outer perimeter of a **MULTIPOLY** must be closed with each entity meeting ***endpoint*** to ***endpoine***.

An enclosed entity inside a filled area will leave a hole or unfilled area. The entities that can be included in a **MULTIPOLY** *are* ***lines***, ***arcs***, ***circles***, ***ellipses***, ***paths***, ***polygons***, ***splines***, ***text***, and ***existing*** **MULTIPOLYS**

Below is an example of a **3d** ***multipoly*** made of ***lines, arcs and circles.***



It was then **EXTRUDEd two inches** ***vertically*** into the **Z axis** as seen below.



Std. 3d Views

3d Icons bar

Draw the entities that will be included in the 3d Multipoly

1. Before defining the **MULTIPOLY**, the entities were drawn. Four ***circles***, four ***lines*** for the perimeter then the **FILLET & TRIM** command was used on the four ***lines*** to make the ***rounded corners***. Four ***arcs*** were drawn against the inside edges of the four ***lines*** and the **BREAK** command was used to remove the line segments that crossed theend of each ***arc***.

Define the 3d Multipoly

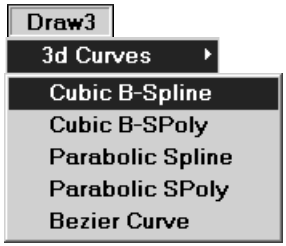
2. Select the **[Draw3 > 3d Multipo;y]** command and select all the entities that will be included in the ***3d multipoly***.
3. The **STD. 3D VIEWS** icon was clicked in the ***3d icons bar*** to set up the standard ***top, front, right side and Left oblique view*** *windows as seen below*. This makes it easier to pick points in ***3d space*** and see how the entities look from four different view angles.
4. The ***material*** of ***metallic gold paint*** was assigned to the ***multipoly*** by using the **{edit > hange > Material]** command.
5. The **EXTRUDE** command was used to ***extrude*** *it vertically from* **(0,0,0)** *to* **(0,0,2")** *in the* **Z axis.**
6. The ***Render view on/off*** icon in the ***upper-right view window***, as seen below, was toggled to render it using the **Light Works** renderer with a ***metallic gold paint material*** applied to it as seen in the illustration below.

Printing the Rendered Image

This ***rendered image*** could be a part for s machine. The ***rendered image*** may be exported the **BMP, TIF, JPG, Post Script** or **TGA** file formats for opening in another application such as a word processor and printed or directly from **FastCAD** to you printer.

Text equivalent: ***MPOLY3***

3d Cubic B-Spline



3d CUBIC B-SPLINE ***works like the two-dimensional*** **SPLINE** ***command, except that you can specify points anywhere in three-dimensional space.*** **3d Cubic B-Spline** (**AutoCAD® smooth polyline**) ***draws an open smooth curve defined by a series of node points called a frame. The curve touches the first and last points and follows an average path among the other nodes. A*** **3d Cubic B-Spline** ***must have at least 3 nodes.***

Use the **[View > Std 3d views]** command to set up the standard ***top, front, right side and overall view*** *windows*. This makes it easier to pick points in ***3d space*** and see how the entities look from four different view angles.

To draw a **3d Cubic B-Spline,** select **[Draw > 3d Splines > Cubic B-Spline]:**

1. The prompt reads "**1st point:**". To specify the ***first spline node***, pick a point, or type ***3d numeric coordinates***.

   FastCAD displays a rubber-band cursor, anchored at the ***1st point***.

2. The prompt reads "**Next point [Done]:**". Specify the ***second node*** in the ***3d spline***.

   FastCAD displays a dynamic cursor showing both the curve and the frame.

3. The prompt reads "**Next point:**". Continue to specify nodes as desired. Right-click to ***end node selection***.

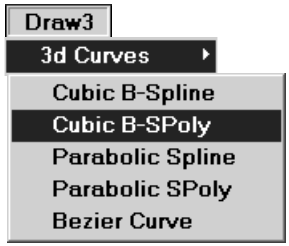
   FastCAD draws the ***3d spline*** and terminates the command.

Using frames

After the curve is drawn, **FastCAD** normally hides the frame that you used to define the spline. To force display of frames on or off throughout your drawing, select **TOGGLE FRAMES** **[Specs > Toggle Frames]**, or press **CTRL+F** on the keyboard.

Text equivalent: ***SPLINE3***

3d Cubic B-Spoly



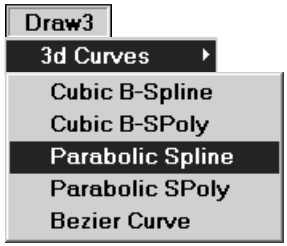
3d CUBIC B-SPOLY ***works like a*** **2d CUBIC B-SPOLY** ***but each point has*** **XYZ** ***coordinates.***

Use the **[View > Std 3d views]** command to set up the standard ***top, front, right side and overall view*** *windows*. This makes it easier to pick points in ***3d space*** and see how the entities look from four different view angles.

Depth mode may be used to to place **3d** points for each node of a ***3d cubic b-spoly***.

*Text equivalent:**SPOLY3***

3d Parabolic Spline



3d PARABOLIC SPLINE ***works like the two-dimensional*** **Parabolic Spline** ***command, except that you can specify points anywhere in three-dimensional space.*** **3d Parabolic Spline** ***draws an open, curved shape defined by a series of node points called a frame. The curve draws through all the nodes. A*** **3d Parabolic Spline** ***must have at least 3 nodes.***

Use the **[View > Std 3d views]** command to set up the standard ***top, front, right side and overall view*** *windows*. This makes it easier to pick points in ***3d space*** and see how the entities look from four different view angles.

To draw a **3d Parabolic Spline,** select **[Draw > 3d Splines > Parabolic Spline]:**

1. The prompt reads 

   To specify the ***first spline node***, pick a point or type ***3d numeric coordinates***.

   FastCAD displays a rubber-band cursor, anchored at the ***1st point***.

2. The prompt reads 

   Specify the ***second node*** in the ***spline***.

   FastCAD displays a dynamic cursor showing both the curve and the frame.

3. The prompt reads "**Next point:**". Continue to specify nodes as desired. Right-click ***to end node selection***.

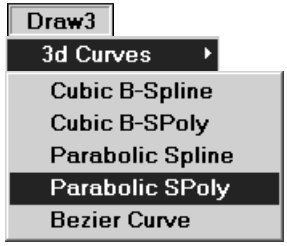
   FastCAD draws the ***parabolic spline*** and terminates the command.

Using frames

After the curve is drawn, **FastCAD** normally hides the frame that you used to define the parabolic spline. To force display of frames on or off throughout your drawing, select **TOGGLE FRAMES** **[Specs > Toggle Frames]**, or **press CTRL+F** on the keyboard.

Text equivalent: ***PSPLINE3***

3d Parabolic SPoly



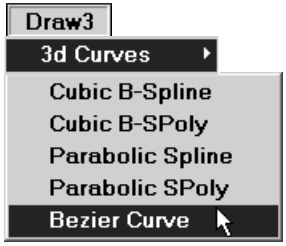
3d PARABOLIC SPOLY *works like a* 2d PARABOLIC SPOLY *but each point has* XYZ *coordinates.*

Use the **[View > Std 3d views]** command to set up the standard ***top, front, right side and overall view*** *windows*. This makes it easier to pick points in ***3d space*** and see how the entities look from four different view angles.

Depth mode may be used to to place **3d** points for each node of a ***3d parabolic spoly***

Text equivalent: ***PSPOLY3***

3d Bezier Curve

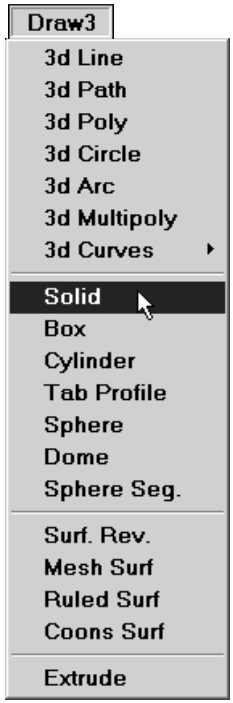


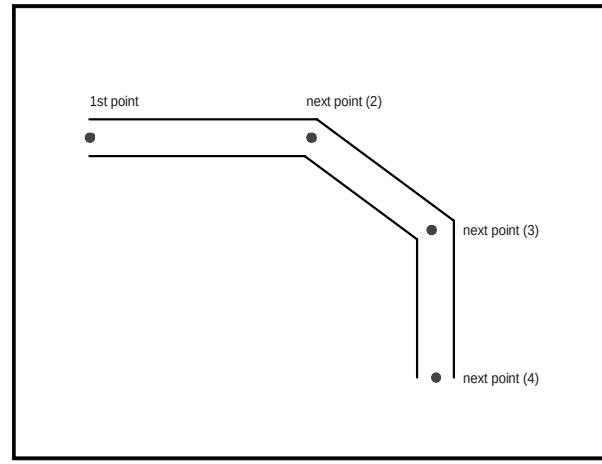
3d BEZIER CURVE ***works like a*** **2d BEZIER CURVE** ***but each point has*** **X,Y,Z** ***coordinates.***

Use the **[View > Std 3d views]** command to set up the standard ***top, front, right side and overall view*** *windows*. This makes it easier to pick points in ***3d space*** and see how the entities look from four different view angles.

Text equivalent: ***BEZ3***

Solid





SOLID ***draws three-dimensional solid objects anywhere in*** **3d** space. ***A*** **FastCAD** ***solid includes a number of flat sections*** (**polygons with one or more nodes**). ***Each section can have a different number of nodes.*** **FastCAD** ***connects each section to the nest automatically, using*** **3-** ***or*** **4-*****sided panels as necessary.***

The prompt reads "**Section start [end solid]:**". Enter the first **3d** point. The prompt reads "**Next node [end section]:**". Enter a series of points, just as for a ***polygon***. When you finish the first section, right-click to end the section. Enter additional sections the same way. To complete the ***solid***, right-click or press ENTER when you see "**[end solid]**" in the prompt.

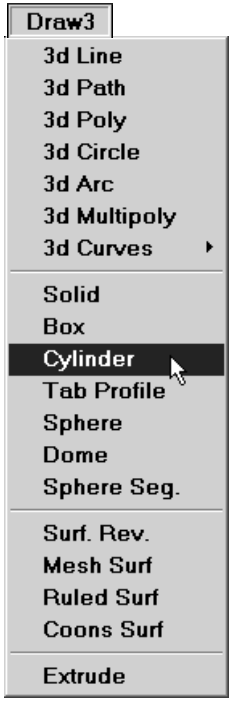
Text equivalent: ***SOLID***

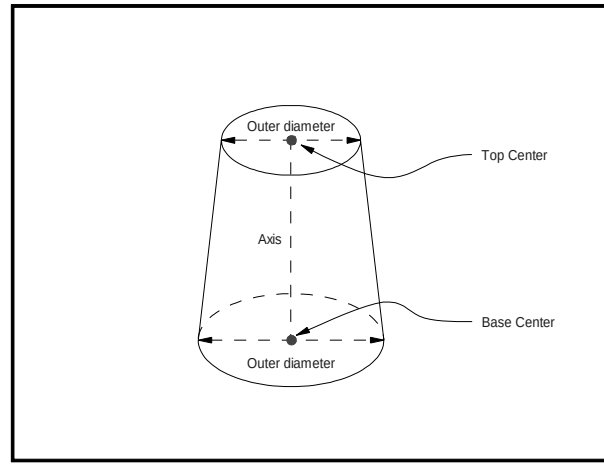
3d Box

3d BOX ***is a shortcut command that draws a block-shaped*** **SOLID** ***entity.*** **FastCAD** ***asks you to select two corner points. Pick the points in*** **3d** ***space, or type x,y,z coordinates.***

Text equivalent: ***BOX3***

Cylinder





CYLINDER ***draws right cylinders, rings, and cones at any angle***. ***Cylinder entities have two circular or ring-shaped ends that can have differing diameters.*** **FastCAD** ***tilts the ends so they are at right angles to the axis*** (t**he line from base center to top center**). ***You can give the cylinder walls thickness by specifying a different inner and outer radius. Either end can have a diameter of*** **0.0 to make a cone, and either or both ends can be closed.**

Note: Before you draw ***3d entities*** you should select the **[View > Std. 3d Views]** command to place ***four view windows*** on the drawing screen. The ***upper-left window*** is the ***top view***, the ***lower-left window*** **is the** ***front view,*** the ***lower-right window*** is the ***right side view*** and the ***upper-right window*** is the ***overall view looking down on the drawing from an upper-left oblique angle.*** This will help you better visualize the entities and pick points in ***3d space***

To draw a **cylinder**, select **[Draw3 > Cylinder]**.

1. The prompt reads "**Bottom center:**". Specify a center for one circular end.

2. The prompt reads "**Top center:**". Specify a center for the opposite circular end.

B The prompt reads "**Bottom radius [x]:**", where x is the default or prior radius. Right-click to accept the default, or type in a new radius and press ENTER.

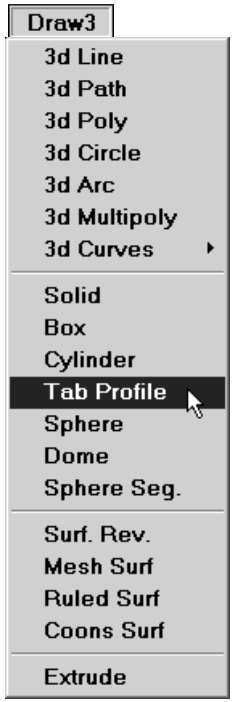
4. The prompt reads "**Top radius [x]:**", where x is the default or prior radius. Right-click to accept the default, or type in a new radius and press **ENTER**.

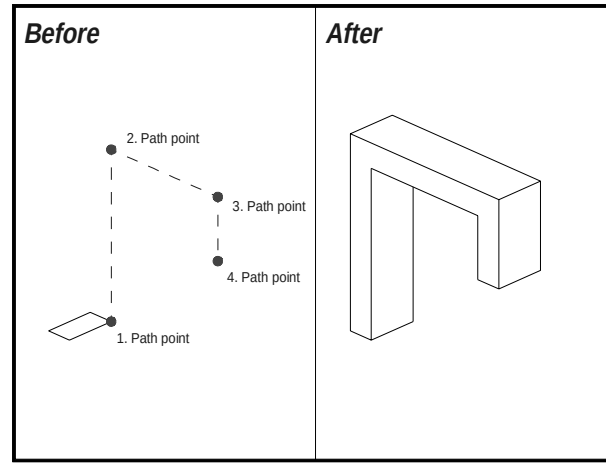
FastCAD draws the ***cylinder*** and the command terminates.

Note: After **FastCAD** draws the ***3d entities***, select the **[View > Std. 3d Views]** command to ***center the view in each of the four windows***.

Text equivalent: ***CYL***

Tab Profile





TAB PROFILE (**Tabulated Profile**) ***draws a connected chain of sections based on profile entities*** (**edges**) ***that you select. These profiles can include*** **2d** ***or*** **3d** ***circles, arcs, lines, points, polygons, splines, smooth polygons, and paths. You draw a path for the tabulated profile to follow, and specify optional scaling factors for the sections at each node. Either or both ends can be closed. Tabulated profiles are especially useful for representing piping, ducts, and so on.***

Note: Before you draw ***3d entities*** you should select the **[View > Std. 3d Views]** command to place ***four view windows*** on the drawing screen. The ***upper-left window*** is the ***top view***, the ***lower-left window*** **is the** ***front view,*** the ***lower-right window*** is the ***right side view*** and the ***upper-right window*** is the ***overall view looking down on the drawing from an upper-left oblique angle.*** This will help you better visualize the entities and pick points in ***3d space***

To draw a **tab progile**, select **[Draw3 > Tab Profile]**.

1. The prompt reads "**Tab path point [done]:**". Pick a starting point for the path. (Usually you'll pick a point on or in the center of the edge entity.) For each point, **FastCAD** asks for a "**Node scale [1.000]:**". Press ENTER to accept the default (for a section the same size as the original edge entity), or type a scale factor.

   Right-click or press ENTER to stop adding path points.

2. Finally, select one or more edges as you would for an editing command. FastCAD ignores any entities that don't work as profiles. **FastCAD** draws the tabulated profiles (one for each edge entity you selected). The last section is perpendicular to the last leg of the path.

Note: Although you can create any number of ***tabulated profiles*** at once, this command is easier to work with if you select only one edge entity and begin the path on or in the center of that entity. If you apply scaling to any of the nodes, **FastCAD** scales all the edges around that node. In other words, if you selected more than one edge, the tabulated profiles will converge toward the path.

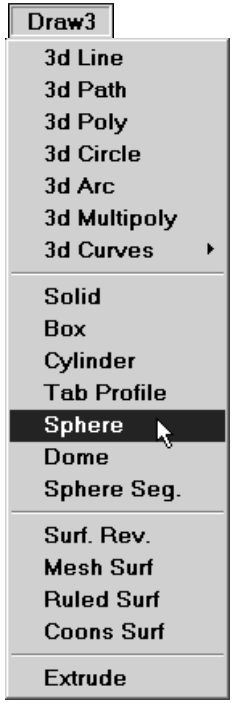
After **FastCAD** draws the ***3d entities***, select the **[View > Std. 3d Views]** command to ***center the view in each of the four windows***.

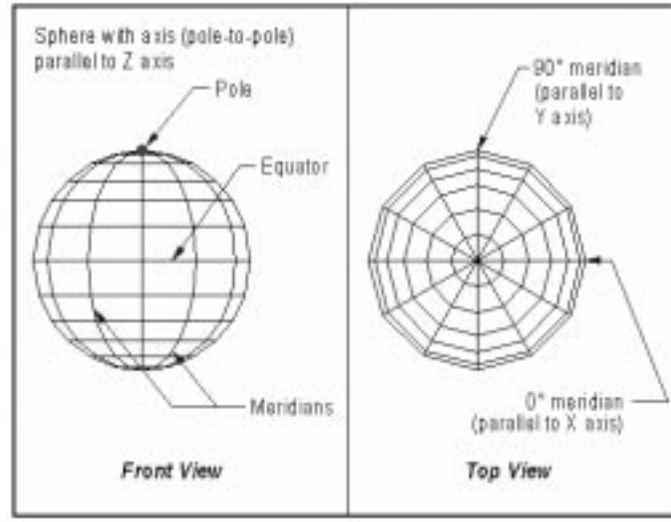
Editing tabulated profiles:

If you edit a ***tabulated profile*** with **SCALE XYZ**, **FastCAD** applies the scaling to the first section of the profile (the original edge entity), then copies the scaled edge to all the other sections along the tabulation path. The tabulation path itself is also scaled.

Text equivalent: ***TPRO***

Sphere





SPHERE ***draws spheres***

Note: Before you draw ***3d entities*** you should select the **[View > Std. 3d Views]** command or the **[File > New 3d Drawing]** command to place ***four view windows*** on the drawing screen. The ***upper-left window*** is the ***top view***, the ***lower-left window*** **is the** ***front view,*** the ***lower-right window*** is the ***right side view*** and the ***upper-right window*** is the ***overall view looking down on the drawing from an upper-left oblique angle.*** This will help you better visualize the entities and pick points in ***3d space***

To draw a **sphere**, select **[Draw3 > Sphere]**.

1. ***The prompt reads*** Sphere center:

   Pick a point in **3d** space or type a ***3d coorinate..***

2. ***The prompt reads*** Pole point or diameter [1.00000]:

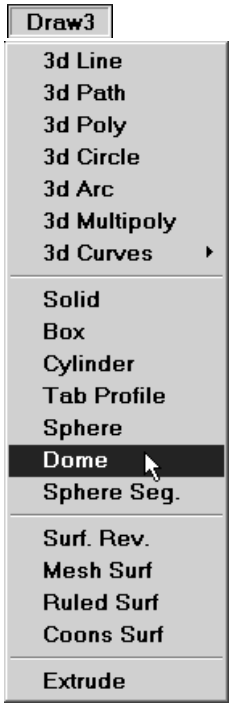
   - Pick a point in any window. This point determines the ***radius*** and becomes the "**north**" pole of the ***sphere***.
   - Type a ***diameter***. **FastCAD** automatically places the north pole directly above the center, so the polar axis is perpendicular to the **XY** plane.

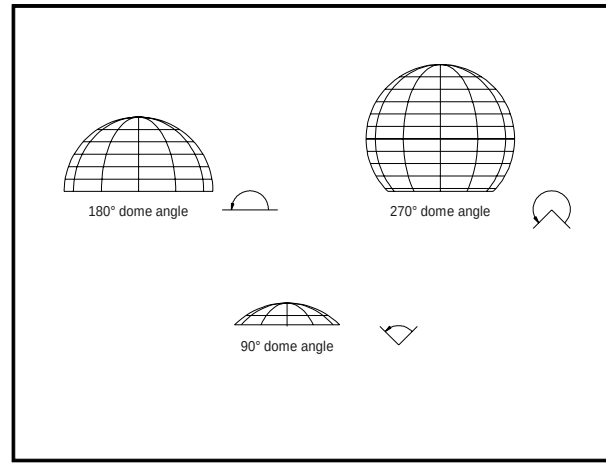
FastCAD draws the ***sphere***.

Note: After **FastCAD** draws the ***3d entities***, select the **[View > Std. 3d Views]** command to ***center the view in each of the four windows***.

Text equivalent: ***SPHERE***

Dome





DOME ***draws a partial sphere or partial ellipsoid.***

Note: Before you draw ***3d entities*** you should select the **[View > Std. 3d Views]** command to place ***four view windows*** on the drawing screen. The ***upper-left window*** is the ***top view***, the ***lower-left window*** **is the** ***front view,*** the ***lower-right window*** is the ***right side view*** and the ***upper-right window*** is the ***overall view looking down on the drawing from an upper-left oblique angle.*** This will help you better visualize the entities and pick points in ***3d space***

To draw a **dome**, select **[Draw3 > Dome]**.

1. The prompt reads "**Sphere centert:**". Pick a point in ***3d space*** in any window..

2. The prompt reads "**Pole point or diameter [1.00000]:**". You can:

- Pick a point in any window.. This point determines the ***radius*** and becomes the "**north**" pole of the dome.
- Type a ***diameter***. **FastCAD** automatically places the north pole directly above the center, so the polar axis is perpendicular to the **XY** plane.

3. The prompt reads "**Dome angle [180°]:**". You can right-click to accept the drfault::

- Type an ***angle***. **FastCAD** automatically points the 0° meridian along the **X** axis.

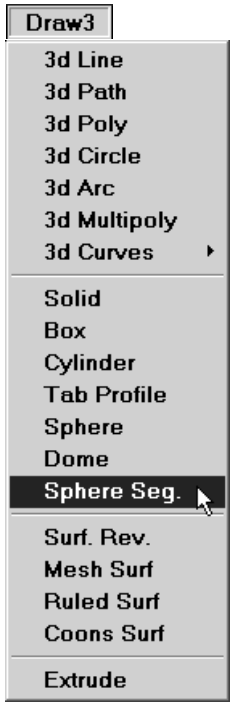
Note: Dome angle is part of a full circle-for example, a half-sphere dome has an angle of **180°** (not **90°** as on most globes).

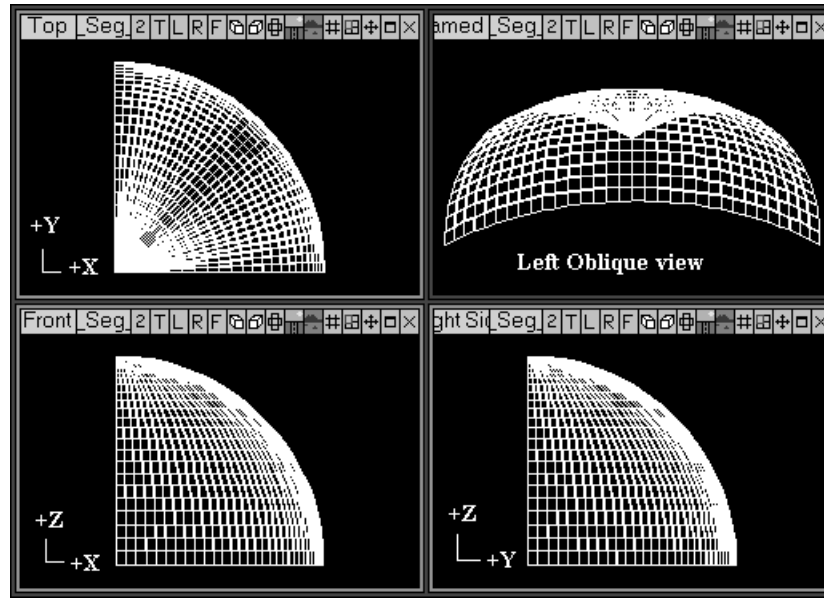
FastCAD draws the ***dome***.

Note: After **FastCAD** draws the ***3d entities***, select the **[View > Std. 3d Views]** command to ***center the view in each of the four windows***.

Text equivalent: ***DOME***

Sphere Segment





SPHERE SEG. (**Spherical Segment**) ***draws a wedge-shaped piece of a sphere or dome*** (**like an orange segment**). ***The segment can be part of a perfect sphere or ellipsoid, with optional thickness and an optional opening at the pole.***

Note: Before you draw ***3d entities*** you should select the **[File > New 3d Drawing]** command to start a new 3d drawing with the ***Std. 3d Viewa*** four window screen or select the **[View > Std. 3d Views]** command or ***icon*** to place ***four view windows*** on the drawing screen in the drawing you have already started. The ***upper-left window*** is the ***top view***, the ***lower-left window*** **is the** ***front view,*** the ***lower-right window*** is the ***right side view*** and the ***upper-right window*** is the ***overall view looking down on the drawing from an upper-left oblique angle.*** This will help you better visualize the entities and pick points in ***3d space***

To draw a **sphere segment**, select **[Draw3 > Sphere Seg.]**.

1. The prompt reads "**Sphere center:**". Pick a point in **3D** space.
2. The prompt reads "**Pole point or diameter 1.00000]:**". You can right-click to accept the default ***diameter*** or
   - Pick a point in any window. This point determines the ***radius*** and becomes the "north" pole of the segment. For predictable results, place the pole above or in front of the ***center***.
   - Type a ***diameter***. **FastCAD** automatically places the north pole directly above the center, so the polar axis is perpendicular to the **XY** plane.

3. The prompt reads "**Dome angle [180°]:**". You can:
   - Type an ***angle***. Note: Dome angle is part of a full circle-for example, a complete orange segment has an angle of **360°**. Half a segment has an angle of **180°**.
   - Pick a point on the bottom of the arc segment. This point also becomes the 0° meridian of the segment.
   - Right-click to accept the default (**prior**) ***dome angle of* 180°.**

4. The prompt reads "**Segment angle [360°]:**". This value tells **FastCAD** what part of a sphere this segment should be. Looking down on the pole, it's like the ending angle for an arc.-you can type an ***angle***, or pick a point at the desired angle from the center or right-click to accept the default ***segment angle*** of.**360°.**

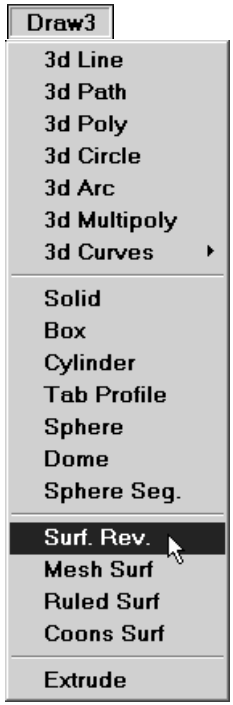
   Note: For spheres and related entities, **FastCAD** ***measures angles counterclockwise looking from the pole toward the center***. (For spheres, this method is more natural than measuring counterclockwise looking away from the center, as in other **3d** angles.)

FastCAD draws the ***sphere segment***.

Note: After **FastCAD** draws the ***3d entities***, select the **[View > Std. 3d Views]** command to ***center the view in each of the four windows***.

Text equivalent: ***SPSEG***

Surface of Revolution

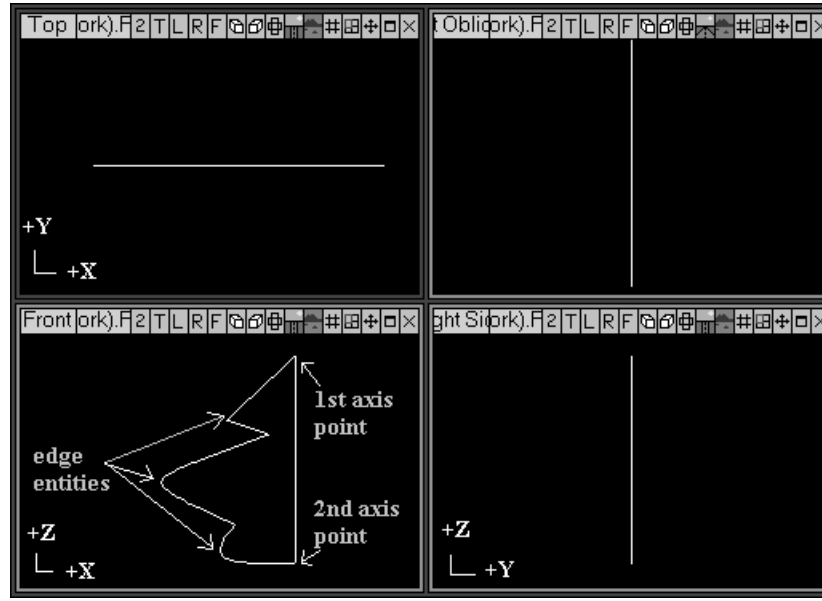


SURF. REV. (**Surface of Revolution**) ***creates three-dimensional surfaces by revolving edge entities around an axis.*** **FastCAD** ***replaces each edge you select with a surface of revolution entity. You can revolve 2d or 3d lines, circles, arcs, paths, polygons, smooth polygons, and splines.***

You mus have already drawn the ***edge entities*** before selecting this command.

Note: Before you draw ***3d entities*** you should select the **[View > Std. 3d Views]** command to place ***four view windows*** on the drawing screen. The ***upper-left window*** is the ***top view***, the ***lower-left window*** **is the** ***front view,*** the ***lower-right window*** is the ***right side view*** and the ***upper-right window*** is the ***overall view looking down on the drawing from an upper-left oblique angle.*** This will help you better visualize the entities and pick points in ***3d space***

The illustration below shows the ***edge entities*** and the ***axis line*** drawn on the surface of the ***lower-left front view*** *window.* This orients the entities ***vertically*** in the ***positive Z axis***.



Simple entities such as ***arcs*** and ***lines****i* used as ***edge entities*** allow you to make some very complex shapes using ***Surf. Rev.*** Use your creativity and experiment with combinations of entities. **3d** ***Circles*** may be revolved to create open or closed tubes, 3d ***polygons*** may be revolver to create rectangulat curver open or closed tubes for airconditioner ducts etc. If you only revolve a ***3d circle*** for **90 degrees** instead of 3**60 degrees**, you create an elbow joint for piping.

To draw a **surface of revolution**, select **[Draw3 > Surf. Rev.]**.

1. **FastCAD** asks you to ***define an axis for the revolution***.

   The prompt reads First axis point:

   Pick the ***1st axis point*** using a ***modifier*** for precision.

   The prompt reads Second axis point:

   Pick the ***second axis point***. For best results, draw the axis in the same plane as the entities you want to revolve, and pick the second axis point behind or below the first. If the axis line isn't in the same plane as the revolve entities, the resulting surface twists. Remember that **FastCAD** revolves the entities ***counterclockwise looking from the first axis point toward the second axis point.***

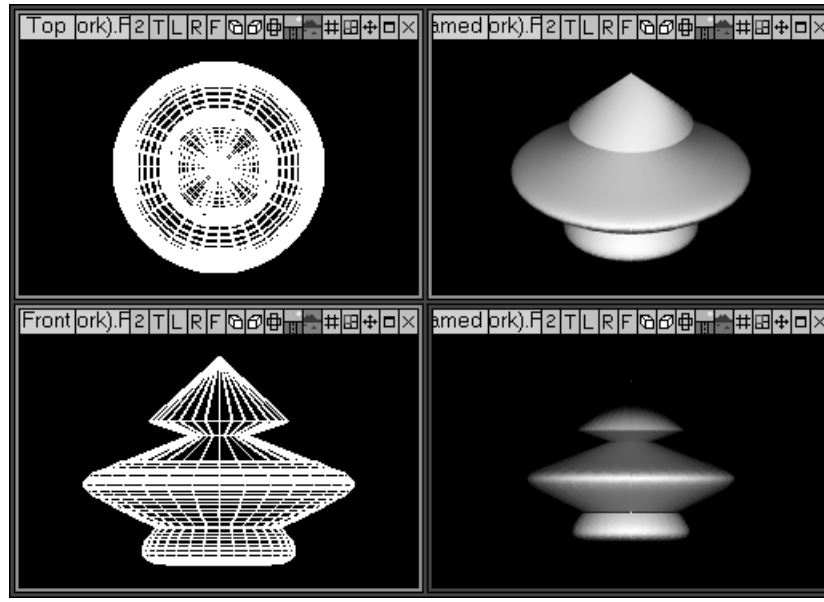
2. Now **FastCAD** asks you what entities to ***revolve***.

   The prompt reads Select entities (H=help) [0 selected]:

   Pick the ***edge entities***. **FastCAD** will ignore entities (**like text**) that do not work with this command.

3. ***The prompt reads*** Revolve for [360°]

   Type an ***angle***, right-click or press ENTER to accept the default of **360°** (**full circle**).

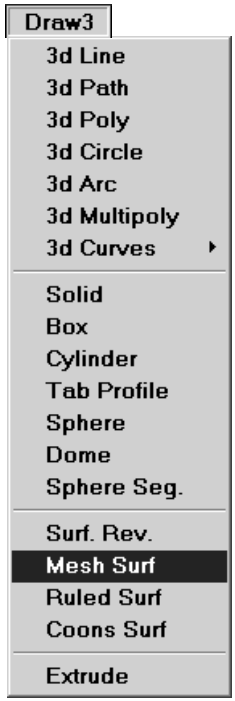


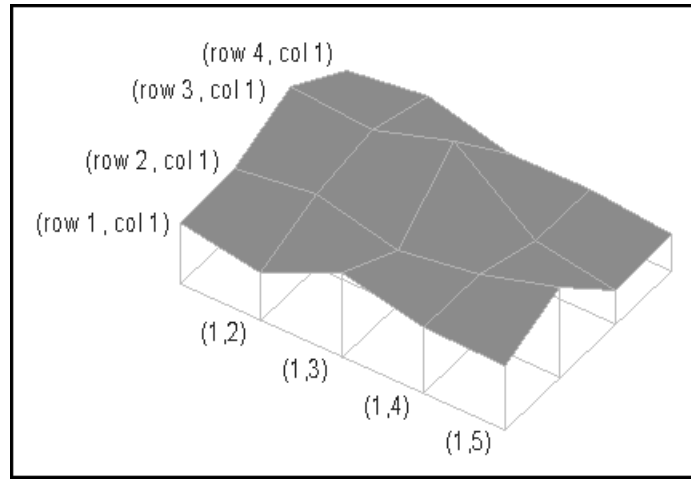
FastCAD ***revolves*** the ***edge entities*** about the ***axis*** as seen in the illustration above. Notice the ***upper-right*** and ***lower-right*** view windows have been rebdered..

Note: After **FastCAD** draws the ***3d entities***, select the **[View > Std. 3d Views]** command to ***center the view in each of the four windows***.

Text equivalent: ***SREV***

Mesh Surface



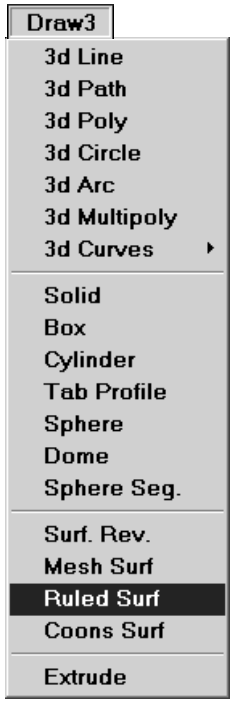


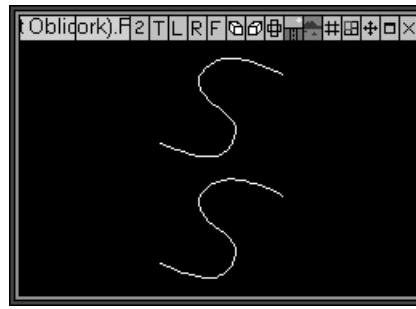
MESH SURFACE (Mesh Surface) ***creates a grid-like entity suitable for topological work and for work that needs uneven three-dimensional surfaces. You place each node in the grid independently*. (Actually, this command is most convenient for use with macros and other forms of automated input-entering a large mesh one node at a time can be very tedious.)**

1. The prompt reads "**Number of columns:**". Type the number of nodes you want in each row. For the mesh illustrated above, type 5 and press **ENTER**.
2. The prompt reads "**Number of rows:**". Type the number of nodes you want in each column (4 in the illustrated mesh).
3. The prompt reads "**Row 1 col 1 point:**". Pick a point for row 1, column 1. The next prompt reads "**Row 1 col 2 point:**". Pick a point for row 1, column 2. **FastCAD** continues to prompt you for points, connecting the points as you work with a gray mesh in progress. You will place all the points in **row 1**, then all the points in **row 2**, and so on.

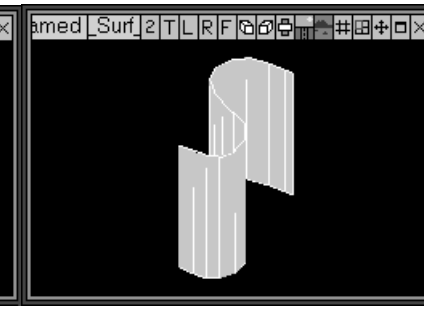
Text equivalent: ***MSURF***

Ruled Surf





Before - Two Splines



After - Ruled Surface

RULED SURF (**Ruled Surface**) ***creates a surface stretched over rules (lines) between two edges.***

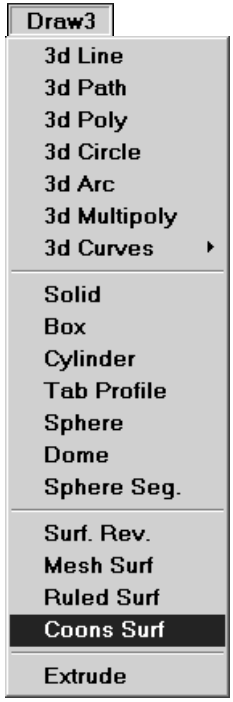
FastCAD prompts you to select two entities for ***edges***. They can be **2d** or **3d** ***arcs***, ***circles***, ***lines***, ***paths***, ***polygons***, ***smooth polygons***, or ***splines***. The edges don't have to be the same type—for example, you can connect a spline to an arc. **FastCAD** draws the surface.

NOTE: FastCAD connects the first points of the first edge to the first point of the second edge. The first point is the 0° point on an arc, the first point of a line, or node 1 for other entities. If the edges you select have their first points at opposite ends, the ruled surface twists in the middle.

To determine which end of an entity is its first point, select **3d LINE** from the **Draw3** menu, then select the ***% Along modifier***. Type 0 for the % value then pick the entity in question. **FastCAD** attaches the line's rubber-band cursor on the first point on the existing entity. You can now right-click to cancel the **3d LINE** command.

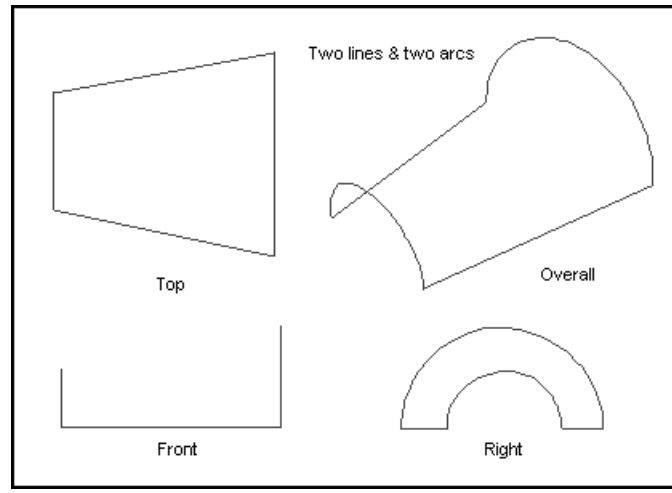
Text equivalent: ***RSURF***

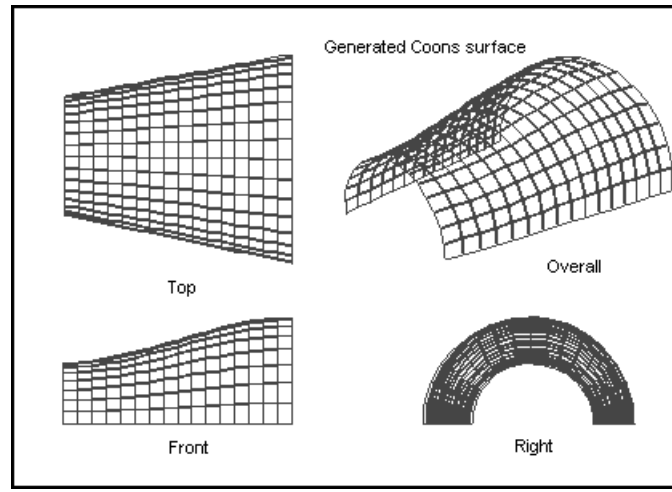
Coons Surface



COONS SURFACE (**Coons Surface, after S.A. Coons, who discovered this type of surface**) ***stretches a mesh over four edges that you select. The edges can be 2d or 3d arcs, lines, paths, or splines, but they must touch at their endpoints.***

FastCAD prompts you to select **four edges**. You must pick them in the order so that an end of **edge 1** connects to an end of **edge 2**, the other end of **edge 2** connects to **edge 3**, and so on.

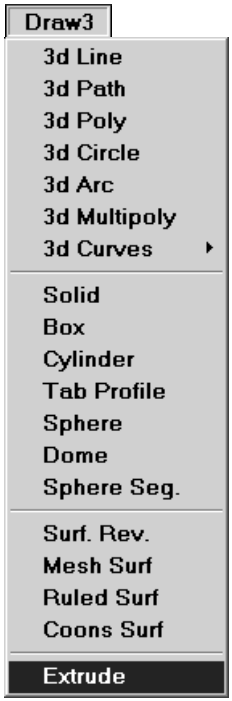




Note: For predictable results, the endpoints of the edges should be mathematically the same. The end of **edge 1** that connects to an end of **edge 2** must be exactly the same point in space (not just approximately the same point). Typing relative coordinates (**@0,0,0**) or using **REF POINT** and modifiers will help you pick the end of your edges.

Text equivalent: ***CSURF***

Extrude



Std. 3d Views

3d Icons Bar

EXTRUDE ***adds another dimension to flat*** **2d** ***or*** **3d** ***entities like lines, polygons, circles, and multipolys. You can extrude entities from any plane, in any*** **X,Y,Z** ***direction.***

FastCAD prompts for the "**Extrude form point or +Z Delta:**" and the "**Extrude to point:**" **and** calculates the **X,Y,Z distance**, or displacement, between the selected entities ***origin*** (**0,0,0**) ***points*** and ***extrudes*** them that far in each direction.

Use the Std. 3d Views command

Before ***extruding*** the entities, click the **Std. 3d Views iicon or** select the **[View > Std 3d views]** command to set up the standard ***top, front, right side and left oblique overall view*** *windows*.

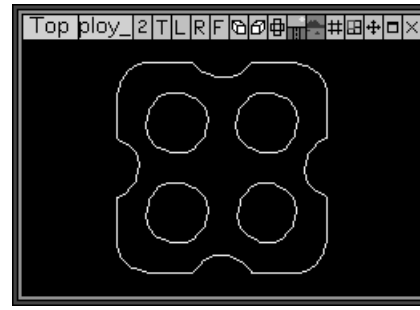
This makes it easier to pick points in ***3d space*** and see how the entities look from four different view angles. Use this command again after you ***extrude*** the entities to reposition them in each view window.

To **extrude** entities that you have drawn into **3d** space, select the **[Draw3 > Extrude]** command.

1. Select the entities you wish to ***extrude.*** The prompts reads "**Extrusion point or +Z delta**".
2. Type (**0,0,0)** meaning from the entities ***current location*** in **3d** ***space***. Now the prompt reads "**Extrude to point:**".
3. If you want the entities to ***extrude vertically two inches*** in the ***positive*** **Z** ***axis,*** type (**0,0,2"**) and press **ENTER.**

FastCAD calculates the **X,Y,Z distance** from the entities ***current location*** to the the new **X,Y,Z** ***coordinates*** and ***extrudes*** the entities that far ***in each direction***.

Example: In the example below a **2d MULTIPOLY** was defined selecting all of the ***lines, circles*** and ***arc*** entities



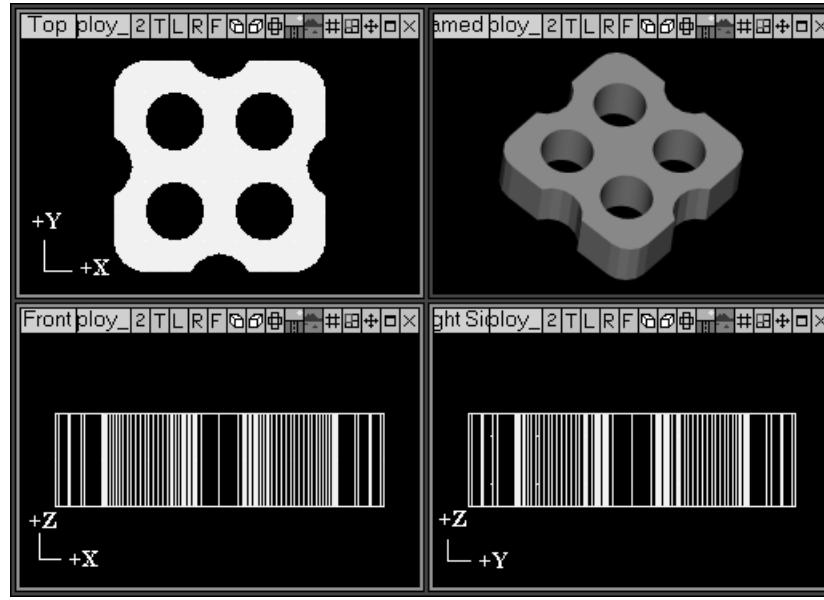
The **EXTRUDE** command was selected and at the "**Extrusion point or +Z delta**" prompt the ***absolute coordinates*** (**0,0,0**) were typed and at the "**Extrude to point:**" the ***coordinates*** (**0,0,2"**) were typed.

Typing Absolute Coordinates (X,Y,Z) for the "Extrude to point:"

Absolute coordinates were typed at the "**Extrude to point:**" prompt ***extruding*** the ***multipoly*** from the (**X=0), (Y=0)** ***axis, vertically two inches, in the positive Z axis*** **(Z=2") direction.**

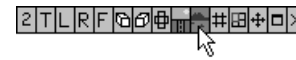
Typing Relative Coordinates (@X,Y,Z) for the "Extrude to point:"

Relative coordinate could have been used by typing (**@0,0,2"**) instead. The results of the ***extrusion*** are the same.



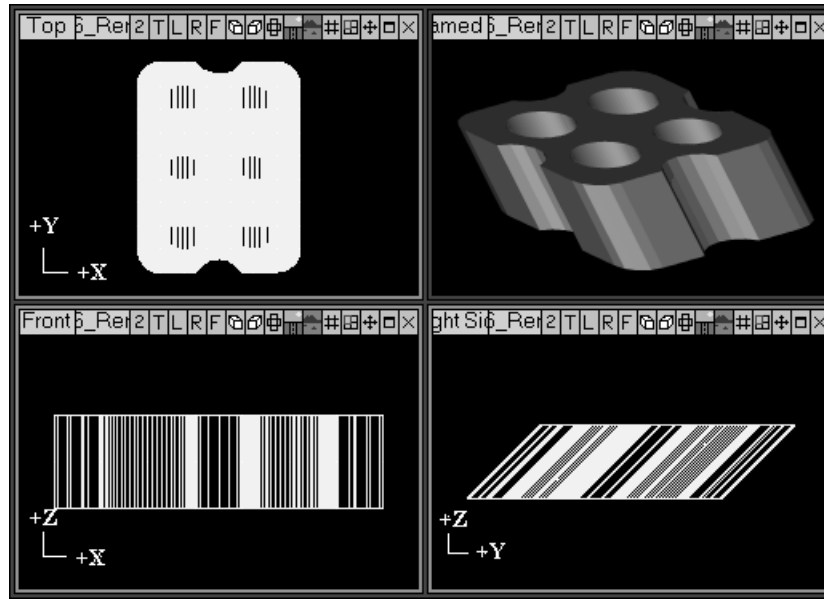
2d Multipoly Extruded 2" vertically in the Z direction

After the ***extrusion***, the **[Edit > Change > Material]** command was used to assign a **material** to the ***multipoly***. Then the ***upper-right view window's Render view on/off icon*** was toggled to ***render*** the ***multipoly***



Slanting the top of the *extruded* entities (the “Extrude to point”) from the original entities X,Y position

The top of the ***extruded multipoly*** could have been offset in the **Y axis** by ***two inches relative to the location of the original multipoly,*** by typing ***absolute coordinates*** (**0,2”,2”**) or ***relative coordinates*** **(@0,2”,2”).** The illustration below shows the results of the ***extrusion***.



The ***extrusion*** could have just as easily been ***two inches*** in all three **X,Y,Z** directions at the same time by typing (**2”,2”,2”**) in ***absolute coordinates*** or (**@2”,2”,2”**) in ***relative coordinates***.

Entity type (2D or 3D)	After EXTRUDE
Arc, Circle, Path, SPoly, Spline	**Tabulated profile with edge type based on the selected entity**
Ellipse, Elliptical Arc, text, Dimension, Arrow	**Not extruded**
Line	**3D Polygon**
Multipoly	**Parallel Multipolys (one on each end) connected by Tabulated Profiles.**
Point	**3D Line**
Polygon	**Solid**

Text equivalent: EXTRUDE

Chapter 11

Dimension Commands

Add measurement and scale to your drawings

Draw Dimensions Faster

When you draw dimensions, it's important not to "eyeball" any points.

Typically, you would use modifiers to assist in precision selection. The most common modifier used is EPT (endpoint).

A faster method is to turn on the Attach mode. The Attach button is located at the bottom of the screen on the *Button bar.* Right-click it and make sure that Nearest Endpoint is the selected mode.

When Attach mode is active,you won't need the EPT modifier. When you select a point on an entity, it will automatically snap to the entity's nearest endpoint.

Dimensions make your drawing useful by providing real life measurements that are used to construct, scale, or layout the information described in your drawings. **FastCAD** facilitate dimensioning by providing a set of specialized commands to measure distances and angles, and placing that information on the screen.

FastCAD dimensions are associative. If you dimension an entity and then stretch or scale both the dimension and entities together, the dimension will update to reflect the new measurement.

Dimension commands fall into three broad categories:

- **Linear, which measure distances between specified points.**
- **Angular, which measure angular widths.**
- **Circular, which measure radii or diameters with proper symbols.**

Use the **DIMENSION STYLE** command **[Specs > Dimension Style]** to set properties for ***dimensions*** you are about to draw. Use **CHANGED** command **[Edit > Change > Dimension Style]** to ***change properties*** of ***dimensions*** you have already drawn. Invoking either command displays the ***Dimension Style*** *dialog* box.

Dimension entities also possess ***Line Width*** and ***Line Style*** properties and draw with these properties. The **CHANGE PEN THICKNESS**, **CHANGE-LINE WIDTH** and **CHANGE-LINE STYLE** commands work with dimensions.

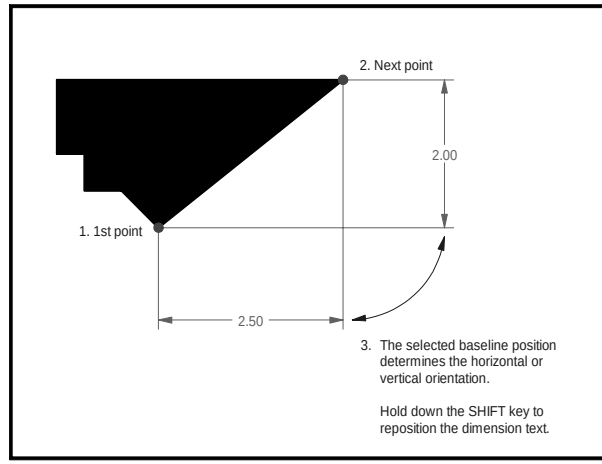
To set the ***text color different from the rest of the dimension***, use the **CHANGEC2** command **[Edit > Change > Fill (2nd) Color],** and apply it to the selected ***dimension(s)***. ***Only the text will redraw with the secondary color.*** The leaders, extension lines, and arrowheads will remain the same as the primary color.

The **DYNEDIT** command **[Edit > Dynamic Edit]** provides the following capabilities on drawn ***dimensions:***

Linear, Horizontal, Vertical, Parallel,* or *Baseline Use **DUNAMIC EDIT** to ***visually reposition the dimension line***. If you hold down the **SHIFT** key, you can ***visually reposition the dimension text***. The ***dimension remains associative***.

Ordinate Use **DYNAMIC EDIT** to *insert* or *delete* leaders.

Dimension > Linear



LINEAR ***dimensions*** measure and display either the ***horizontal or vertical*** distance between two points (or of an existing entity). If you ***pick the dimension points horizontally*** **FastCAD** determines that you want a ***horizontal dimension***. If you ***pick the two dimension points vertically***, **FastCAD** determines that you want a ***vertical dimension.***

Drawing Horizontal and Vertical Dimensions around entities at the same time:

You can actually ***dimension*** around each side of an entity or entities such as **polygons** and **segments of paths** and depending on where the pick points are,.**FastCAD** will draw the proper ***horizontal*** or ***vertical dimension***. This means that if you had a rectangle and you wanted to ***dimension*** the ***top horizontal*** **side** and the ***right vertical side***, pick the first snd second ***dimension*** points at the left and right endpoints of the top horizontal side and then pick the lower endpoint of the right side for the third ***dimension*** point **FastCAD** will draw a ***horizontal dimension on the top*** and a ***vertical dimension on the right side***.This is a real time saver.

FastCAD allows ***dynamic placement of the dimension and tex***t. Final placement and orientation is dependent upon your dynamic selection.

1. The prompt reads "**1st geometry point [ENTER to select edge]:**". To select a point, use the crosshairs or type numeric coordinates. **Modifiers** or the **Attach mode** should be used to ensure precise point selection.

   If you right-click the mouse or press **ENTER** the prompt reads

   "**Edge entity to dimension:**".

   If you left-click on a **Line** or the **Edge Segment** of a **Path** or a **Polygon FastCAD** will **automatically find its exact ENDPOINTS** and **attach the dimension leaders to them** and prompt you for the "**Text baseline point (Shift =Spec Text Center):**". Left-click to place the **Dimension**.

2. The prompt reads "**Next geometry point:**". **FastCAD** displays a dynamic cursor of the **dimension**. You can continue picking **Dimension points inline** using **different Modifiers** or **Attach Mode** or right-click the mouse or press **ENTER** again and the prompt reads

   "**Edge entity to dimension:**".

   Again left-click on a **Line** or the **Edge Segment** of a **Path** or a **Polygon** and **FastCAD** will **automatically find its exact ENDPOINTS** and **attach the dimension leaders to them** and prompt you for the "**Text baseline point (Shift =Spec Text Center):**".

3. The prompt reads "**Text baseline point (Shift =Spec Text Center):**". Depending on where you move the dynamic cursor, **FastCAD** will draw a ***horizontal*** or ***vertical dimension***. Text placement snaps to default positions, again depending upon the position of the dynamic cursor. If you hold down the SHIFT key and move the cursor, the text "**separates**" from the extension and dimension lines, allowing custom placement of the text.

   Keyboard entry is not applicable for this input. Left-click when you are satisfied with the **placement** and **orientation** of the **dimension**. **FastCAD** draws the ***dimension.***

4. The prompt reads "**Next geometry point:**". You can right-click to end dimensioning. Otherwise, **FastCAD** allows you to continue the dimension chain in the orientation (horizontal or vertical) in which you placed the first dimension entity. From the previous point, select a point at the other end of the distance to be measured.

5. The prompt reads "**Text baseline point (Shift = Spec Text Center):**". Again, you have dynamic control over the placement of the dimension line. The new dimension builds off the last point of the previous dimension. Right-click to accept the default which draws dimension text and arrows lined up with (in line with) the last dimension you drew. Keyboard entry is not applicable for this input.

FastCAD keeps **extension lines at right angles** to **Dimension Arrows** during all drawing and editing operations.

When **FastCAD** places a ***linear dimension text***, it adheres to the ***text alignment*** parameters as specified in the ***Dimension Style*** *dialog* box.

Setting properties for new linear dimensions:

Use the **DIMENSION STYLE** command **[Specs > Dimension Style]**.

Position editing of baselines or text of an existing Linear Dimension:

Use the **DYNAMIC EDIT** command **[Edit > Dynamic Edit]** to dynamically reposition the dimension line. Hold down the **SHIFT key** to reposition the text. The **Dimension remains associative**.

Editing properties for an existing Linear Dimension:

Use the **[Edit > Edit]** command **[Edit > Edit]** to change the arrowhead, leader options, tolerances, and text properties and position for an existing ***linear***

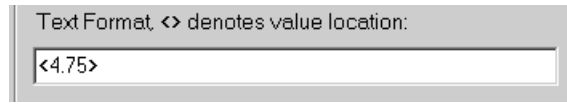
dimension. Only options you "**touch**" will take effect, unless you apply a named ***dimension style***.

To enter a fixed dimension text value:

Use the **EDIT** command, select the ***dimension(s)***, and enter a new value between the ***less than*** and ***greater than*** brackets in the window labeled ***'Text format <> denotes value location'*** in the ***Dimension Style*** dialog box..The value you enter must be numerical (alphabetic characters and other non-numerical characters are not valid). ***The value you enter will not change if the dimension is stretched or scaled***.

If you wish to ***revert the fixed value back to its default associative mode***, use the **[Edit > Edit]** command again, select the same ***dimension*** and remove the value from between these brackets..You may also add ***prefix*** and ***sufix text*** to ***dimensions*** by placing the ***prefix text on left side*** and ***the suffix text on the right side*** of the brackets (**prefix<fixed value>suffix**).

EXAMPLE:



Fixed value only - <4.75>

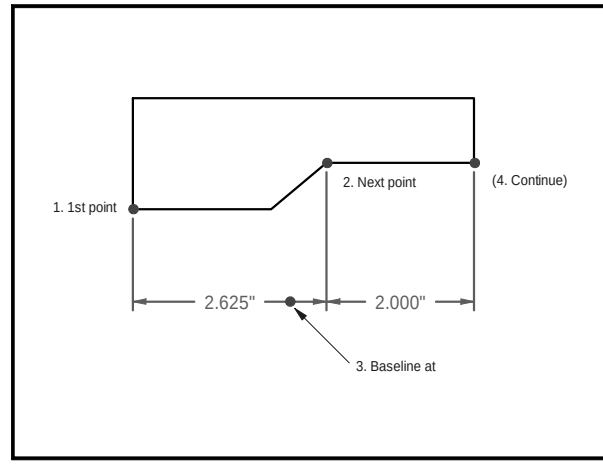
Prefix & fixed value - aprox<4.75>

Prefix, suffix & fixed value - aprox<4.75>less .5

Text equivalent: ***DIML***

Horizontal Dimension Icon

Dimension > Horizontal



HORIZONTAL ***dimensions*** measure and display only the ***horizontal*** distance (**along the X axis**) between two points (or of an existing entity), ignoring any vertical difference between the points. Dimension text and arrows are ***horizontal***.

The **HORIZONTAL DIMENSION** entity data structure is identical to that created by the **LINEAR DIMENSION** command.

1. The prompt reads "**1st geometry point [ENTER to select edge]:**". To select a point, use the crosshairs or type **numeric coordinates**. **Modifiers** or the **Attach mode** should be used to ensure precise point selection.

   If you right-click the mouse or press **ENTER** the prompt reads

   "**Edge entity to dimension:**".

   If you left-click on a **Line** or the **Edge Segment** of a **Path** or a **Polygon FastCAD** will **automatically find its exact ENDPOINTS** and **attach the dimension leaders to them** and prompt you for the "**Text baseline point (Shift =Spec Text Center):**". Left-click to place the **Dimension**.

2. The prompt reads "**Next geometry point:**". **FastCAD** displays a dynamic cursor of the ***dimension***. You can continue picking **Dimension points inline** using **different Modifiers** or **Attach Mode** or right-click the mouse or press **ENTER** again and the prompt reads

   "**Edge entity to dimension**:".

   Again left-click on a **Line** or the **Edge Segment** of a **Path** or a **Polygon** and **FastCAD** will **automatically find its exact ENDPOINTS** and **attach the dimension leaders to them** and prompt you for the "**Text baseline point (Shift =Spec Text Center):**".

3. The prompt reads "**Text baseline point (Shift = Spec Text Center):**". Text placement snaps to default positions, depending upon the position of the dynamic cursor. If you hold down the **SHIFT** key and move the cursor, the text "**separates**" from the extension and ***dimension lines***, allowing custom placement of the text.

   Keyboard entry is not applicable for this input. Left-click when you are satisfied with its placement and orientation. **FastCAD** draws the ***horizontal dimension***.

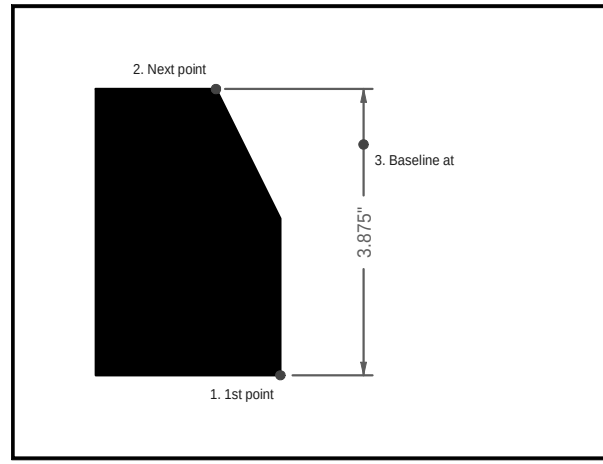
4. The prompt reads "**Next geometry point:**". Right-click to end ***dimensioning***. Otherwise, **FastCAD** allows you to continue the ***dimension chain*** in the direction in which you selected the first two points. From the previous point, select a point at the other end of the distance to be measured. ***The next dimension will automatically be drawn inline with the dimension text and arrows of the last dimension you drew***. Keyboard entry is not applicable for this input.

FastCAD keeps extension lines at right angles to ***dimension arrows*** during all drawing and editing operations.

Text equivalent: ***DIMH***

Vertical Dimension Icon

Dimension > Vertical



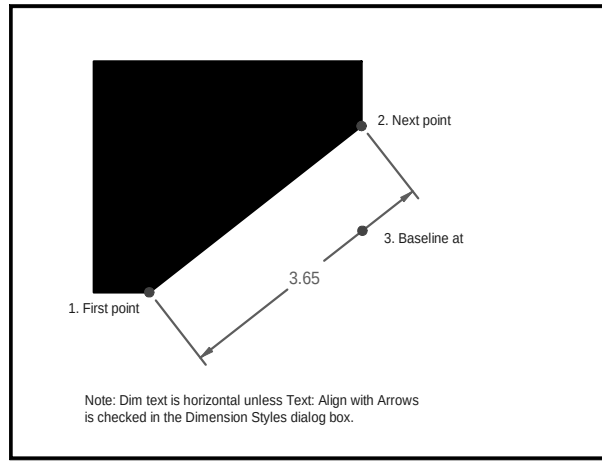
VERTICAL ***dimensions*** measure and display only the ***vertical distance*** (**along the Y axis**) between two points (or of an existing entity), ignoring any **horizontal** difference between the points. ***Dimension arrows are horizontal***. You can control the orientation of ***vertical dimension text*** with **DIMENSION STYLE command [Specs > Dimension Style]**. Check the ***Text: Align text with arrows*** *check box* if you want the ***dimension text to read vertically instead of horizontally***.

The **VERTICAL DIMENSION** entity data structure is identical to that created by the **LINEAR DIMENSION** command.

To draw **vertical dimensions**, follow the methodology outlined for the **HORIZONTAL DIMENSION** command, except in the vertical orientation.

Text equivalent: ***DIMV***

Dimension > Parallel



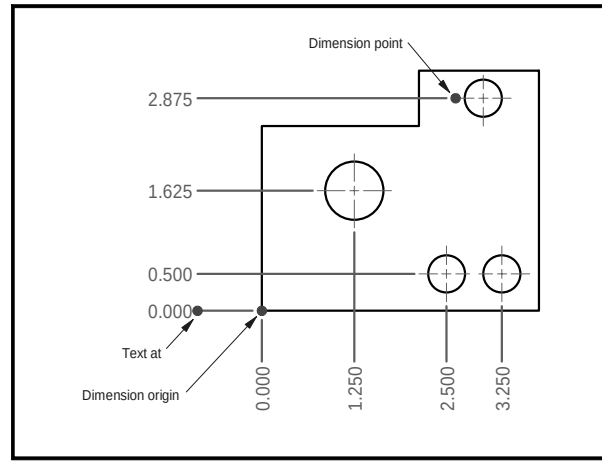
PARALLEL ***dimensions*** measure and display the actual distance, ***at any angle***, between two points (or of an existing entity). ***Parallel dimensions*** have identical length leader lines.

To draw **PARALLEL DIMENSION**, follow the methodology outlined for the **HORIZONTAL DIMENSION** command, except in an **orientation parallel to the reference entity**.

Text equivalent: ***DIMP***

Ordinate Dimension Icon

Dimension > Ordinate



ORDINATE (also called rectangular coordinates) ***dimensions*** show the relative ***horizontal*** or ***vertical*** distance from a ***local origin point*** that you select. An ***ordinate dimension*** entity consists of an ***origin leader*** and a series of ***additional leaders***, each of which displays its ***horizontal distances*** across the bottom and ***vertical distances*** up the side.

1. The prompt reads "**Dimension origin:**". This is the point from which you wish to measure. To select the ***origin***, use the cursor or type numeric coordinates.

   FastCAD displays a rubber-band cursor anchored at the ***origin point***.

2. The prompt reads "**Text baseline at:**". Select a location for the ***origin point's text***. This point also determines whether you are drawing a ***horizontal*** or ***vertical dimension***. Pick a point to the left or right of the origin to measure vertical distances; pick a point above or below the origin to measure horizontal distances. **Modifiers** (and the active **attach mode**, if toggled on) can be used.

   FastCAD draws the origin text and leader using the active color and line style. Note that the rubber-band cursor is locked parallel to the ***origin leader***.

3. The prompt reads "**Dimension point:**". Select a point on the object you want to measure.

   FastCAD draws the ordinate dimension value and leader.

4. The prompts reads "**Dimension point:**". Continue selecting dimension points as needed. To end the command, right-click or press ENTER.

   Typically, at this point, you would construct ***ordinate dimensions*** in the ***perpendicular direction***. For example, if your first set of ***ordinate dimensions*** were ***horizontal,*** you would now create ***ordinate dimensions*** for those same objects in the ***vertical direction***.

Setting properties for new dimensions (of any kind) Use the **DIMENSION STYLE** command **[Specs > Dimension Style]**.

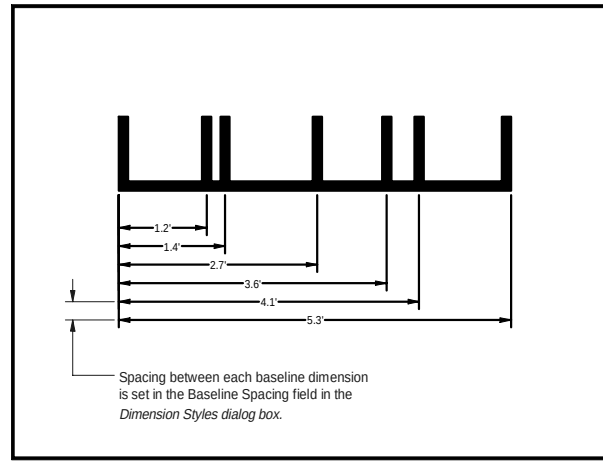
Adding or removing leaders to an existing ordinate dimension Use the **DYNAMIC EDIT** command **[Edit > Dynamic Edit]** to add a new leader. After you select any ***ordinate leader*** you are automatically in ***insert mode*** to add new leaders. Hit the **DELETE** key and click a leader to ***remove itr***. After remocing a leader you are automatically back to ***insert mode*** so press the **DELETE** key again to remove another leader. Right-click to terminate the **DYNAMIC EDIT** command.

Editing properties for an existing ordinate dimension Use the **EDIT** command **[Edit > Edit]** to change the arrowhead, leader options, tolerances, text properties and position for an existing ordinate dimension. Only options you "**touch**" will take effect, unless you apply a named dimension style.

You may add ***prefix*** and ***sufix*** text to on ***Ordinate dimensions*** using the ***Dimension Style*** *dialog* but if you add a ***fixed value*** it will appear in ***all leaders*** of the dimension.

Text equivalent: ***DIMO***

Dimension > Baseline



BASELINE ***dimensions*** show the relative ***horizontal*** or ***vertical*** distance from a local origin point that you select. Each dimension you add is automatically "**stacked**" adjacent to the previous baseline dimension.

Each ***baseline dimension*** entity is identical to that created by the **LINEAR DIMENSION** command, and their construction method is also similar. However, ***baseline dimensions*** and ***ordinate dimensions*** share more similar usage traits. Your own style requirements will dictate usage of one over the other. Typically, ***ordinate dimensions*** are more common in ***mechanical drawings***, while ***baseline dimensions*** are more prevalent in ***architectural drawings***.

1. The prompt reads "**1st geometry point:**". To select a point, use the crosshairs or type numeric coordinates. **Modifiers** or the **Attach mode** should be used to ensure precise point selection. To select an entity to dimension, right-click or press **ENTER**, then use the pick cursor to select an entity.
2. The prompt reads "**Next geometry point:**" FastCAD displays a dynamic cursor of the dimension.
3. The prompt reads "**Text baseline point (Shift = Spec Text Center):**". Depending on where you move the dynamic cursor, FastCAD will draw a horizontal or vertical dimension. Text placement snaps to default positions, again depending upon the position of the dynamic cursor. If you hold down the **SHIFT** key and move the cursor, the text "**separates**" from the extension and dimension lines, allowing custom placement of the text.

   Keyboard entry is not applicable for this input. Left-click when you are satisfied with the placement and orientation of the ***dimension***. **FastCAD** draws the ***dimension***.
4. The prompt reads "**Next geometry point:**". You can right-click to end dimensioning. Otherwise, **FastCAD** allows you to continue the ***dimension chain*** in the orientation (***horizontal*** or ***vertical***) in which you placed the first ***dimension*** entity.

From the previous point, select a point at the other end of the distance to be measured.

FastCAD draws the ***dimension***, spaced as specified in the **Baseline Spacing** field in the ***Dimension Style*** *dialog*. Continue adding more points, or right-click to end.

FastCAD keeps extension lines at right angles to ***dimension arrows*** during all drawing and editing operations.

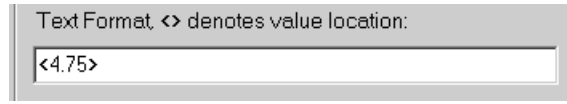
Setting properties for new dimensions (of any kind) Use the **DIMENSION STYLES** command **[Specs > Dimension Style]**.

Position editing of baselines or text of an existing baseline dimension Use the **DYNAMIC EDIT** command **[Edit > Dynamic Edit]** to dynamically reposition the ***dimension line.*** Hold down the **SHIFT** key to reposition the text. The **dimension remains associative.**

Editing properties for an existing Baseline Dimension Use the **EDIT** command **[Edit > Edit]** to change the arrowhead, leader options, tolerances, and text properties and position for an existing ***baseline dimension***. Only options you "**touch**" will take effect, unless you apply a named ***dimension style***.

To enter a Fixed Dimension Text Value Use the **[Edit > Edit]** command, select the ***dimension(s)***, and enter a new value between the ***less than*** and ***greater than*** brackets in the window labeled ***'Text format <> denotes value location'*** in the ***Dimension Style*** dialog box..The value you enter must be numerical (alphabetic characters and other non-numerical characters are not valid). ***The value you enter will not change if the dimension is stretched or scaled***. If you wish to ***revert the fixed value back to its default associative mode***, use the **EDIT** command again, select the same ***dimension*** and ***remove the value from between these brackets***..You may also add ***prefix*** and ***sufix text*** to ***dimensions*** by placing the ***prefix text on left side*** and the ***suffix text on the right side of the brackets*** **(prefix<fixed value>suffix**).

EXAMPLE:



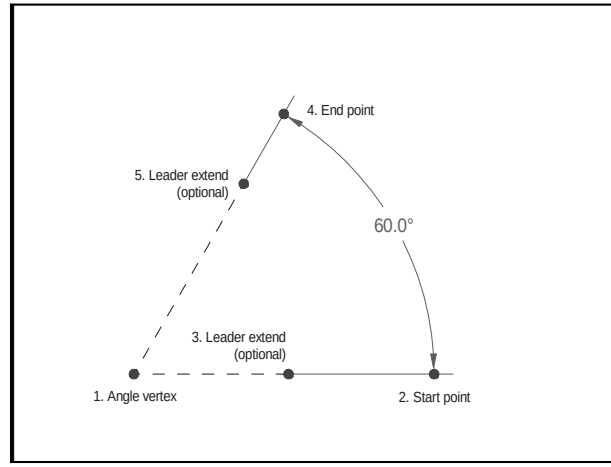
Fixed value only - <4.75>

Prefix & fixed value - aprox<4.75>

Prefix, suffix & fixed value - aprox<4.75>less .5

ext equivalent: ***DIMB***

Dimension > Full Angular



FULL ANGULAR ***dimensions*** measure an ***angle width*** (**in degrees**) and displays that value. You specify the ***center***, ***starting point***, and ***ending point*** of the ***angle***. Optional leaders are drawn in the current line style. ***Angular dimensions*** are specified in the ***positive angular direction***.

To draw ***angular dimensions***, select **[Dimensions > Full Angular]**:

1. The prompt reads "**Angle vertex:**". Pick a point (for example, the intersection of two lines) or type numeric coordinates. The ***angle vertex*** also determines the ***center*** of the ***dimension arc***.

   FastCAD draws a rubber-band cursor from the center.

2. The prompt reads "**Start point on arc:**". The ***starting point*** specifies:

   - ***Starting angle*** of the ***dimension arc***
   - ***Radius*** of the ***dimension arc***

   Use the dynamic cursor to pick a point.

   FastCAD displays a rubber-band cursor, anchored at the ***starting point*** and radially aligned to the ***angle vertex***.

3. The prompt reads "**Start leader extend [none]:**". If no leader is desired, right-click or press **ENTER**. To create a leader, stretch the cursor and click when the leader is the desired size.

   FastCAD draws the leader and displays dynamic cursor of the dimension form.

4. The prompt reads "**End arc bearing point:**". Stretch the cursor and pick a point when the angular width of the dimension is correct.

   FastCAD displays a rubber-band cursor for the second leader.

5. The prompt reads "**End leader extend [none]:**". When **FastCAD** asks for the length and orientation of the second leader, specify as you did for the first.

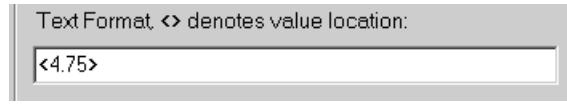
   FastCAD draws the dimension counterclockwise from the starting point to the ending point, and the command terminates

To set properties for new dimensions, use **DIMENSION STYLE**.

To edit properties for an existing ***angular dimension***, use **[Edit > Edit]**.

To enter a Fixed Dimension Text Value Use the **[Edit > Edit]** command, select the dimension(s), and enter a new value between the ***less than*** and ***greater than brackets*** in the window labeled ***'Text format <> denotes value location'*** in the ***Dimension Style*** *dialog box*..The value you enter must be numerical (alphabetic characters and other non-numerical characters are not valid). The value you enter will not change if the dimension is stretched or scaled. If you wish to r***evert the fixed value back to its default associative mode***, use the **[Edit > Edit]** command again, select the same dimension and ***remove the value from between these bracket***s..You may also add ***prefix*** and ***sufix text*** to dimensions by placing the ***prefix text on the left side*** and the ***suffix text on the right side of the brackets*** (**prefix<fixed value>suffix**).

EXAMPLE:



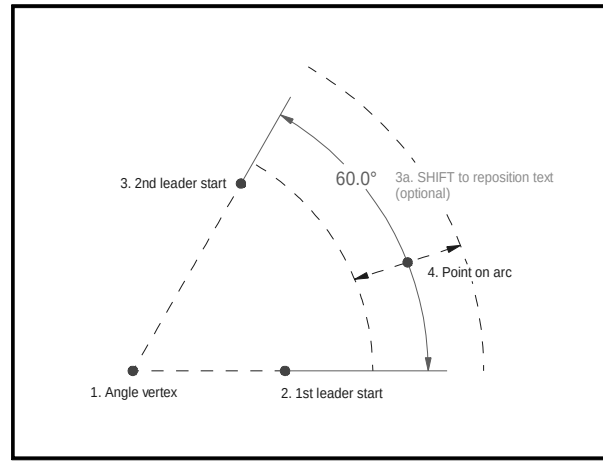
Fixed value only - <4.75>

Prefix & fixed value - aprox<4.75>

Prefix, suffix & fixed value - aprox<4.75>less .5

Text equivalent: ***DIMA***

Dimension > Quick Angular



QUICK ANGULAR ***dimensions*** measure an ***angle*** (**in degrees**) and displays that value. You specify the ***center***, ***starting point***, and ***ending point*** of the ***angle***. Optional leaders are drawn in the current line style.

This command uses dynamic cursors to facilitate ***angular dimension*** construction. Since you visually specify the final dimension, the two leader points can be picked in any order (by contrast, the **DIMA** dimension always draws the dimension in a counterclockwise direction). Note that leaders cannot be suppressed.

To draw ***quick angular dimensions***, select **[Dimensions > Quick Angular]**:

1. The prompt reads "**Angle vertex:**". Pick a point (for example, the intersection of two lines) or type numeric coordinates. The ***angle vertex*** also determines the ***center*** of the ***dimension arc***.

   FastCAD draws a rubber-band cursor from the ***center***.

2. The prompt reads "**1st leader start point:**". The 1st point specifies:

   - ***Starting angle*** of the ***dimension arc***
   - ***Starting point*** of the ***first leade***r

   Use the dynamic cursor to pick a point.

   FastCAD displays a rubber-band cursor, anchored at the ***starting point*** and radially aligned to the ***angle vertex***.

3. The prompt reads "**2nd leader start point:**". The 2nd point specifies:

   - ***Starting angle*** of the ***dimension arc***
   - ***Starting point*** of the ***first leader***

Use the dynamic cursor to pick a point.

FastCAD displays dynamic cursor of the dimension form. The dynamic cursor is anchored at the ***vertex***, and the ***radius*** and leaders shrink or grow according to mouse movement.

4. The prompt reads "**Point on arc (SHIFT to set fixed text position):**". Stretch the cursor and pick a point to place the ***dimension text***. If you move the mouse through the "**Angle vertex**" point (step 1), the dimension switches to measure the outer angle width.

   During this step, the text can be repositioned by holding down the SHIFT key and moving the mouse. When you satisfied with the location of the text, release the SHIFT key.

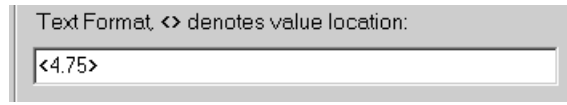
   After you pick a point, **FastCAD** ***draws the dimension counterclockwise from the starting point to the ending point***, and the command terminates.

To set properties for new dimensions, use **DIMENSION STYLE**.

To edit properties for an **existing angular dimension**, use **[Edit > Edit]**.

To enter a Fixed Dimension Text Value Use the **[Edit > Edit]** command, select the ***dimension(s)***, and enter a new value between the ***less than*** and ***greater than brackets*** in the window labeled ***'Text format <> denotes value location'*** in the ***Dimension Style*** *dialog box.* The value you enter must be numerical (alphabetic characters and other non-numerical characters are not valid). The value you enter ***will not change if the dimension is stretched or scaled***. If you wish to ***revert the fixed value back to its default associative mode***, use the **[Edit > Edit]** command again, select the same dimension and ***remove the value from between these brackets***..You may also add ***prefix*** and ***sufix text*** to ***dimensions*** by placing the ***prefix*** **text on** ***the left side*** and the ***suffix text on the right side of the brackets*** (**prefix<fixed value>suffix**).

EXAMPLE:



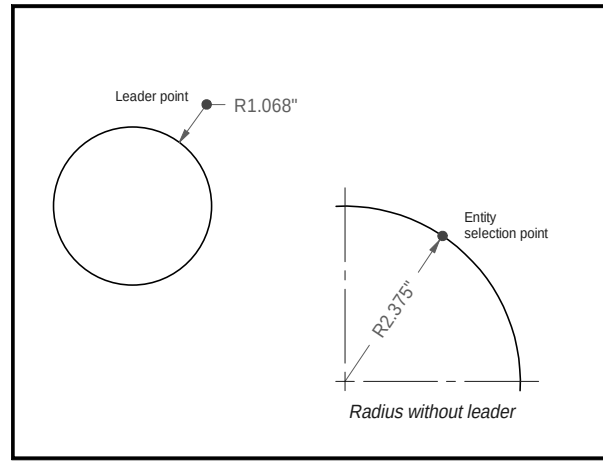
Fixed value only - <4.75>

Prefix & fixed value - aprox<4.75>

Prefix, suffix & fixed value - aprox<4.75>less .5

Text equivalent: ***DIMAQ***

Dimension > Radius



RADIUS ***dimensions*** measure the ***radius*** of a ***circle*** or ***arc*** and display that value.

To draw a ***radius dimension***, select **[Dimension > Radius]**:

1. The prompt reads "**Circular entity:**". Pick a circle or arc at the arrow leader attachment point.

   FastCAD displays a rubber-band cursor anchored to the pick point, and radially aligned with the entity's ***center point***.

2. The prompt reads "**Bend point [inside]:**". If you right-click, **FastCAD** will automatically draw the dimension from the ***center point*** to the pick point, and the command terminates.

   Otherwise, you can pick the bend point for the arrow leader. The leader can be either inside or outside the entity. Use the rubber-band cursor to visually determine the leader length.

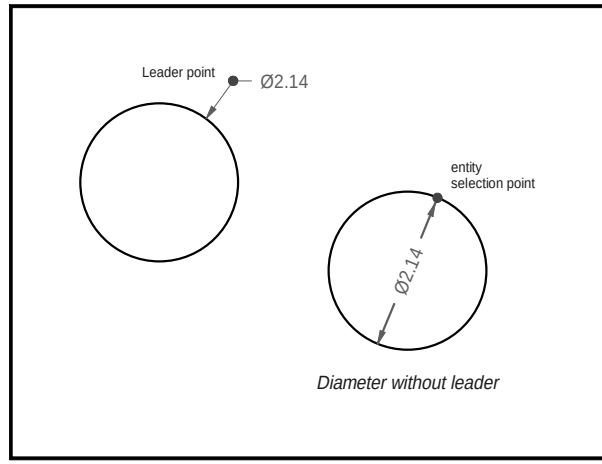
3. The prompt reads "**Leader end point [standard]:**". If you right-click, **FastCAD** automatically draws the horizontal portion of the leader, connecting the text to the leader bend point. **FastCAD** uses the current dimension arrow length as the "**standard**" leader length.

   Otherwise, you can manually specify the leader length by picking a point on the screen.

   FastCAD draws the radius dimension adjacent to the leader and terminates the command. Radius dimensions begin with the capital letter **R**.

Text equivalent: ***DIMR***

Dimension > Diameter



DIAMETER ***dimensions*** measure the ***diameter*** of a ***circle*** or ***arc*** and display that value.

To draw a ***diameter dimension***, select **[Dimension > Diameter]**:

1. The prompt reads "**Circular entity:**". Pick a ***circle*** or ***arc*** at the arrow leader attachment point.

   FastCAD displays a rubber-band cursor anchored to the pick point, and radially aligned with the entity ***center point.***

2. The prompt reads "**Bend point [inside]:**". If you right-click, **FastCAD** will automatically draw an interior dimension from the pick point through the ***center***, and the command terminates.

   Otherwise, you can pick the bend point for the arrow leader. The leader can be either inside or outside the entity. Use the rubber-band cursor to visually determine the leader length.

3. The prompt reads "**Leader end point [standard]:**". If you right-click, **FastCAD** automatically draws the horizontal portion of the leader, connecting the text to the leader bend point. **FastCAD** uses the current dimension arrow length as the "**standard**" leader length.

   Otherwise, you can manually specify the leader length by picking a point on the screen.

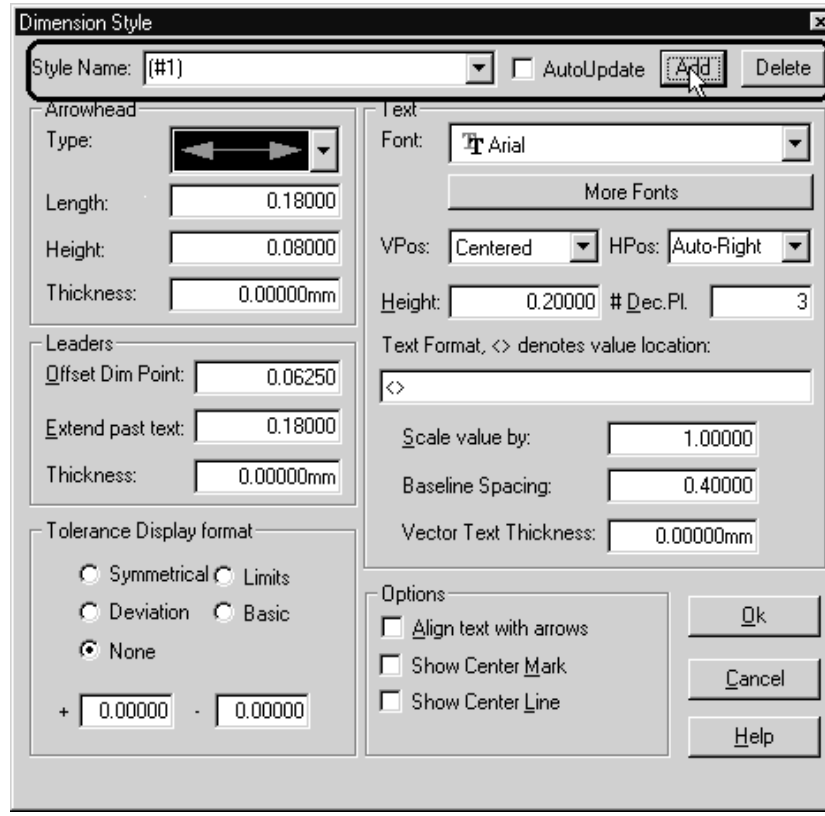
FastCAD draws the diameter dimension adjacent to the leader and terminates the command. ***Diameter dimension*s** begin with the letter **phi** as the first character in the ***dimension***.

Text equivalent: ***DIMD***

Dimension Style

Dimension Style Dialog Box

This dialog box is displayed when you invoke the **DIMENSION STYLES** command bu clicking the ***Dimension Style*** *icon* as seen at the left or selecting **[Specs > Dimension Styles]** from the menu, the **CHANGE-DIMENSION STYLE** command **[Edit > Change > Dimension Style],** or when selecting a dimension entity using the **EDIT** command **[Edit > Edit].** All of the perameters that control how a **Dimension** will appear on the screen are set from this dialog box.



Style Name

Style Name Window

This window displays the names of the Dimension Style sets you set up.Setting up a series of **DImension Style Sets** can be a real time saver. You set the desired **Dimension Text Height**, **Font Style, Arrowhead style** and **size**, **Dimension Text Position. Thickness** for the **Leaders,** type of **Dimension Tolerance** etc**.** You then click the **Add** button and type a **Style Name** in the window and click **OK.** You may set up several **Dimension Style Sets** and select them from the **DStyle Status** window drop-down menu. All of **Dimension** properties of the **Dimension Style Set** are changed at once. You may use the **[Edit > Change > Dimension Style]** and select one or more **Dimension** entities and select a different **DImension Style Name** and they all change at once to the properties of the new **Dimension Style Set.**.

AutoUpdate Box

When checked, **FastCAD** automatically applies the current format to all **dimensions** with the same **named dimension style**. If you want dimensions of the same named style to have different formats, leave this option unchecked.

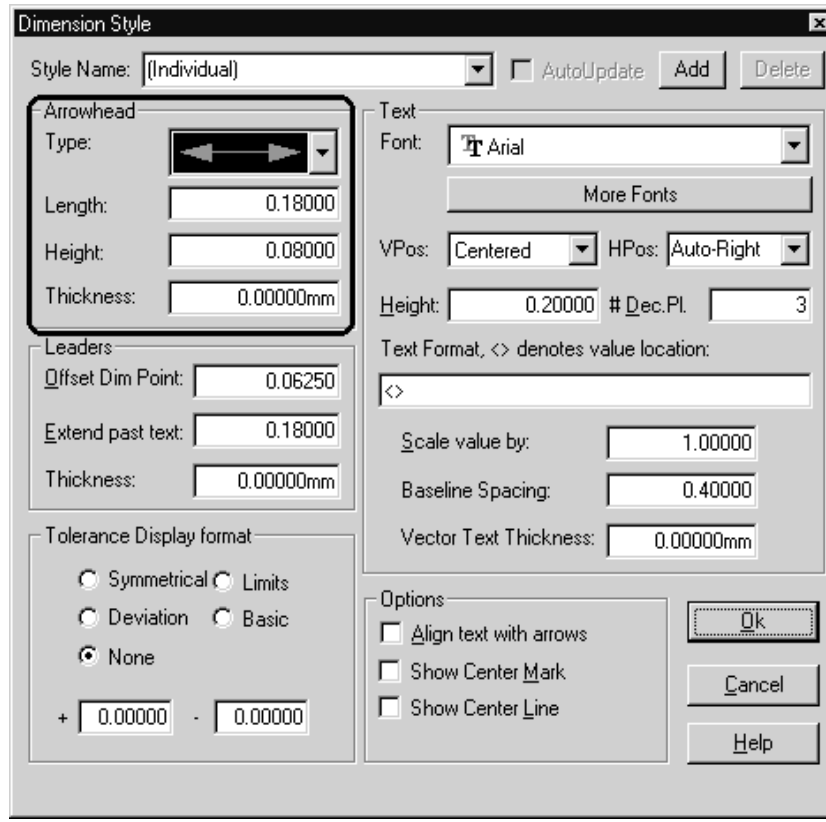
Add Button

Creates a new **Dimension Style Name**. Initially, the new style will acquire the current listed properties. As styles are added, they are auto-named **#n**, where n is the next ordinal number. To rename any style, click into the Name field and type in a new name.

Delete Button

Deletes the selected Dimension Style.

Dimension Arrowhead



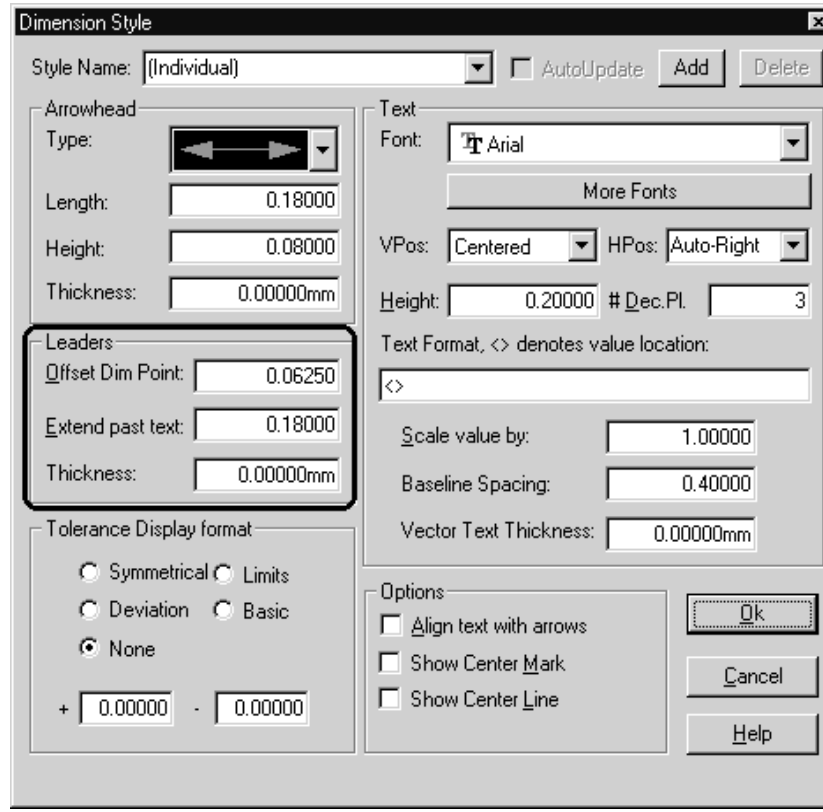
Type Select the arrowhead style from a drop-down list of available styles.

Length Length of arrowhead from endpoint to base (drawing units).

Height Base width of arrowhead (drawing units).

Thickness Pen Thickness of arrowheads.

Dimension Leaders

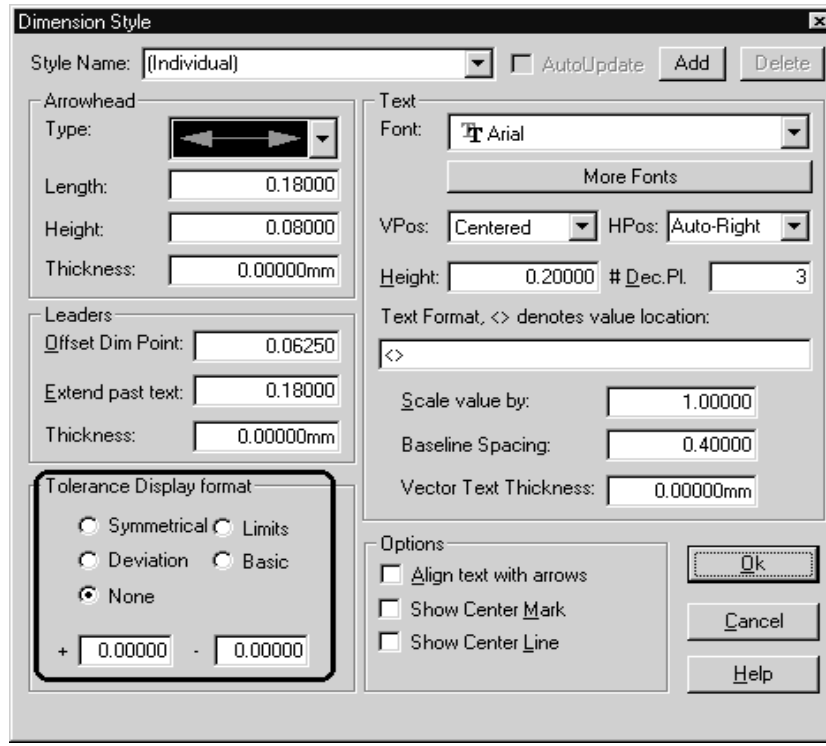


Offset Dim Point Specifies the distance from the edge of a dimensioned entity to the start of the extension line.

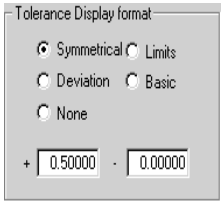
Extend past text Specifies the distance of the extension line beyond the dimension line.

Thickness Pen thickness of leader lines in millimeters.

Dimension Tolerance Display Format

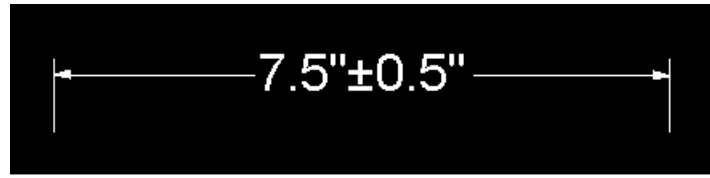


Symmetrical Format



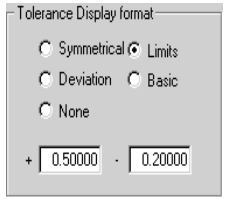
Symmetrical Dimension Tolerance

This format adds a **plus/minus** value of tolerance displaying a ± after the **dimension text.** Enter the tolerance value in the window to the right of the **+ plus sign**.



NOTE: Values placed in the window to the right of the **– minus sign** will be ignored by **Symmetrical Tolerance**. The value in the window to the right of the **+ plus sign** is the only one that will be displayed in the **Symmetrical Tolerance** method.

Limits Format

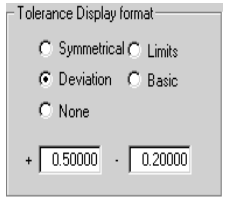


Limits Dimension Tolerance

This method displays a **Limit dimension** which displays a **maximum** and a **minimum** value.. The **maximum value** is the **dimension value plus the value entered in + window**. The **minimum value is the dimension value minus the value entered in + window.**

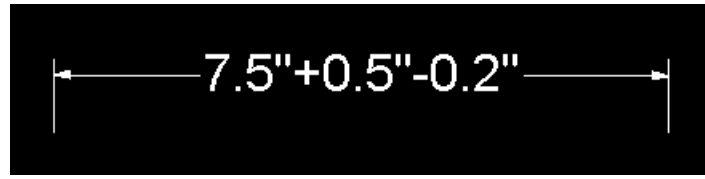
The actual **Dimension value** in the **Dimension** below is **7.5"** so the **upper limit is 7.5" + 0.5" = 8".** The **lower limit** is **- .2"** or **7.5" - .2" = 7.3"**

Deviation Format

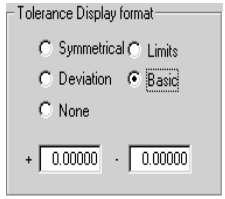


Deviation Dimension Tolerance

This format displays a **plus/minus tolerance value** suffix to the **Dimension text.** The **plus value** and **minus values** of the **Deviation** of the **Dimension** are entered into the **+** and **-** windows in the dialog box



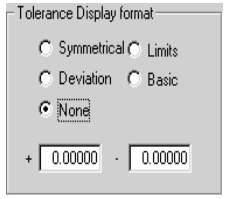
Basic Format



Basic Dimension Tolerance

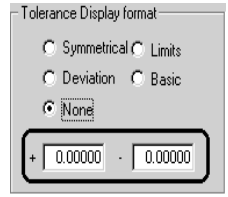
This method draws a **box** around the **dimension** signifying that its value may vary or be changed later.

None



None Disables the display of Dimension Tolerance

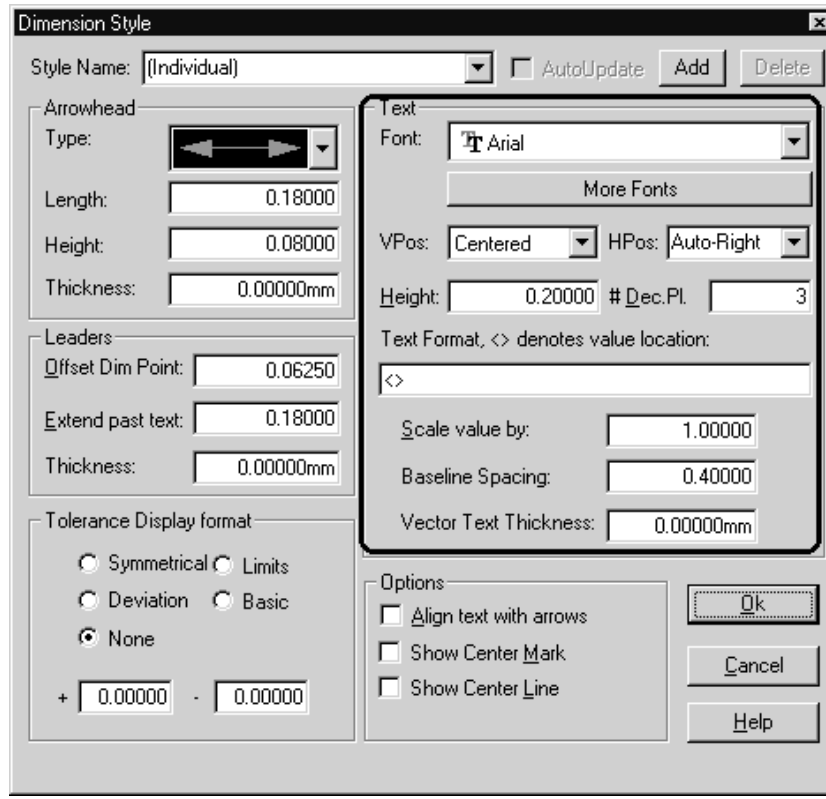
Tolerance Value



+ Window contains Maximum value for Tolerance

-- Window contains Minimum value for Tolerance

Dimension Text



Font Select a **FONT** for **Dimension Text** and numbers from the drop-list of available **fonts**.

More Fonts Select to **ADD FONTS** to the drop-down list. For more information on specific font types, see the **TEXT PROPERTIES** command.

VPos Select placement of dimension text relative to the dimension line, for horizontal dimensions. Available choices are **Above/Centered/Below**. Centered text will break the dimension line. During placement, Any of these settings can be manually overridden by holding down the **SHIFT** key.

HPos **Select placement of Dimension Text relative to the Dimension Line, for horizontal dimensions. Available choices:**

- **Auto-Left:** Dynamic cursor displays **Dim Text** in middle; but if text will not fit, the **Dim Text** position defaults to **LEFT**. Can be overridden during placement

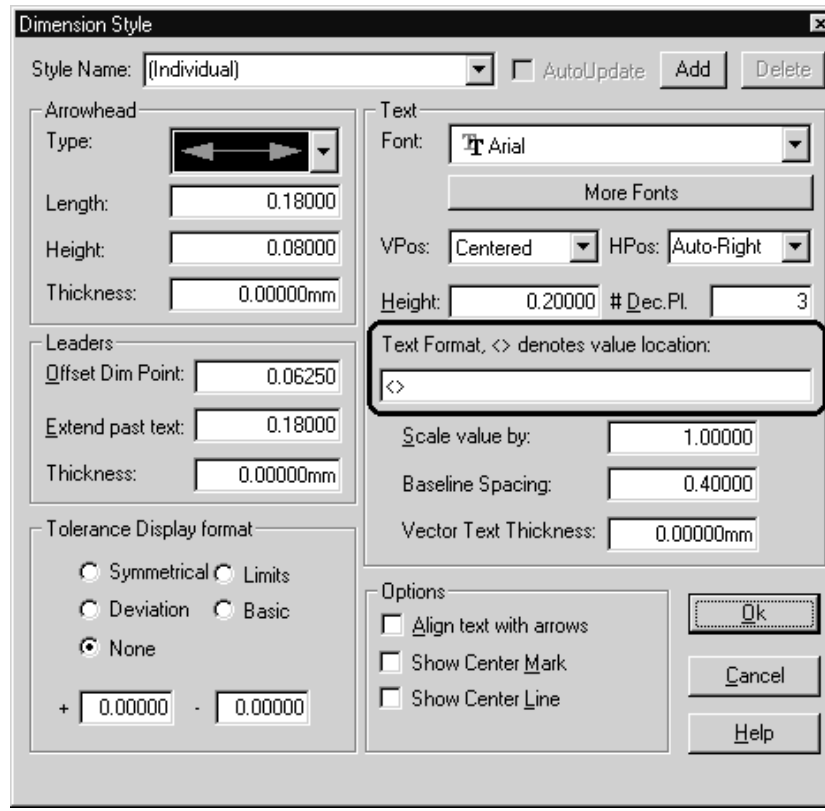
- **Auto-Right:** Dynamic cursor displays **Dim Text** in middle; but if text will not fit, the **Dim Text** position defaults to **RIGHT**. Can be overridden during placement.

- **Center/Left/Right: Dim Text** forced to **middle/left/right** position, respectively. Cannot be overridden during placement (unless manually repositioned by holding down **SHIFT key**).

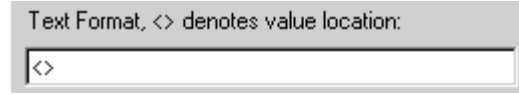
Height Specify the **height** of **dimension text** in drawing units.

 # Dec. Pl. **Specify the number of decimal places to be displayed for individual dimensions drawn in this style.**

Dimension Text Fixed Values Format



Text Format **Use this field to specify fixed prefix or suffix text that is appended to the true dimension value. The "<>" is used as a placeholder for the true value. If the value placeholder is omitted, the value is not included in the dimension output.**



Examples, where the actual dimension value is 18"

Input in Text Format Window	Dimension Displayed on Drawing
<>	18"
<> ROUGH BORE	18" ROUGH BORE
MULTIPLES OF <>	MULTIPLES OF 18"
MIN. <> CRAWL	MIN. 18" CRAWL
MIN. <24"> CRAWL	MIN. <24"> CRAWL

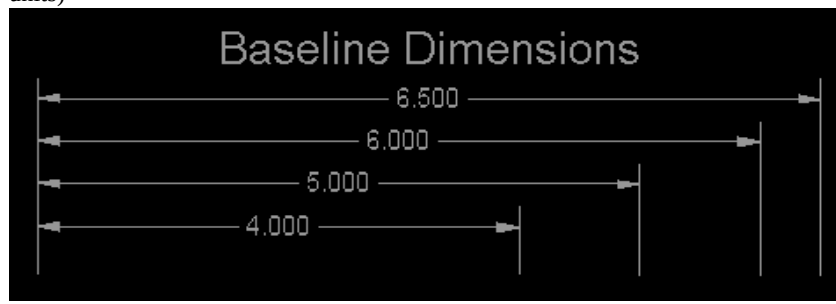
{empty} Delete Brackets {no text} Dimension is Blank

Scale value by

Typically used only to deal with the **AutoCAD®.DWG/DXF** import/export issues. This value is used as a scale factor. The true dimension is multiplied by the scale factor, and the scaled value is represented by the dimension text. Scale value = 1 represents actual dimension.

Baseline Spacing

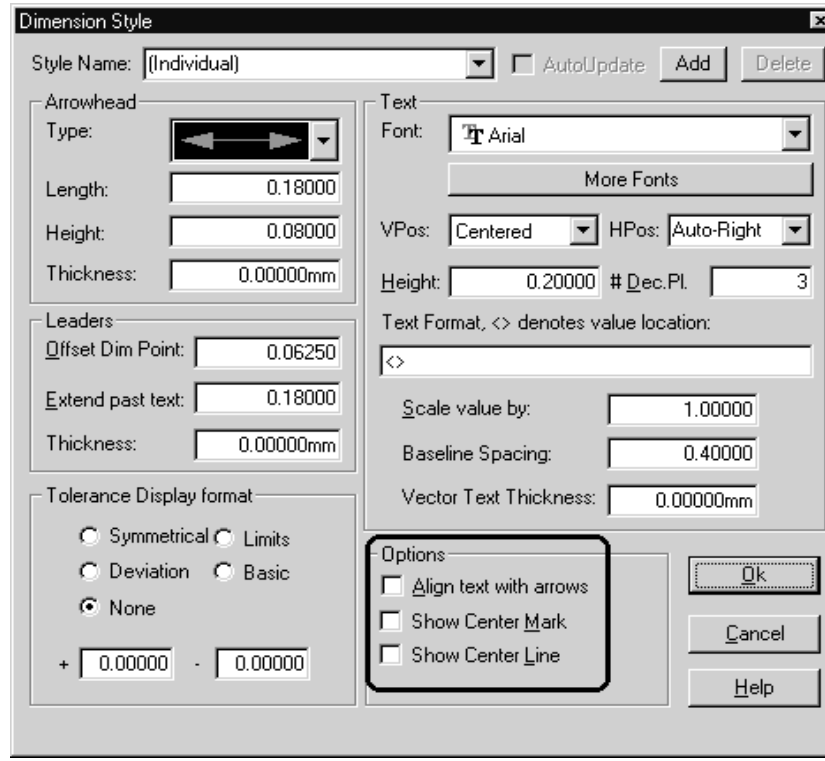
Specifies the **automatic spacing between BASELINE DIMENSIONS** (drawing units)



Vector Text Thickness

Pen thickness of **VECTOR** text. Only applicable if **FastCAD DOS .FNT** or **AutoCAD® .SHX** fonts are used (not **TrueType®**).

Dimension Options



Align text with arrows

Check to align text with the dimension line, regardless of orientation. If not checked, dimension text will always display horizontally.

Show Center Mark

Check to display center marks, which are small crosses **FastCAD** creates at the center of dimensioned circles or arcs.

Show Center Line

Check to display center lines, which consist of alternating short and long dashes. **FastCAD** automatically draws them as extension lines for dimensioned circles and arcs. Center marks are always created whenever center lines are drawn.

Chapter 12

Edit Commands

Modify and manipulate the entities in your drawing

The ***Edit*** *menu* commands let you modify entities that you have already drawn. You can alter *geometry*, such as the endpoint of a line or the radius of a circle. You can change *entity properties*, such as color or layer. With the advanced editing commands **FastCAD** provides, you can manipulate entities in almost any way you desire.

- ***Erase* commands let you delete entities from your drawing.**
- ***Edit* commands let you modify the geometry of entities you have drawn, either parametrically or with dynamic cursors that preview the changes.**
- ***Change* commands let you edit the properties of entities you have drawn, such as color, layer, fill style, etc.**
- ***Move* commands let you displace entities you have drawn, while *scale* and *rotate* commands let you transform entities.**
- ***Trim* commands let you shorten or extend entities.**
- ***Break* commands let you separate one entity into two (or more).**
- ***Front/Back* commands let you arrange entity order, so that they redraw in the sequence you desire.**
- ***Explode* commands reduce complex entities into their simpler components.**

You can *undo* all **Edit** commands. Left-click or right-click after any editing command to auto-repeat it.

Some Edit commands work on one entity at a time, while others allow you to select multiple entities before performing the command. ***Chapter 2: Communicating with* FastCAD,** deals in depth with entity selection. Choosing entities is an integral step in almost every editing command. **FastCAD** provides several options to optimize entity selection to the way you think and work. Review the methods and choose the one that works best for you.

Commands that *duplicate* entites are a wide and special subset of editing commands. As such, they are grouped under their own ***Copy*** *menu* heading ***(see Chapter 13: Copy Commands for more information)***

Erase icon

Erase

The **ERASE** command removes the selected entities from your drawing.

You cannot erase entities on hidden/frozen layers, hidden sheets, reference files, or **XREFs**. Entities within a part cannot be erased until the part is ungrouped or unlocked. Entities *within* a symbol reference cannot be erased unless the symbol referenced is exploded, or the definition is edited using the **EDSYMDEF** command.

Use the **UNDO** command to recover any unintended erasures. You can undo an **ERASE** as long as it was one of the last ten commands run.

Note: **FastCAD** erases entities from your screen by drawing over them in the background color. This may leave apparent holes in other entities, especially if the erased entities were filled or crosshatched. Select **REDRAW [View > Redraw]** to clean up the screen.

Text equivalent: ***ERASE***

Edit

Edit? Or Dynamic Edit?

FastCAD provides EDIT and DYNAMIC EDIT commands. At first glance, these might seem quite similar. Fundamentally, they are, since they both allow you to modify existing entities.

So when to use either command?

In general, use EDIT when you need to alter the geometry of an entity to known specifications. You aren't provided with visual aids, so you must type in new coordinates, radii, etc.

DYNAMIC EDIT is more flexible, because you can visually modify the entities. Modifiers or attach mode can also be used to connect or reference other entities.

For text, use EDIT to change the actual text or its properties (such as font, justification, etc.) Use DYNAMIC EDIT to visually reposition, resize, or rotate the text.

The **EDIT** command is a general-purpose command that lets you modify most entities one at a time, using dialog boxes for changing parametric values. When you invoke **EDIT**, the prompt reads “Entity to edit:”, and a pick box cursor appears on the drawing screen. Select a single entity (remember to select text entities on their base line). **EDIT** works differently depending on the type of entity you select.

EDIT is used to single edit the ***geometry*** of entities. If you wish to similarly edit an entity's ***properties*** (color, layer, line style, etc.), use the **PEDIT** command **[Edit > Properties]** instead.

Note: The values in dialog boxes can be modified with algebraic expressions. For example, select a circle. After pointing to the end of the **Radius** value in the ***Edit Circle*** *dialog*, enter “***2**”. The radius will double in length. Click **OK** to accept the value and display the larger circle.

Selectable entities:

LINES

- **Edit endpoint coordinates**
- **Toggle arrowheads at either endpoint**
- **Change the line style**

CIRCLES

- **Edit center point coordinates**
- **Edit radius value**

ARCS

- **Edit center point coordinates**
- **Edit radius value**
- **Edit start angle**
- **Edit angle width**
- **Flip start angle & angle width**
- **Toggle arrowheads at either endpoint**

ELLIPSES

- **Edit center point coordinates**
- **Edit major radius value**
- **Edit minor radius value**
- **Edit inclination value**

ELLIPTICAL ARCS

- **Edit minor radius value**
- **Edit inclination value**
- **Edit start angle**
- **Edit angle width**
- **Flip start angle & angle width**
- **Toggle arrowheads at either endpoint**

TEXT

- **Edit text**
- **Change text properties**
- **Convert to file text**

PATHS / POLYS / SPLINES

- **Edit node coordinates**
- **Toggle smoothing**
- **Toggle closed (poly) or open (path)**
- **Toggle arrowheads at either endpoint (open only)**
- **Edit trim parameters (open only)**
- **Change line style**

POINT

- **Edit node coordinates**

DIMENSIONS

- **Change any dimension style setting**

PICTURE

- **Change source bitmap image file**
- **Change picture options**

XREF

- **Change path\filename of XREF**

Wall network entities cannot be edited with **EDIT**. Instead, use the individual wall commands.

Other non-selectable entities include symbol references, grouped entities, and multipoly entities.

Symbol definitions can be edited using the **EDIT SYMDEF** command **[Insert > Edit Symbol Def…]**.

Text equivalent: ***EDIT***

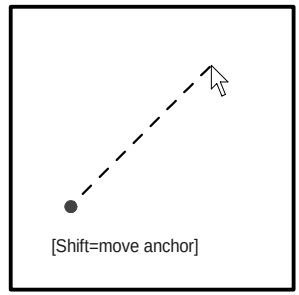
Dynamic Edit

The **DYNEDIT [Edit > Dynamic Edit]** command is a general-purpose command that lets you modify most entities one at a time, using visual screen methods to modify them.

Because most dynamic edit instances utilize dynamic cursors, the **[SHIFT=move CTRL=scale SHIFT+CTRL=rotate]** prompt alows the use of these keys to expand your editing capabilities.

If you need to edit an entity's geometry with precision, consider using the **EDIT** command **[Edit > Edit]**. To edit entity properties, use the **PEDIT** command **[Edit > Properties]**.

Dynamic edit does not work with ***elliptical arcs*** or ***WALL*** entities (there are special commands to edit walls).

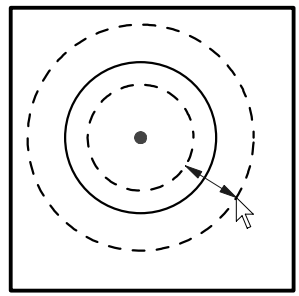


LINES

Repositions the end.

prompt reads **"Next point [done]"**

Use SHIFT to re-position the anchor point.

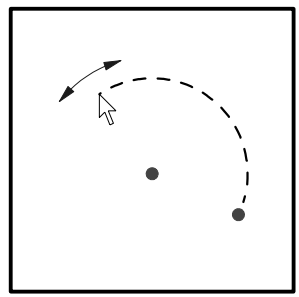


CIRCLES

Changes the **radius.**

prompt reads "**[SHIFT=move]**"

Use **SHIFT** to re-position the **center point**.



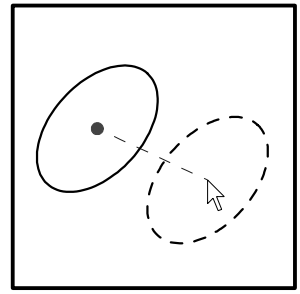
ARCS

Changes the **endpoint** (**arc end angle**).

prompt reads "**[SHIFT=move CTRL=scale]**"

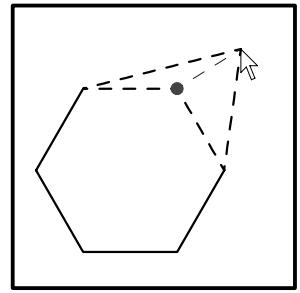
Use **SHIFT** to re-position the center point.

Use **CTRL** to change the radius.



ELLIPSES & ELLIPTICAL ARCS

Repositions the center point.



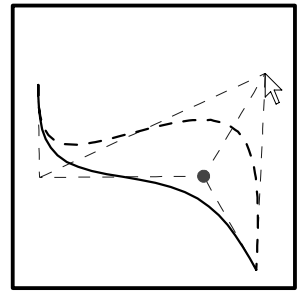
PATHS & POLYS

Repositions any node.

prompt reads "**[INS DEL <--> New position [done]**"

Use INSERT or DELETE to add or remove nodes.

Use LEFT ARROW or RIGHT ARROW to cycle through nodes. Right-click to terminate editing mode.



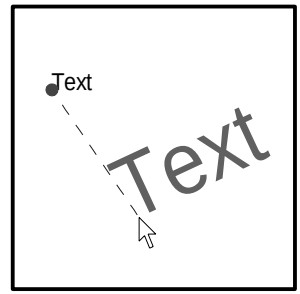
SPLINES & SPOLYS

Repositions any control point.

Use **INSERT** or **DELETE** to add or remove points (Bezier splines must have exactly four points).

Use **LEFT ARROW** or **RIGHT ARROW** to cycle through points.

Right-click to terminate editing mode.



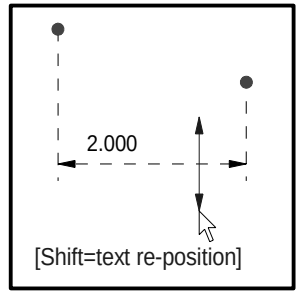
TEXT

Repositions resizes and rotates.

prompt reads "**[CTRL=scale CTRL+SHIFT=rotate]**"

Use **CTRL** to resize.

Use **SHIFT+CTRL** to rotate.



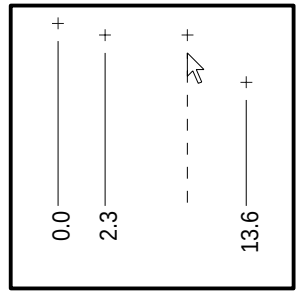
LINEAR DIMENSIONS

(**Linear/Horizontal/Vertical/Parallel**)

Repositions dimension Text.

prompt reads "**[Text baseline point (SHIFT=Spec Text Center]"**

Use **SHIFT** to manually re-position text.



ORDINATE DIMENSIONS

(**Linear/Horizontal/Vertical/Parallel**)

Inserts or Delete a dimension leader.

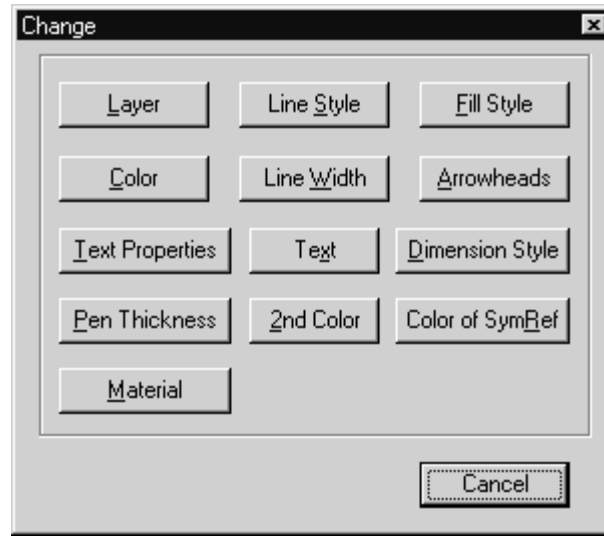
prompt reads "**Insert leader at [DEL to delete]**"

Use **DEL** and pick a leader to remove.Right-click to end the Dynamic Edit command.

Text equivalent: ***DYNEDIT***

Change

When you select **CHANGE**, the ***Change*** *dialog box* appears in the center od the screen, allowing you to select the properties of entities to modify. All entities have a common set of properties that can be edited on an individual basis:



Selecting any option invokes the specific **CHANGE** command for that property. For instance, if you click the **Color** button, **FastCAD** starts the **CHANGEC** command.

Use the **PEDIT** command to change multiple properties of a single entity.

Text equivalent: ***CHANGE***

Change > Properties

C**HANGE > PROPERTIES** lets you change multiple properties for any number of selected entities, using an interactive dialog box. Modified properties are applied to all selected entities. This command works like **PEDIT**, except that you can select more than one entity.

The properties you can edit are like those you can edit using the various CHANGE commands:

- **Layer**
- **Line Style**
- **Fill Style**
- **Line Width (geometric width)**
- **Pen Thickness (paper scale thickness)**
- **Color (entitys outline color)**
- **Fill [2nd] Color**
- **Text Properties (when applicable)**
- **Text (what the Text reads)**
- **Dimension Style (when applicable)**
- **Color of SymRef (applies to Symbols only)**
- **Arrowheads**

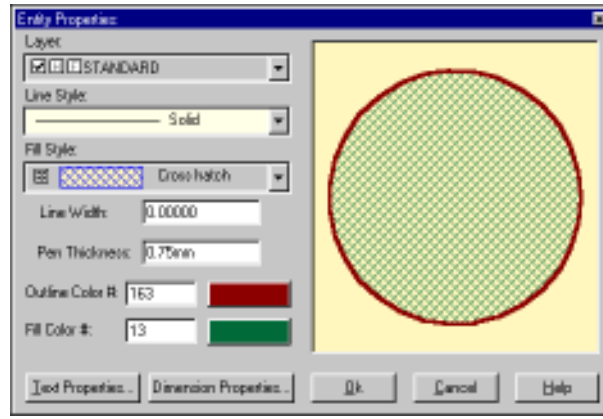
To change the properties of existing entities, select **[Edit > Change > Properties…]**:

1. Select entities to edit. Groups and multipolys select as such, and any changes affect the entire group or multipoly. Symbol references can be selected, but property changes affect the selected reference(s) only, never the symbol definition(s).
2. **FastCAD** displays the appropriate ***Entity Properties*** *dialog box*. Modify the desired properties and click **OK**.

FastCAD applies the modified properties to all of the selected entities.

Entity Properties dialog

FastCAD displays the ***Entity Properties*** *dialog box* when you invoke the **PCHANGE** command or **PEDIT** command **[Edit > Properties]**. Note that changes made in this dialog do not affect the current drawing properties in any way-just those of the selected entity. For example, if you change an entity's color, the current drawing color does not change.



Sample Displays an extents sample of the selected entity. As you make changes to its properties, the entity updates accordingly.

Layer Changes the entity layer. Initially, the current entity layer is displayed. To change layer, click to open a list of usable layers, and select a layer name.

Line Style Changes the entity line style. Initially, the current entity line style is displayed. To change line style, click to open a list of usable line styles, and select a line style name.

Fill Style Changes the entity fill style. Initially, the current entity fill style is displayed. To change fill style, click to open a list of usable fill styles, and select a fill style name.

Line Width Changes the entity line width. Initially, the current entity line width is displayed. To change line width, type in a new value (in drawing units).

Pen Thickness Changes the entity pen thickness. Initially, the current entity pen thickness is displayed. To change pen thickness, type in a new value. The value is in drawing units, but this can be overridden by specifying a mm (millimeter) or " (inch) suffix.

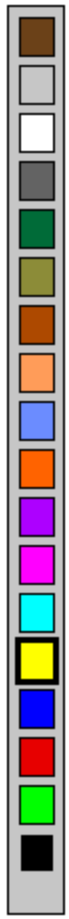
Outline Color # Changes the entity color. Initially, the current entity color is displayed. To change color, type in a new color ID#, or click the color button to

display the 256-color palette. When the color is changed, **Fill Color #** also changes to the same color.

Fill Color # Changes the fill style color, when applicable. This changes the text color in dimensions. Initially, the current entity fill color is displayed. To change fill color, type in a new color ID#, or click the color button to display the 256-color palette.

Text Properties Opens the *Text Properties dialog box*. To modify specific text properties, make desired changes in the dialog, then click **OK**.

Dimension Properties Opens the ***Dimension Styles*** *dialog box*. To modify specific dimension properties, make desired changes in the dialog, then click **OK**.

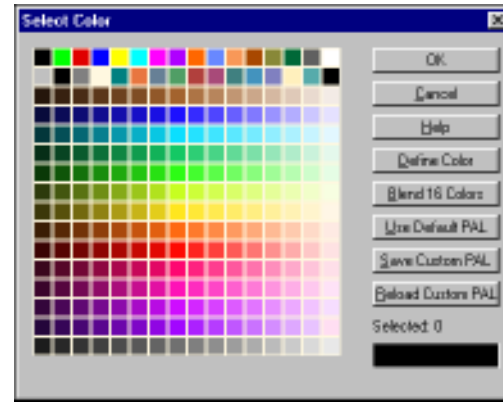


Color bar

In this figure, yellow (color #4) is the active color, as indicated by the heavy black border.

Change > Color

CHANGE > COLOR redraws selected entities in a different color.



To change entity colors, select **[Edit > Change > Color]**:

1. Select entities to modify.
2. The prompt reads "**Color value [dialog]:**". To select a new color:
   - Pick a color icon from the ***Color*** *bar*;
   - Right-click to display the 256-color palette, pick a new color, and click **OK**;
   - Type a color number and press ENTER.

FastCAD redraws the entity in the selected color.

Note: If an entity uses a secondary color, use the **CHANGEC2** command to change it. **CHANGEC** only changes an entity's primary color.

Text equivalent: ***CHANGEC***

Change > to Color of SymRef

CHANGE > TO COLOR OF SYMREF sets selected entities to "**color by block.**" This command is only relevant for entities that will be included in a symbol definition.

When an entity in a symbol has **By Block color**, it will behave similarly as if it were drawn in color #31. Entities in a ***symbol definition*** that have the by block setting will draw in the current color when inserted. It is recommended that this method be used rather than using the #31 "**no change**" color.

You can use the **EDSYMDEF** command **[Insert > Edit Symbol Def]** to modify an existing symbol. Reference entities that are changed to By Block color will then update to the color active at the time of the symbol insertion.

To quickly change **ALL** of a drawing's symbol definitions to By Block color, you can use the **CHANGESCBB [Insert > Change All Sym Defs > Change All Color by Block]** command or type "**CHANGESCBB**" at the command line.

Text equivalent: ***CHGCBB***

Change > Fill (2nd) color

CHANGE>FILL (2ND) COLOR lets you assign a secondary color to an entity.

When set, the secondary color is used for:

- Fill pattern color on valid entities. Valid entities include polygons, mulitpolys, circles, ellipses, walls, entities drawn with line width > 0.0, etc.
- Text on dimension entities

When entities are created, the secondary color is always the same as the primary color. **CHANGEC2** is the only way to change the secondary color.

To change entity secondary colors, select **[Edit > Change > Fill (2nd) color]**:

1. Select entities to modify.
2. The prompt reads "**Color value [dialog]:**". To select a new color:
   - Pick a different color from the ***Color*** *bar*;
   - Right-click to display the 256-color palette dialog, from which you can pick a new color and click **OK**;
   - Type a color number and press ENTER.

FastCAD redraws the entity with the selected secondary color (if a valid use exists).

Note that this command will not change an entity's primary color. Entities cannot be selected by their secondary color.

Text equivalent: ***CHANGEC2***

Change > Pen Thickness

CHANGE > PEN THICKNESS lets you change the pen width of an existing entity.

To change entity ***pen thickness***, select **[Edit > Change > Pen Thickness]**:

1. Select entities to modify.
2. The prompt reads "**Pen thickness:**". Type in a value. Note that the value is specified in current drawing units, so explicit entry, such as 1.5mm or 0.007, is recommended.

FastCAD redraws the entity using the new pen width.

Use the **PTHICK** command **[Specs > Pen Thickness]** to set the current pen width. You can also access the ***Pen Thickness*** *dialog* by clicking directly on the ***Pen Thickness*** *indicator* on the ***Status*** *bar*.

Text equivalent: ***CHANGEPT***

Change > Layer

CHANGE > LAYER moves entities to a different layer. The target layer must already exist.

To change entity ***layer***, select **[Edit > Change > Layer]**:

1. Select entities to modify.
2. The prompt reads "**Layer name or # [dialog]:**". To select a new layer:
   - Type an existing layer name and press ENTER.
   - Type a layer ID# and press ENTER.
   - Right-click to display the ***Select Layer*** *dialog*, and pick the layer name.

FastCAD moves the entity to the target layer, and the command terminates.

Important: Use of this command does not change the ***current layer***. Note that the "**L**:" setting on the ***Status*** *bar* does not change. To change the ***active layer***, use the **SETLAYER** command **[Specs > Layer Options > Set Current]**.

Text equivalent: ***CHANGEL***

Change > Line Style

Change Line Style icon

CHANGE > LINE STYLE redraws entities in a different, pre-defined line style.

To change the ***line style*** of an existing entity, select **[Edit > Change > Line Style]**:

1. Select entities to modify.
2. The prompt reads "**Line style name or # [dialog]:**". To select a new line style:
   - Type an existing line style name and press ENTER.
   - Type a line style ID# and press ENTER.
   - Right-click to display the ***Line Style*** *dialog*, and pick a line style name.

FastCAD redraws the entity in the target ***line style***, and the command terminates.

Important: Use of this command does not change the ***current line style***. Note that the "LS:" setting on the ***Status*** *bar* does not change. To change the active line style, use the **LINE STYLE** command **[Specs > Line Style]**.

Text equivalent: ***CHANGELS***

Change > Fill Style

Change Line Style icon

CHANGE > FILL STYLE redraws applicable selected entities with a different, pre-defined ***fill style.***

To change the ***fill style*** of an existing entity, select **[Edit > Change > Fill Style]**:

1. Select entities to modify. The selected entities must be of a type valid for fills (polygons, SPolys, circles, ellipses, mulipolys).
2. The prompt reads "**Fill style name or # [dialog]:**". To select a new fill style:
   - Type an existing ***fill style*** name and press ENTER.
   - Type a fill style ID# and press ENTER.
   - Right-click to display the ***Fill Style*** *dialog*, and pick a style name.

FastCAD redraws the entity with the target ***fill style***, and the command terminates.

Important: Use of this command does not change the ***current fill style***. Note that the "**FS:**" setting on the ***Status*** *bar* does not change. To change the active ***fill style***, use the **FILL STYLE** command **[Specs > Fill Style**].

Change > Material

CHANGE > MATERIAL applies selected entities with a different, pre-defined ***material*** for on-screen and rendering to a file.

This command is NOT available in FastCAD LE.

To change the ***Material*** of an existing entity, select **[Edit > Change > Material]**:

1. Select **3d** entities to modify. The selected entities must be of a type valid for materials (**3d** polygons, **3d** circles, **3d** ellipses, solids, boxes, cylinders, tab profiles, spheres, sphere sections, domes, ruled surfaces, coons surfaces).
2. The prompt reads "**Material name or # [dialog]:**". To select a new material:
   - Type an existing material name and press **ENTER**.
   - Type a material ID# and press **ENTER**.
   - Right-click to display the ***Select Material*** *dialog*, and pick a material name.

FastCAD renders the entity with the new material, and the command terminates.

Important: Use of this command does not change the current material style. Note that the "**M:**" setting on the ***Status*** *bar* does not change. To change the active material, use the **MATERIAL** command **[Render > Material]** or **[Specs > Material]**.

Text equivalent: ***CHANGEMS***

Change > Line Width

CHANGE > LINE WIDTH redraws selected entities with user-modified line widths. Each entity possesses a line width property that is set at the time the entity is created.

1. Select entities to modify.
2. The prompt "**New width:**", where ***x.x*** is the last value used for this command. Type in the desired line width and press **ENTER**, or right-click to accept the prior value.

FastCAD redraws the entity with the new line width, and the command terminates.

Text equivalent: ***CHANGELW***

Change > Dimension Style

CHANGE > DIMENSION STYLE redraws selected ***dimensions*** with user-modified properties. Dimension properties include ***arrowhead***, ***leaders***, ***tolerances***, ***center lines***, ***text metrics***, and ***text positioning***. You can either apply a ***defined dimension style*** to the entire dimension(s), or just change individual properties.

This command operates similarly to **CHANGE > TEXT PROPERTIES**. If you choose to apply a ***named dimension style***, observe the settings at the bottom of the ***Dimension Style*** *dialog*. If **"Change only touched items"** is checked, then you must "**touch**" each edit control with a mouse click in order for that property change to take effect. Non-touched properties are not changed. However, if you check **"Change everything"**, then ***all dimension properties change to the displayed settings***. Please see the **DIMENSION STYLES** command for more information on ***named dimension styles.***

To change any dimension properties of an existing dimension, select **[Edit > Change > Dimension Style]**:

1. Select dimension entities to modify.
2. **FastCAD** automatically displays the ***Dimension Styles*** *dialog box*. The current dimension specifications are displayed. Clicking on and/or changing any value will result in the selected entities acquiring the new property values. Unless the **"Change everything"** option (**located at the bottom of the dialog**) is checked, then only options that are "**touched**" will affect the selected dimension(s).

   This action does not change the current dimension specifications, only those of the selected dimension entities.
3. Click **OK** to accept the changes and close the dialog box. The dimension(s) will redraw with the updated properties.

Text equivalent: ***CHANGED***

Change > Text

CHANGE > TEXT changes ***selected text entities*** to the ***entered text***. Use this command to change multiple, existing text entities to the ***same text***. Each changed entity keeps its own location and properties.

1. **FastCAD** displays the ***Text*** *dialog box*. Type in the new text. For longer text, you may check the **Multi-line** option. Click the **OK** button when done.
2. Select text entities to modify. When you select **OK-Do it**, **FastCAD** changes all selected text to the new value.

To change a ***single text entity*** to a new value, use the **EDIT** command **[Edit > Edit]**.

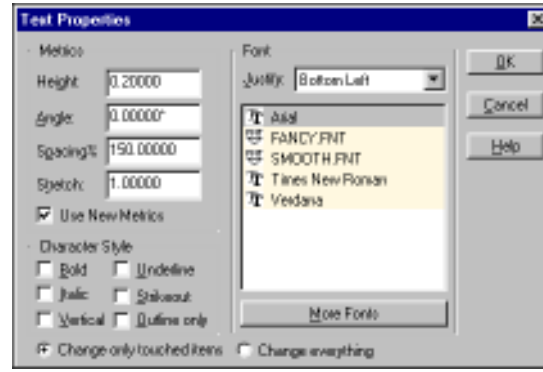
To change any text metrics or properties of an existing text entity, use the **CHANGE > TEXT PROPERTIES** command **[Edit > Change > Text Properties]**.

Text equivalent: ***CHANGETX*** *or* ***CHANGEX***

Change > Text Properties

CHANGE > TEXT PROPERTIES redraws selected text with user-modified properties. Text properties include ***Font***, *Metrics*, and ***Character style***.

This command operates similarly to **CHANGE > DIMENSION PROPERTIES**. If you choose to apply a ***named text style***, observe the settings at the bottom of the ***Text Properties*** *dialog*. If **"Change only touched items"** is checked, then you must "**touch**" each edit control with a mouse click in order for that property change to take effect. Non-touched properties are not changed. However, if you check **"Change everything"**, then all dimension properties change to the displayed settings. Please see the **TEXT PROPERTIES** command for more information on ***named text styles***.



Text Properties dialog

To change any ***text properties*** of an existing text entity, select **[Edit > Change > Text Properties]**:

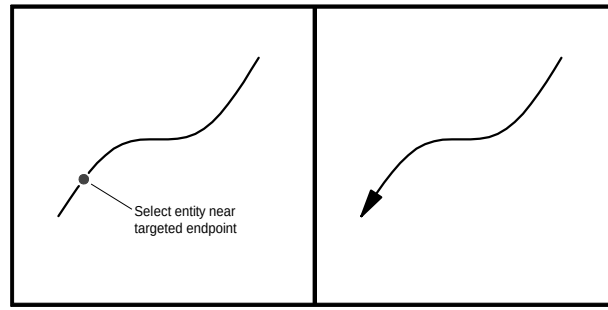
1. Select text entities to modify.
2. **FastCAD** automatically displays the ***Text Properties*** *dialog box*. The current text specifications are displayed. Clicking on and/or changing any value will result in the selected entities acquiring the new property values. Unless the **"Change everything"** option (**located at the bottom of the dialog**) is checked, then only options that are "**touched**" will affect the selected text.

   This action does not change the current text specifications, only those of the existing selected entities.

3. Click **OK** to accept the changes and close the dialog box. The selected text will redraw with the modified properties.

Text equivalent: ***CHANGET***

Change > Arrowhead



CHANGE > ARROWHEAD toggles on or off the ***current Arrow style*** on the nearest end of a selected ***line***, ***arc***, ***path*** or ***spline***. To add an arrow to one of these entities, invoke the command and pick an entity near an endpoint. **FastCAD** adds an arrowhead to the nearest endpoint.

To reverse the operation, apply the arrowhead command again to the entity, or select the **UNDO** command **[Clip > Undo]**.

To set the ***arrow style*** for entities that don't yet have an arrowhead, use the **ASTYLE** command **[Specs > Arrowhead Style]**.

To change the arrow style on an entity that already has an arrowhead, use the **CHANGEAS** command **[Edit > Change > Arrow Style]** and select a new style.

Text equivalent: ***AHEAD***

Change > Arrow Style

Use **CHANGE > ARROW STYLE** to change the existing ***arrow style*** on an entity. This command only affects line, path, arc, or spline entities that already have arrowheads. ***Arrowheads*** are ***added*** or ***removed*** using the **AHEAD** command. If you apply **CHANGEAS** to an entity without arrowheads, the edit will have no net effect.

Style Select the arrowhead style from the drop list. The ***arrowhead style*** you select will be applied to currently selected entity.

Length Sets the length of the arrowhead in drawing units. If you edit this value, the height will automatically update to maintain the arrow's aspect ratio.

Height Sets the height of the arrowhead in drawing units, measured as the total height of both arrowheads. When the ***Length value is changed***, ***Height updates by the same factor to maintain the arrow's aspect ratio***. You can override this suggested value by entering a ***different height***.

Text equivalent: ***CHANGEAS***

Change > Radius

Use **CHANGE > RADIUS** commands allows you to select one or more **CIRCLES** and **ARCS** change the **RADIUS** of all of them at one time to a new **RADIUS** you specify and the **CIRCLES** and **ARCS original CENTER XYZ locations** will be retained.

To change **RADIUS** of existing **CIRCLES** and **ARCS** select the **[Edit > Change > Radius]**:command.

1. The prompt reads: "**New radius:**'.

   Type a value like **3"** and press **ENTER**..

2. The prompt reads: "**Select entities**".

   Select the **CIRCLES and ARCS** that you want to have the new **RADIUS** value and press **ENTER.**

FastCAD v7 changes the selected entities to the new **RADIUS** and the **original CENTER XYZ locations** of the **CIRCLES** and **ARCS** remain the same.

Text equivalent: ***CHGRAD***

Change > Diameter

The **CHANGE > DIAMETER** command allows you to select one or more **CIRCLES** and **ARCS** change the **DIAMETER**of all of them at one time to a new **DIAMETER**you specify and the **CIRCLES** and **ARCS original CENTER XYZ locations** will be retained.

To change **DIAMETER**of existing **CIRCLES** and **ARCS** select the **[Edit > Change > Diameter]**:command.

1. The prompt reads: "**New diameter:**'.

   Type a value like **3"** and press **ENTER**..

2. The prompt reads: "**Select entities**".

   Select the **CIRCLES** and **ARCS** that you want to have the new **DIAMETER** value and press **ENTER.**

FastCAD v7 changes the selected entities to the new **DIAMETER** and the **original CENTER XYZ locations** of the **CIRCLES** and **ARCS** remain the same.

Text equivalent: ***CHGDIA***

Properties

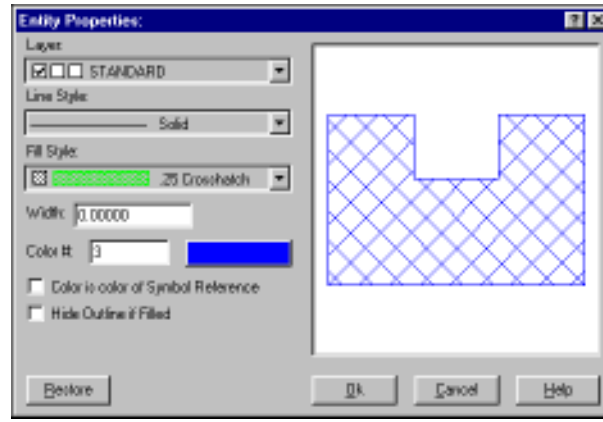
PROPERTIES allows you to edit the standard properties of ***a single selected entity*** through an interactive dialog box. When you wish to apply property changes to a single entity, **PEDIT** can often be much quicker than using a series of **CHANGE** commands.

The properties you can edit are like those you can edit using the various **CHANGE** commands:

- **Layer**
- **Line Style**
- **Fill Style**
- **Line Width**
- **Pen Thickness**
- **Outline Color**
- **Fill Color**
- **Text Style (when applicable)**
- **Dimension Style (when applicable)**

To change the ***properties*** of ***an existing entity***, select **[Edit > Properties]**:

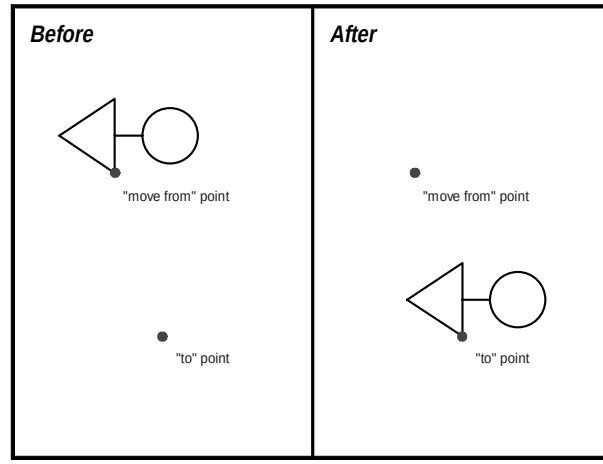
1. **FastCAD** displays a pick cursor. Select ***any single entity***. You cannot pick multiple entities with this command. A single entity can be discretely selected even if it is grouped. ***Multipolys select as a single entity*** (a**ny changes affect the entire multipoly**). ***Symbol references*** can be selected, but all property changes are made to the ***selected reference(s) only***, ***not to the symbol definition***.
2. **FastCAD** displays the *Entity Properties dialog box* (see **PCHANGE** command). Modify the desired properties and click **OK**.



Entity Properties dialog box

Text equivalent: ***PEDIT***

Drag




Drag icon

DRAG moves selected entities ***dynamically*** - you see the entities moving on-screen. The **DRAG** command utilizes a displacement (distance and direction) that you specify with two points.

To ***drag*** entities, select **[Edit > Drag]:**

1. Select entities to drag.
2. The prompt reads "**Origin:**". To specify a base point for the move, pick the point or type numeric coordinates.

   FastCAD displays the selected entities as an outline cursor that moves with your mouse.
3. The prompt reads "**CTRL=scale SHIFT+CTRL=rotate**". As you move the mouse, the entities move to follow. If snap locking is turned on, the entities jump over positions between snap points. Pick a point when the entities are where you want them positioned.

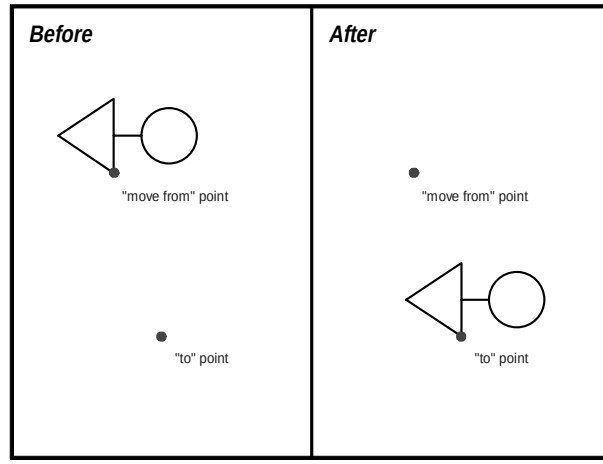
FastCAD moves the selected entities to their new position.

This command uses dynamic cursors. Because of this capability, the functionality of the **DRAG** command can go far beyond simply repositioning entities and makes it much more powerful than the **MOVE** command. Holding down keys while moving the mouse provides the following action:

- **CTRL** dynamically scales objects, making them larger or smaller depending upon direction of the mouse movement.
- **CTRL+SHIFT** dynamically rotates objects about the "**Origin:**" point. Potentially, you may find this more useful than using the **ROTATE** command **[Edit > Rotate]**.

Text equivalent: ***DRAG***

Move



MOVE ***moves*** the selected entities in relation to the rest of the drawing, using a displacement (distance and direction) that you specify with two points.

To ***move*** entities, select **[Edit > Move]**:

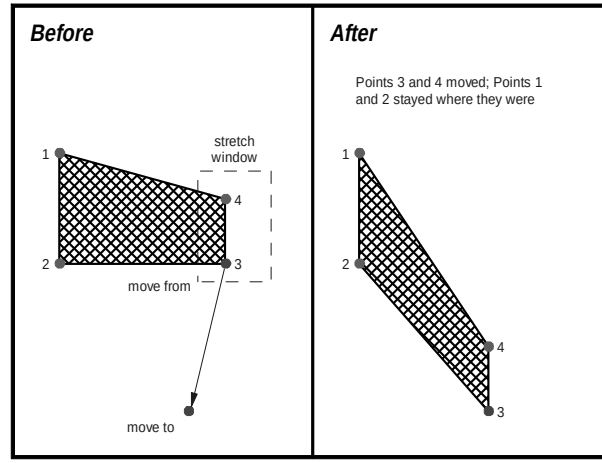
1. Select entities to move.
2. The prompt reads "**Move from point:**". Select a base point for the move with the crosshairs or type a numeric coordinate. Use ***modifiers*** such ***endpoint*** for precision.
3. The prompt reads "**Move to point:**". Select a second point with the cursor or by typing in numeric coordinates. Use ***modifiers*** such as ***endpoint*** for precision.

FastCAD calculates the distance and direction between the points, and ***moves*** the selected entities accordingly.

You can force the selected entities to move ***only horizontally*** or ***only vertically*** by turning on ***orthogonal locking*** by clicking the **ORTHO** button.

Text equivalent: ***MOVE***

DynStretch



DYNSTRETCH reshapes most entities by moving some of their end or node points in relation to the others. **DYNSTRETCH** differs from the regular **STRETCH** command by its use of dynamic cursors. **DYNSTRETCH** displays a ghost image of the entities as they are stretched to help you visually distort the selected entities.

When stretching dimensioned entities, you should select the actual **DIMENSION**(s) as well. A stretched **DIMENSION** ***will automatically update its new value***. You can verify this by watching the dynamic cursors during a **DYNSTRETCH** operation involving **DIMENSIONS**.

To ***Dynamically Stretch*** entities with dynamic cursors, select **[Edit > DynStretch]**:

1. Select the entities to stretch.

2. The prompt reads **'First corner:'**.

FastCAD is asking you to ***draw a selection window around the end points or nodes you want to move*** **[STRETCH]**.

This selection window you draw must include the actual entity points. In Step 1, you select the entities you want to **STRETCH** then you select with a ***window*** which points on those entities that will be modified. Points not selected in this step remain fixed.

If you enclose entities in this step that were not selected in **Step 1**, they are ignored and not modified in any way.

3. The prompt reads: **'Opposite corner:'**.

Finish drawing the ***selection window around the stretch points.***

4. The prompt reads: **'Origin:'**.

To select a ***base point for the move***, pick a point using a ***modifier*** for precision or type ***numeric coordinates***.

FastCAD displays dynamic cursors that preview the **STRETCH** based on cursor movement.

5. The prompt reads: **'SHIFT=move CTRL=scale CTRL+SHIFT=rotate:'**.

Use the dynamic cursors to pick a point in the drawing window.

This point indicates the distance and direction you want the selected points to move (in relation to the origin specified in **Step 4**). If you use modifiers in this step, the dynamic cursors are suspended.

FastCAD moves the endpoints that were inside the stretch window, leaving other endpoints where they were.

Stretch Tips:

· You can force the selected points to **move only horizontally** or **only vertically** by turning on **orthogonal locking**

· If you enclose the **center** of a **circle**, **ellipse** or **arc** in the selection window these entities are moved ***by the distance of the*** **STRETCH**.

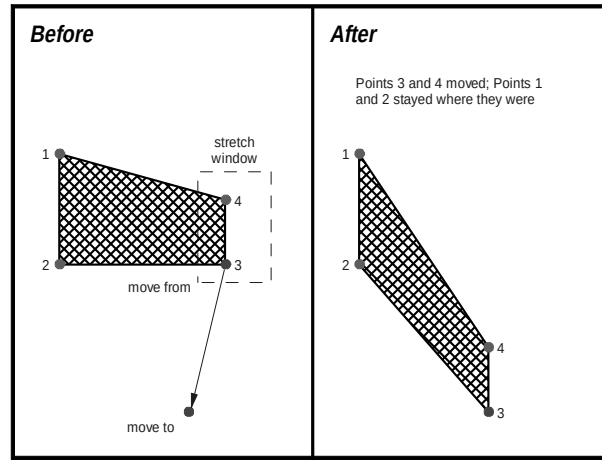
This command uses **dynamic cursors**. In ***Dynamic Track mode***, holding down keys while moving the mouse provides the following action:

- **CTRL dynamically scales** objects, making them larger or smaller depending upon direction of the mouse movement. In **STRETCH**, the scale origin stays fixed relative to the selected entities, not it's absolute coordinates when specified.

- **CTRL+SHIFT dynamically rotates** objects about the 'Origin:' point. In **STRETCH**, the rotate origin stays fixed relative to the selected entities, not it's absolute coordinates when specified.

Text equivalent: ***DSTRETCH***

Stretch



STRETCH ***reshapes*** most entities by moving some of their end or node points in relation to the others.

To ***stretch*** entities, select **[Edit > Stretch]**:

1. Select the entities to stretch.
2. The prompt reads "**First corner:**". **FastCAD** is asking you to draw a selection window around the end points or nodes you want to move. ***This selection window you draw must include the actual entity points***. In Step 1, you ***select the entities you want to stretch***, then you ***select which points on those entities will be modified***. Points not selected in this step remain fixed.

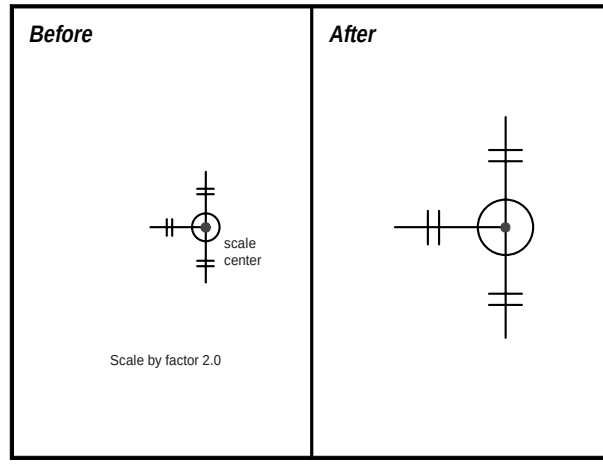
   If you enclose entities in this step that were not selected in Step 1, they are ignored and not modified in any way.
3. The prompt reads "**Opposite corner:**". Finish drawing the selection window for stretch points.
4. The prompt reads "**Origin:**". To select a base point for the move, pick a point or type numeric coordinates. Use ***modifiers*** such as ***endpoint*** for precision.

   FastCAD displays dynamic cursors that preview the stretch based on cursor movement.
5. Pick a point to indicate the *distance* and *direction* you want the points in the window to move. Use ***modifiers*** such as ***endpoint*** for precision.

FastCAD ***moves the endpoints that were inside the stretch window***, leaving other endpoints where they were unless the entities were selected in step one. If so they will be ***moved*** by the amount of the ***stretch distance***.

Text equivalent: ***STRETCH***

Scale > Scale Value



SCALE > SCALE VALUE ***shrinks*** or ***enlarges*** the selected entities based on a ***scale value you enter***.

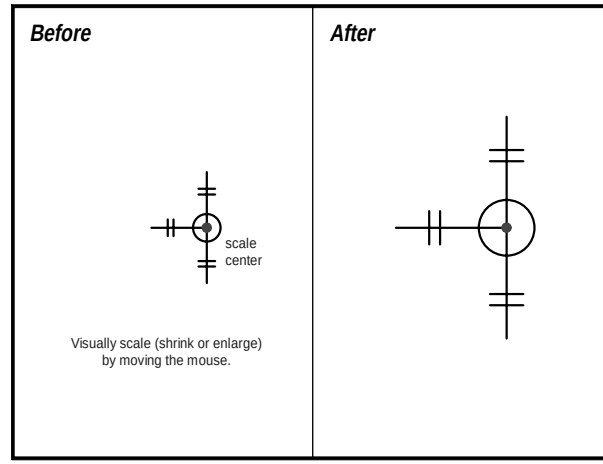
To ***scale*** entities, select **[Edit > Scale > Scale value]**:

1. Select entities to ***scale***.
2. The prompt reads "**Scale by factor [n]:**". If you wish to accept the default, right-click or press **ENTER**. To enter a new scaling factor, type a scaling value and press ENTER. ***Scaling factors > 1 will make entities larger. Scaling factors < 1 will make entities smaller.*** For example, to make the selected entities twice as large as they are now, type "**2**" and press **ENTER**. To make them half as large, type "**0.5**" and press ENTER.
3. The prompt reads "**Origin point [x,y,z]:**". **FastCAD** scales the entities about this ***scale center***. Use ***modifiers*** such as ***endpoint*** for precision. To specify define the origin:
   - Right-click or press **ENTER** to accept the default or prior center.
   - Pick a point in a drawing window.
   - Type numeric coordinates and press **ENTER**.

FastCAD redraws the entities at the ***new scale***. The scale center remains fixed.

Text equivalent: ***SCALE***

Scale > Dyn Scale



SCALE > DYNAMIC SCALE visually ***shrinks*** or ***enlarges*** selected entities, using a ***dynamic cursor***.

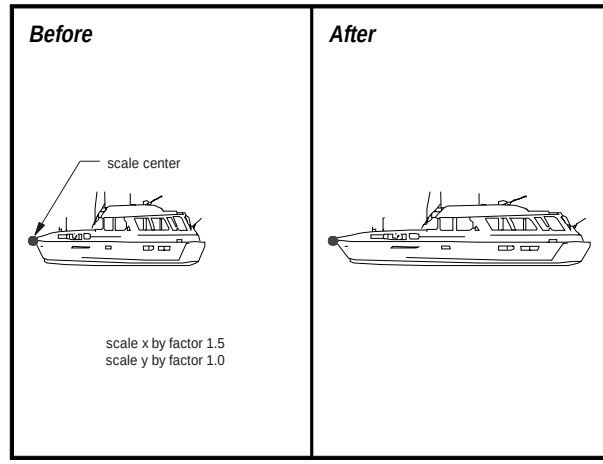
To ***dynamically scale*** entities, select **[Edit > Scale > Dyn Scale]**:

1. Select entities to ***scale***.
2. The prompt reads "**Origin:**". The drag from point is the point around which **FastCAD** should scale the entities (the "**scale center**"). To define the scale center:
   - Pick a point in a drawing window, or;
   - Type numeric coordinates and press **ENTER**.
3. The prompt reads "**SHIFT=move CTRL=scale SHIFT+CTRL=rotate:**". Move the mouse on the screen to visually resize the selected entities. They will shrink or grow depending on the direction of mouse movement. When you are satisfied with the size of the entities, left-click to create the scaled entities.

The "**SHIFT=move CTRL=scale SHIFT+CTRL=rotate:**" prompt tells you that dynamic cursors are in effect. While scaling the selected entities, you can also move the scale center or rotate the entities. To ***move the scale center***, hold down the **SHIFT** key while moving the mouse. To ***rotate*** about the ***scale center***, hold down the **SHIFT+CTRL** keys while moving the mouse.

Text equivalent: ***DYNSCL***

Scale > ScaleXYZ



SCALE > SCALE XYZ reshapes selected entities, scaling them by different amounts **horizontally** (**x**) and **vertically** (**y**) and **depth** (**z**).

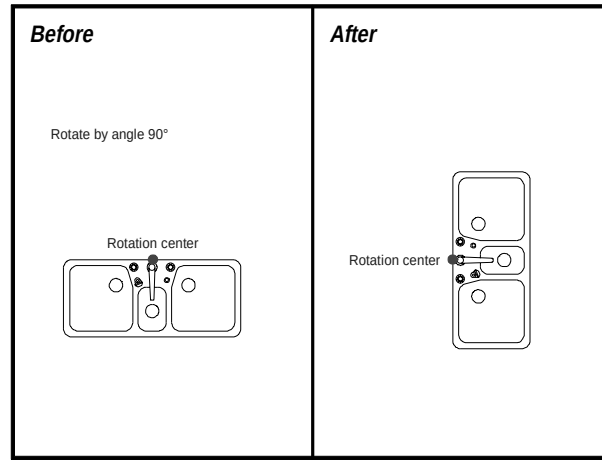
To ***scale XYZ*** entities, select **[Edit > Scale > Scale XYZ]**:

1. Select entities to ***scale***.
2. The prompt reads "**Scale X by factor [n]:**". If you wish to accept the default, right-click or press **ENTER**. To enter a new scaling factor, type a value and press **ENTER**. ***Scaling factors > 1 will make entities larger***. ***Scaling factors < 1 will make entities smaller.*** For example, to make the selected entities twice as large in the X direction as they are now, type "**2**" and press **ENTER**. To make them half as large in the **X** direction, type "**.5**" and press **ENTER**.
3. The prompt reads "**Scale Y by factor [n]:**". Select or enter a value as you did for **Scale** X.
4. The prompt reads "**Scale Z by factor [n]:**". Select or enter a value as you did for **Scale Y**..
5. The prompt reads **"Origin point [x,y,z]:**". The ***scale center*** is the point around which **FastCAD** should scale the entities. To define the ***scale center***:
   - Right-click or press **ENTER** to accept the default or ***prior center***;
   - Pick a point in a drawing window, or;
   - Type numeric coordinates and press **ENTER**.

FastCAD redraws the entities at the new **X, Y** and **Z** scales. The scale center remains fixed.

Text equivalent: ***SCALEXYZ***

Rotate > Rotate ang



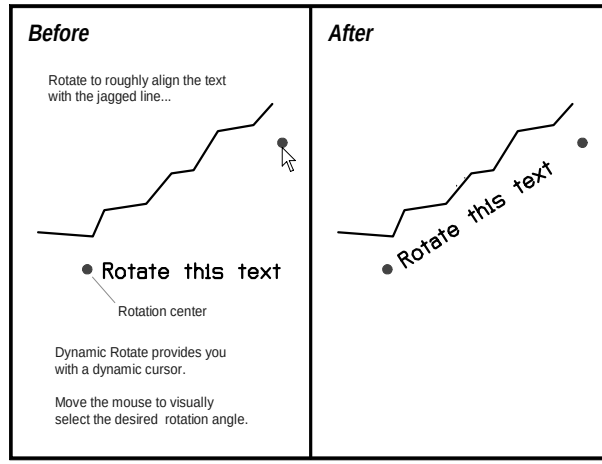
ROTATE > ROTATE ANGLE turns entities around a ***specified center*** and ***angle***. To rotate entities, select **[Edit > Rotate > Rotate ang]:**

1. Select entities to ***rotate***.
2. The prompt reads "**Rotate by Angle [n]:**", with the last angle you used suggested as a default or prior value. You can:
   - Right-click or press **ENTER** to accept the default or ***prior rotation***.
   - Type a positive or negative angle, which can include values like **90.0005°** or **22+1/2°.** (**Remember that positive angles turn counterclockwise.**) For to turn the entities **45° counterclockwise** type "**45**" and press **ENTER**.
3. The prompt reads "**Origin point [x,y,z]:**". For **2D** work, this point is the ***center*** about which **FastCAD** rotates the entities. To specify the ***rotation center***:
   - Right-click or press **ENTER** to accept the default or ***prior center***.
   - Pick a point in a drawing window.
   - Type numeric coordinates and press **ENTER**.
4. The prompt reads "**Rotate Axis Point [XY plane]:**". For **3D** rotation, **FastCAD** rotates the selected entities around any axis in three-dimensional space. This axis is defined by the ***Origin point*** specified in the previous step, and a point specified in this step. ***Positive angles rotate counterclockwise as you look along the axis from the center to the axis endpoint***. To specify the ***axis point***:
   - Right-click or press **ENTER** to accept the **XY** plane.
   - Pick a point on the axis. You'll get best results if you place the axis point farther away from you than the center of rotation.

FastCAD ***rotates*** the entities around the ***axis***, which remains fixed.

Text equivalent: ***ROTATE***

Rotate > Dyn Rotate



ROTATE > DYNAMIC ROTATE turns selected entities around a specified center. It works like the **ROTATE** command, except that it lets you visually see and specify the rotation angle using dynamic cursors.

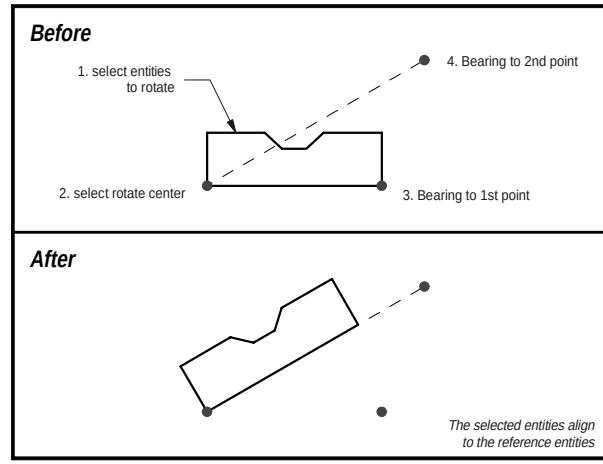
To ***dynamically rotate*** copies of entities, select **[Edit > Rotate > Dyn Rotate]**:

1. Select entities to ***rotate***.
2. The prompt reads "**Origin**:". The ***origin point*** is the point around which **FastCAD** should turn the entities. To define this ***origin***:
   - Pick a point in a drawing window.
   - Type numeric coordinates and press **ENTER**.
3. The prompt reads "**SHIFT=move CTRL=scale SHIFT+CTRL=rotate:**". Move the mouse on the screen to visually rotate the selected entities. They will rotate to the bearing defined by the cursor position to the rotation center. When you are satisfied with the rotation of the entities, left-click to draw the rotated entities.

The "**SHIFT=move CTRL=scale SHIFT+CTRL=rotate:**" prompt tells you that dynamic cursors are in effect. While rotating the entities, you can also move the rotation point or scale the selected entities. To move the rotation point, hold down the SHIFT key while moving the mouse. To scale about the rotation center, hold down the CTRL key while moving the mouse.

Text equivalent: ***DYNROT***

Rotate XY Align



Use the **ROTATE XY ALIGN** command to align selected entities to an existing reference entity by rotation. The rotation angle is derived from selected points.

To ***rotate align*** entities, select **[Edit > Rotate > Rotate XY Align]**:

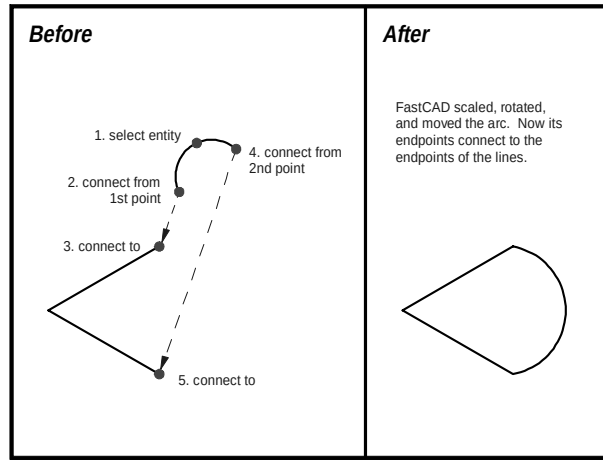
1. Select entities to ***rotate***.
2. The prompt reads "**Origin point [x,y,z]:**", where **x,y** is the ***last center point*** selected. You can pick a ***center point,*** type in numeric coordinates, or right-click to accept the default value.
3. The prompt reads "**1st bearing point:**". The rubber band cursor stretches from the center point. Pick a point to establish the ***bearing*** of the entities to be rotated.
4. The prompt reads "**2nd bearing point:**". The rubber band cursor again stretches from the center point. Pick a point to establish the new bearing for the selected entities. Selecting another point on the reference entity works best, provided that it emanates from the rotate center point.

FastCAD *rotates* the entities to the same bearing as the reference entity.

This command works best when lines on both the target and reference entities share the rotation center point. This allows you to easily establish bearing angles. **ROTALN** works by determining the difference between the two bearing angles, and then rotating the entities to that difference.

Text equivalent: ***ROTALN***

Connect



CONNECT moves, scales, and rotates an entity or entities, connecting it to other entities. You supply **FastCAD** with four points, two on the entity to be moved (source entity) and two on the target entity.

To ***connect*** entities, select **[Edit > Connect]:**

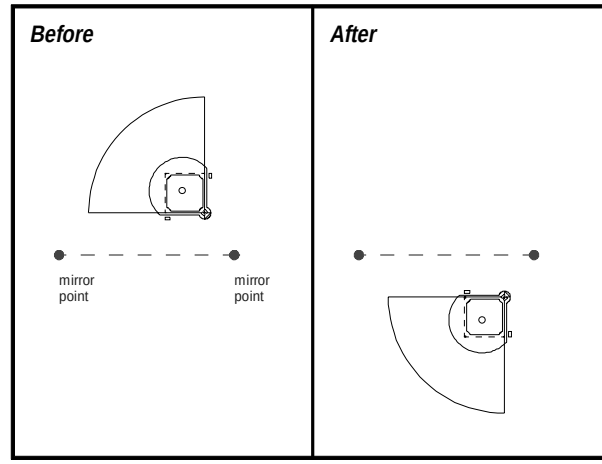
1. Select entities to ***connect***. Select only the entity or entities that should be changed to make the connection. For example, to move an ***arc so it connects to two lines***, which stay in place, select the ***arc***, not the lines.
2. The prompt reads "**Connect from 1st point:**". Pick a point on the source entity.
3. The prompt reads "**Connect it to point:**". Pick the first connection point on the target entity.
4. The prompt reads "**Connect from 2nd point:**" Pick the second connection point on the source entity.
5. The prompt reads "**Connect it to point:**". Pick the second connection point on the target entity.
6. The prompt reads "**Align meridian point [2d plane]:**". **FastCAD** is asking you how to spin the source entity after connecting it to the target entity.
   - Right-click or press **ENTER** to accept the **2D** plane.

     FastCAD calculates the changes needed to fit the "**connect from**" points to the "**connect to**" points and redraw the entity as desired. The command ends.
   - Pick a point on the axis. You'll get best results if you place the axis point farther away from you than the center of rotation.

FastCAD calculates the changes needed to fit the "**connect from**" points to the "**connect to**" points and redraw the entity as desired.

Text equivalent: ***CONNECT***

Mirror



Mirror icon

MIRROR transforms an entity into a ***mirror image*** of itself, flipped over a mirror line that you specify.

The ***mirror line*** you specify is never actually drawn.

To ***mirror*** entities, select **[Edit > Mirror]**:

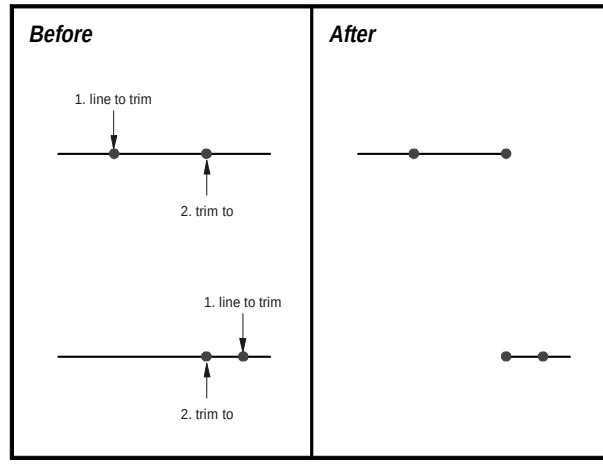
1. Select entities to ***mirror***.
2. The prompt reads "**Mirror line 1st point:**". Pick the mirror line's starting point in the drawing window, or type numeric coordinates.
3. The prompt reads "**Mirror line 2nd point:**". Pick the mirror line's ending point in the drawing window, or type numeric coordinates. The mirror line can be drawn at any angle.

   Hint: To lock the 2nd point precisely ***vertical*** or ***horizontal*** to the 1st point, use the **ORTHO** button.

FastCAD flips the entities and terminates the command.

Text equivalent: ***MIRROR***

Trims > Trim



Trim icon

TRIMS > **TRIM** trims or extends an entity to a ***new endpoin***t.

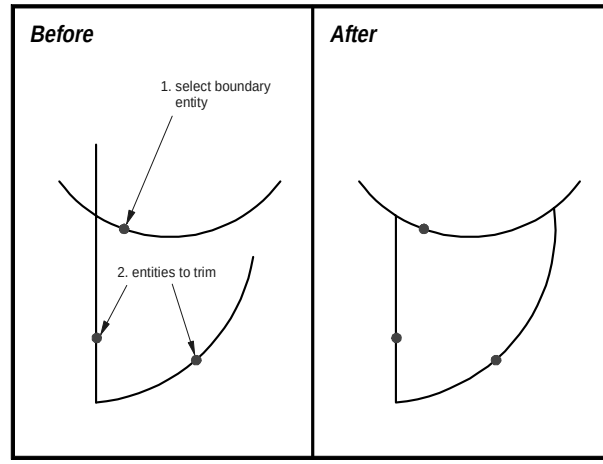
To ***trim*** an entity, select **[Edit** > **Trims** > **Trim]**:

1. The prompt reads "**Entity to trim:**". Pick the entity you wish to trim, selecting a point on the end of the entity you want to keep.
2. Now the prompt reads "**Trim to point [pick point]:**". You can:
   - Pick a new endpoint for the entity. **FastCAD** trims or extends the entity to the "**Trim to**" point.
   - Right-click or press ENTER. The point you used to select the entity will be used as the "**Trim to**" point. **FastCAD** erases the shorter entity segment.

If snap locking is on when you use **TRIM**, **FastCAD** ***trims*** the entity to the point closest to the nearest snap point. As always, the use of modifiers or the attach mode overrides snap locking.

Text equivalent: ***TRIM***

Trims > Trim to Entity



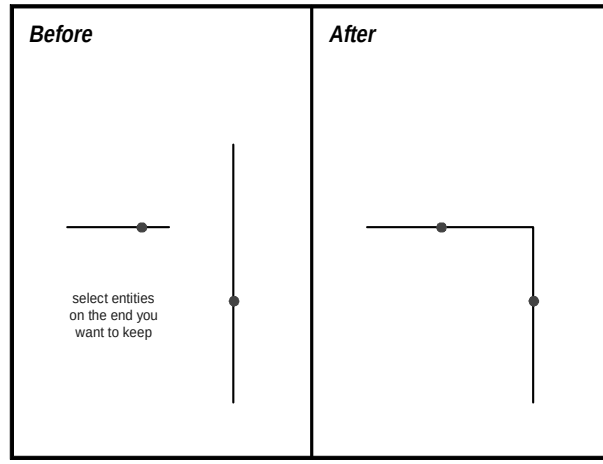
Trim to icon

TRIMS > **TRIM TO ENTITY** ***trims*** or extends selected entities to terminate cleanly at their intersections with a selected boundary entity.

1. The prompt reads "**Select entity to trim to:**". Pick a point on the boundary entity to which you will trim other entities.
2. The prompt reads "**Select entity to trim:**". The entity (or entities) you select will be trimmed to the previously chosen entity. **FastCAD** allows you to continue to select entities to trim as needed. Right-click to end the command.

Text equivalent: ***TRIMTO***

Trims > Trim to Intersection



Trim to Intersection icon

TRIMS > TRIM TO INTERSECTION ***trims*** or extends two entities to a new endpoint at their intersection. This command is very useful for creating precise vertices with two non-parallel entities. Use this command instead of creating zero-radius fillets or zero-length chamfers.

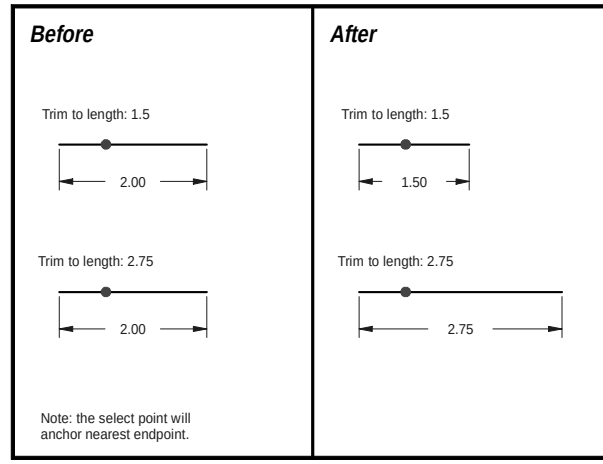
To ***trim to entities to an intersection***, select **[Edit > Trims > Trim to intersection]**:

1. The prompt reads "**1st entity to trim:**". Pick a point on the first entity you want to trim or extend.
2. The prompt reads "**2nd entity to trim:**". Pick a point on a second entity.

Both entities will be redrawn, sharing a common endpoint. **FastCAD** terminates the command.

Text equivalent: ***TRIMINT***

Trims > Trim to Length



TRIMS > TRIM TO LENGTH trims or extends an entity to a pre-defined length.

To ***trim an entity to length***, select **[Edit > Trims > Trim to length]**:

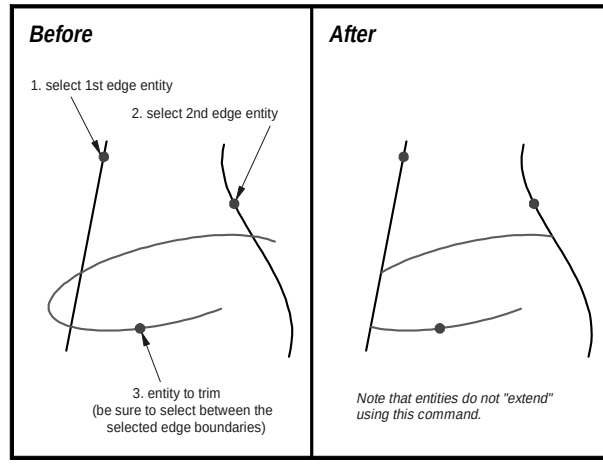
1. The prompt reads "**Trim to length [n]:**". Enter a ***new length*** or accept the default or prior value with a right-click.
2. The prompt reads "**Entity to trim:**". Select each entity near the end you wish to keep or anchor.

The entity you select will be trimmed to the pre-defined length. **FastCAD** stays in the Trim command, allowing you to select additional entities to trim to the specified length as needed. Right-click to end the command.

This command will ***trim*** an ***arc*** to its actual "**arc length**".

Text equivalent: ***TRIML***

Trims > Trim Inside Entities



TRIMS > TRIM INSIDE ENTITIES will ***trim an entity at its intersection with two selected bounding entities***, ***keeping the portion inside the boundaries***. This command can optionally be used with only one bounding entity (trimming a line within a circle, for instance).

To use ***Trim Inside Entities***, select **[Edit > Trims > Trim inside entities]:**

1. The prompt reads "**Trim Edge Entity:**". Pick a point on the ***first boundary entity***.
2. The prompt reads "**2nd Trim Edge Entity [optional]:**". Pick the ***second boundary entity***. The second entity is ***optional***. Right-click or press **ENTER** to bypass second edge selection.
3. The prompt reads "**Entity to trim:**". Pick the entity you wish to ***trim***. Be sure to ***select the entity on a point between the two boundaries***. **FastCAD** trims the selected entity.

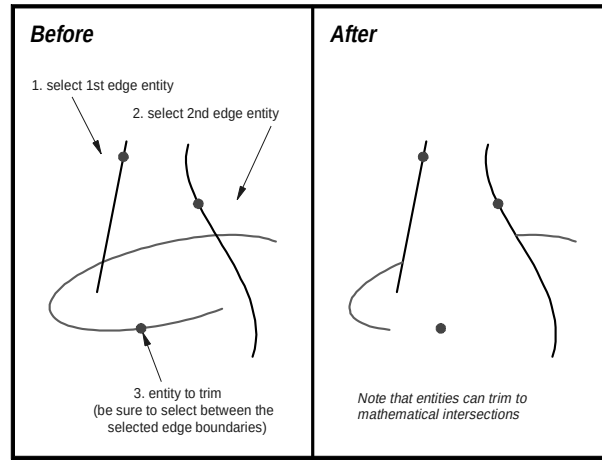
You can continue selecting additional entities to ***trim***, as long as they are appropriately bounded. Right-click to end the command.

NOTE: This command does not "**extend**" entities like most other trim commands do. If you wish to lengthen a line to a boundary entity, use the **TRIMTO** command **[Edit > Trims > Trim to entity]** instead.

Tip: Remember to select the "**Entity to trim**" somewhere on the segment you wish to "**keep**". Using this logic, you can use this command in lieu of **TRIM OUTSIDE ENTITIES.**

Text equivalent: ***TRIMIE***

Trims > Trim Outside Entities



TRIMS > TRIM OUTSIDE ENTITIES will break an entity at its intersection with two selected bounding entities, discarding the portion inside the boundaries. This command can optionally be used with only one bounding entity (breaking a line that crosses a circle, for instance).

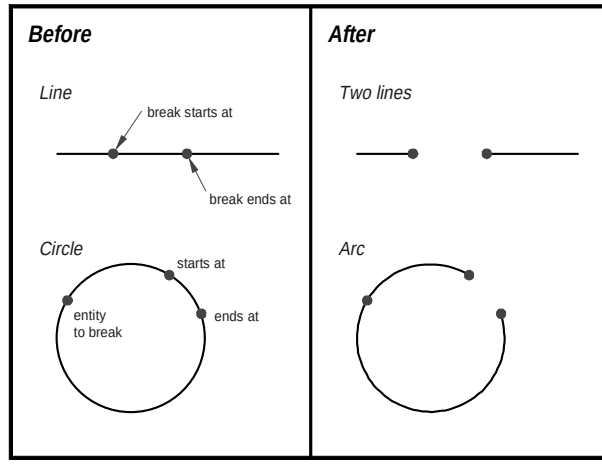
To use ***Trim Outside Entities***, select **[Edit > Trims > Trim outside entities]**:

1. The prompt reads "**Trim Edge Entity:**". Pick a point on the ***first boundary entity.***
2. The prompt reads "**2nd Trim Edge Entity [optional]:**". Select the ***second boundary entity***. The second entity is ***optional***. Right-click or press **ENTER** to bypass second edge selection.
3. The prompt reads "**Entity to trim:**". Select the entity you wish to ***trim*** or extend. Be sure to ***select the entity on a point between the two boundaries***. **FastCAD** breaks the selected entity, creating two new entities.

   You can continue selecting additional entities to ***trim***, as long as they are appropriately bounded. Right-click to end the command.

Text equivalent: ***TRIMOE***

Break

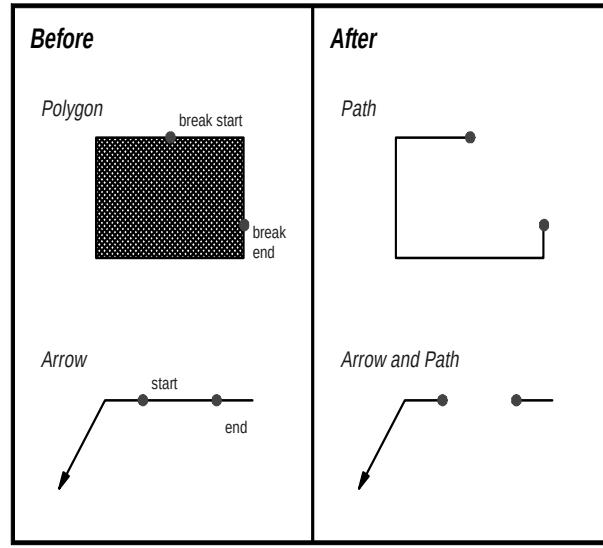



Break icon

BREAK makes ***gaps*** in all entities except points, text, and dimensions. If you **BREAK** a line, arc, path, or spline, **FastCAD** creates two entities where there was one. **BREAK** turns a circle into an arc, a polygon into a path, or a smooth polygon into a spline. If you **BREAK** an arrow or double arrow, **FastCAD** doesn't add arrowheads—it replaces an arrow with an arrow and a path, or a double arrow with two single arrows.

1. The prompt reads "**Entity to break:**". Select the entity you wish to ***break***.
2. The prompt reads "**Break starts at point:**". Pick the s***tart point for the gap***. If you do not select a point directly on the entity, the point will be projected perpendicularly to the entity (or the entity's tangent).
3. The prompt reads "**Break ends at point:**". Pick the ***end point for the gap***.

   FastCAD ***breaks*** the entity and removes the portion selected. The command terminates.



Break Tips

- If the ***break*** start and end points are the same, **FastCAD** creates two entities, even though you can't see the place where they join (as with the **SPLIT** command). If you ***break*** a ***line*** or ***arc*** at one of its ***endpoints***, **FastCAD** creates two entities (**one is zero length**). To ***shorten*** or ***extend*** an ***arc*** or ***line***, use the **TRIM** command instead.
- When you ***break*** a ***polygon*** or ***smooth polygon***, **FastCAD** always keeps node #1. When you ***break*** a ***circle***, **FastCAD** ***keeps the portion of the circle you selected at the*** "**entity to break**:" prompt.
- If ***snap locking is turned on when you use*** **BREAK**, **FastCAD** ***breaks the entity where it passes nearest to a snap point***. You can use ***modifiers*** like **ON**, **% ALONG** or **DEGREE** to ***break*** at specific points on the entity; or use the ***SNAP*** *button* on the command bar.

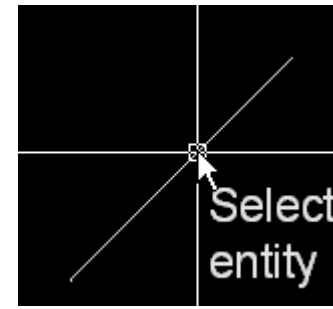
Text equivalent: ***BREAK***

Bend

Thie **BEND** command allows you to select a **LINE** entity and move the mouse cursor to a desired location and left-click to create **TWO LINES** that start at the original **ENDPOINTS** of the selected **LINE** and intersecting at this new location.
Select the **[Edit > Bend]** command.

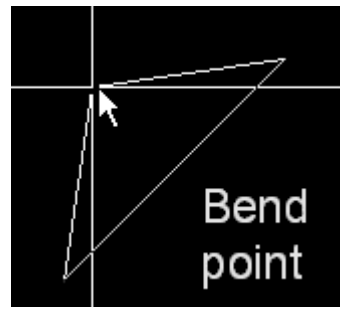
1. The prompt reads:

Select a LINE with a left-click.



2. The prompt now reads:

A dynamic cursor will appear and as you move across the drawing you will see the image of the two new **LINES** that will be created when you left-click to place the intersecting point.



NOTE: The BEND command only works on LINE entities.

Text equivalent: ***BEND***

Split

SPLIT ***divides*** an entity into two entities of the same type at the point you select.

1. The prompt reads "**Entity to split:**". Select the entity you wish to ***divide***.
2. The prompt reads "**Split at point:**". Pick the ***split point***. If you do not pick a point directly on the entity, the point will be projected perpendicularly to the entity (or the entity's tangent). Use ***modifiers*** such as ***endpoint*** for precision.

FastCAD creates two entities, even though you can't see the place where they join, and terminates the command.

Text equivalent: ***SPLITNode Edit > Insert***

The **NODE EDIT> INSERT** commands allow you to ***insert nodes*** in entities that have ***nodes*** as ***control points***, such as ***paths***, ***polygons***, and ***splines***.

For a more intuitive approach to the same function, try using the **DYNAMIC EDIT** command **[Edit > Dynamic Edit]**.

The prompt reads "**Insert node at:**". Pick a point on the entity you wish to modify. Then pick a point for the ***new node***. The endpoints of the original entity stay in place, and **FastCAD** draws two new segments from the endpoints to the ***new point***.

Text equivalent: INSNODE

Node Edit > Delete

The **NODE EDIT> DELETE** commands allow you to ***delete nodes*** in entities that ***have nodes*** as ***control points***, such as ***paths***, ***polygons***, and ***splines***.

The prompt reads "**Delete node at:**". Pick a point on the entity near the ***node*** you wish to ***delete***. The ***node*** is ***deleted*** and a new segment drawn between the two previously adjoining nodes. ***Paths***, ***polygons***, and ***splines must have at least two node***s.

Text equivalent: DELNODE

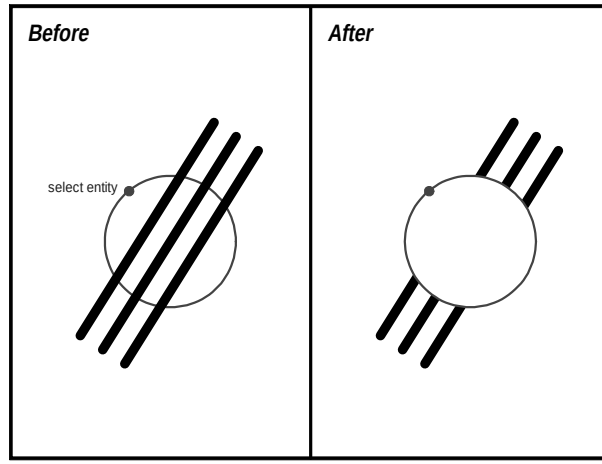
Node Edit > Insert

The **NODE EDIT > INSERT** command allow you to ***insert nodes*** in entities that ***have nodes*** as ***control points***, such as ***paths***, ***polygons***, and ***splines***.

The prompt reads "**Insert node at:**". Pick a point on the entity near the ***node*** where you wish to ***insert a new node***. The ***dynamic cursor*** shows an image of the ***new node*** as you move and a left-click places it. ***Paths***, ***polygons***, and ***splines must have at least two node***s.

Text equivalent: INSNODE

Front



FRONT brings entities to the ***front of your screen***. After you select the entities, **FastCAD** moves them to the end of the drawing file. Because **FastCAD** draws entities in the order that it finds them in the file, entities at the end appear to be on top of or in front of other entities.

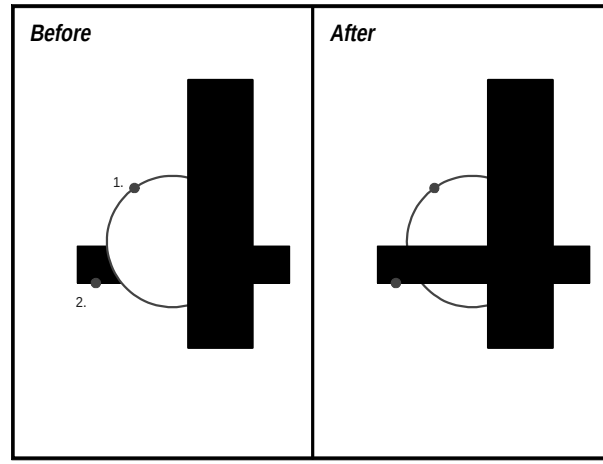
For example, a filled polygon drawn after a line is drawn hides the line on-screen. You could use **FRONT** to select the line to be in front of the polygon.

If the entity style of the polygon uses a scaleable or symbol fill pattern that is transparent and not totally opaque, the line will be seen behind the fill pattern.

Use **UNDO** to reverse the operation.

Text equivalent: ***FRONT***

Above



ABOVE works like the **FRONT** command but is different in that it "**moves**" the selected entities in ***front*** of a specifically selected entity..

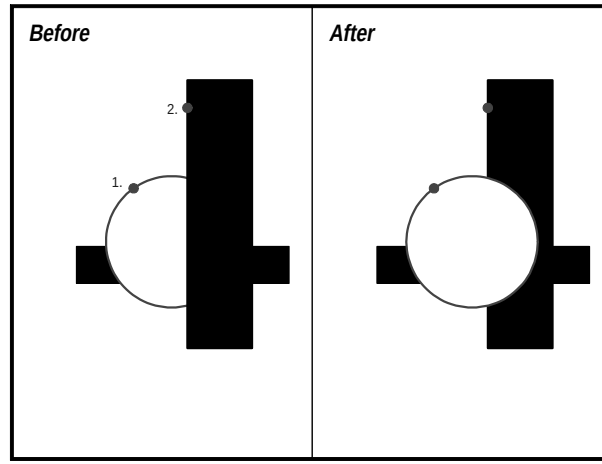
To move an entity ***above*** another, select **[Edit > Above]**:

1. The prompt reads "**Move above entity**:". Pick one entity that the **ABOVEd** entities will move in front of.
2. The prompt reads "**Select entities:**". Use the standard entity selection procedure to select one or more entities to be moved in front of the previously selected entity.

Use **UNDO** to reverse the operation.

Text equivalent: ***ABOVE***

Below



BELOW works like the **BACK** command, drawing selected entities at the ***back of your screen***. **BELOW** is different in that it "**moves**" the selected entities ***behind*** specifically selected entities. **BACK** ***moves the selected entities to draw behind all other entities in the drawing.***

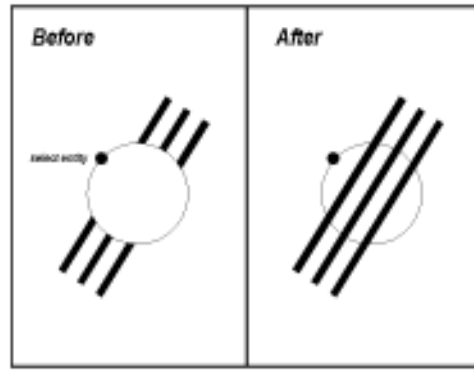
To move an entity ***below*** another, select **[Edit > Below]**:

1. The prompt reads "**Move above entity**:". Pick one entity that the **BELOWed** entities will move behind.
2. The prompt reads "**Select entities**:". Use the standard entity selection procedure to select one or more entities to be moved behind the previously selected entity.

Use **UNDO** to reverse the operation.

Text equivalent: ***BELOW***

Back



BACK moves entities to the **back** of your screen. After you select the entities, **FastCAD** moves them to the front of the drawing file. Because **FastCAD** draws entities in the order that it finds them in the file, entities at the front appear to be in back of or behind other entities.

For example, a filled polygon drawn after a line is drawn hides the line on-screen. You could use **BACK** to select the polygon to be behind the line.

I***f the entity style of the polygon uses a scaleable or symbol fill pattern that is transparent and not totally opaque, the line will be seen behind the fill pattern.***

Use **UNDO** to reverse the operation.

Text equivalent: ***BACK***

Explode

EXPLODE command **[Edit > Explode]** reduces the complexity of entities down to more basic components.

- **Points**, **lines**, **circles**, and **arcs do not reduce.**
- **Ellipses** reduce to a **Polygon** with **one EXPLODE** and a series of **Lines** with a second **EXPLODE**.
- **Elliptical arcs** reduce to a **Path** with **one EXPLODE** and a **series of Lines** with a **second EXPLODE**.
- **Multipolys** and **symbol references** reduce to the **original entities**.
- **Groups** reduce to **original entities** (**all levels**).
- **2D polygons**, **2D paths**, and **2D splines** reduce to a s**eries of lines**.
- **Arrows** reduce to the **base entity** (**path/line/arc**) **without arrow-head** info.
- **Dimensions** reduce to **text**, **lines**, and **polygons** representing the arrowheads.
- **File text** reduces to **multi-line tex**t entities.
- **Multi-line** text reduces to **single line tex**t entities.
- **Walls** reduce to **line** entities.
- **Xrefs** are disassociated from the source drawing; their entities become integral to the host drawing (analogous to an ungrouped part).
- **3D Solids** reduce to **3D polygon** entities.
- **Symbol References reduce** to **original entities after one EXPLODE**. A **second EXPLODE** will reduce **nested Symbols** back to the **original entities**. Using **EXPLODE** on these original entities will reduce them as stated above.

EXPLODE can also be useful for **removing errant blocks**, **symbols**, or **special** entities found after importing some **DXF** files produced by third parties.

WARNING: You should never use the **EXPLODE** command and select entities by **ALL** unless you really mean to do this. **ALL** of the entities in the ***avtive drawing*** will be reduced to the ***basic entities*** described above. If you select **ALL** entities by mistake, use the **UNDO** command immediately after hitting the **DO IT** command. If you have **SAVED** the drawing after using the **EXPLODE** command, **UNDO** will **NOT** work. Do not use the **EXPLODE** command on entities that are **GROUPED**. Use the **UNGROUP** command from the **INSERT** menu or click the **LOCKED** button at the right end of the command line so it reads **UNLOCKED**. This is a temporary unlock for **GROUPS** that will allow you to **EDIT** them. Click the **UNLOCKED** button so it reads **LOCKED** to turn **GROUP** locking back on.

Use **UNDO** to reverse the operation.

Text equivalent: ***EXPLODE***

Make 3d

The **MAKE 3d** command allows you to select any **2d entities** and give them a **Z value of 0**. They may be moved or copied anywhere in 3d space. Without this **Z value** they **are 2d entities and will always remain parallel to the XY plane**. A **2d line** will list **[Info > List]** with only **XY values for its starting point and ending point**. After using the **[Edit > Make 3D]** command on this **2d line** it would list with **XYZ values** for its **starting point** and **ending point**.

Using the **3D LINE** command from the **DRAW3** menu automatically assigns **XYZ values** for the line.

Text equivalent: ***MAKE3D***

Chapter 13

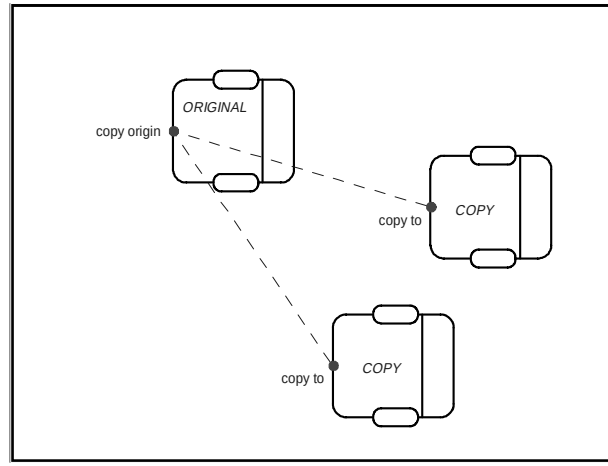
Copy Commands

Duplicate entities in your drawing

Commands in the ***Copy*** *menu* add new entities to your drawing by duplicating existing entities. **FastCAD** marks the entities added as new when you use **Copy** commands. You can edit the copies or make copies of the copies by choosing **Prior** from the ***Entity Selection*** *dialog box*.

- ***COPY*** duplicates existing entities
- ***DRAG COPY*** duplicates entities using dynamic cursors
- ***COPY TO LAYER*** duplicates entities, but to different layers.
- ***COPY CONNECT*** works like CONNECT, but leaves the selected entities unchanged.
- ***Scale Cop****y commands* resize copies of existing entities.
- ***Rotate Copy*** *commands* rotate copies of existing entities.
- ***MIRROR COPY*** works like MIRROR, but leaves the selected entities unchanged.
- ***Array*** *commands* create evenly spaced copies in rectangular or circular patterns.

Copy



Copy icon

COPY duplicates entities from one location to another within a drawing. **COPY** does not rotate or scale the entities.

To ***copy*** entities, select **[Copy > Copy]**:

> **Tip:**
>
> You can force the copies to line up horizontally or vertically with the originals by turning on orthogonal locking.
>
> Turn on ortho by depressing the *Ortho button.*

1. Select the entities to ***copy***.
2. The prompt reads "**Copy origin:**". This is a base point that **FastCAD** uses to determine ***copy*** placement. The cursor turns into crosshairs. To specify the ***copy origin***, pick a point or type numeric coordinates. Usually, you would pick a point on the entity, though that is not a requirement.

   FastCAD displays a rubber-band cursor anchored at the ***copy origin***.
3. The prompt reads "**Copy to point:**". To specify the new ***copy*** location, pick a point or type numeric coordinates.

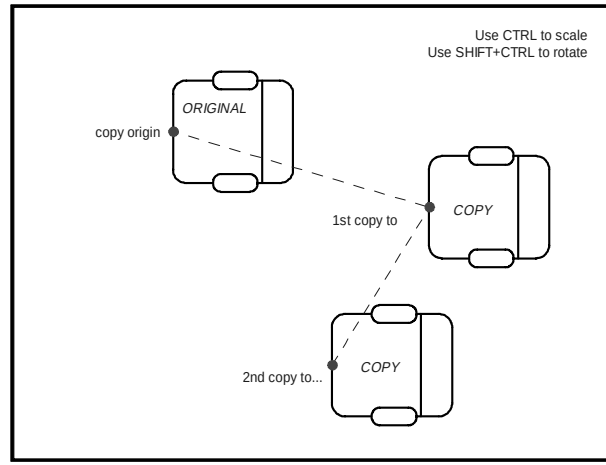
   FastCAD calculates the ***distance*** and **direction** between the points and copies the selected entities accordingly.
4. The prompt still reads "**Copy to point:**". You can make ***multiple copies***, selecting a new location for each, until you right-click to end the command.

Ortho button

You can force the ***copies*** to line up ***horizontally*** or ***vertically*** with the originals by turning on ***orthogonal locking*** by clicking the **ORTHO** *button***.**

Text equivalent: ***COPY***

Drag Copy



DRAG COPY works almost identically to the regular **COPY** command except for two key differences:

1. The **DRAG COPY** command displays a dynamic cursor of the entities that are being ***copied***, exactly like the regular **DRAG** command. This allows you to use the CTRL key to scale, and the SHIFT+CTRL keys to rotate, during placement.
2. For multiple copies from a single source, the "**copy from**" point remains fixed in the regular **COPY** command. With **DRAGCPY**, the "**copy from**" point changes relative to the ***last copy***. This can be helpful when the displacement between each of the copy elements is easier to quantify, rather than the vector between each copy element and the original.

To drag copy entities, select [Copy > Drag Copy]:

1. Select the entities to ***copy***.
2. The prompt reads "**Origin:**". This is a base point that **FastCAD** uses to determine copy placement. The cursor turns into crosshairs. To specify the ***copy origin***, pick a point or type numeric coordinates. Usually, you would pick a point on the entity, though that is not a requirement.

   FastCAD displays a rubber-band cursor anchored at the *copy origin*.
3. The prompt reads "**SHIFT=move CTRL=scale SHIFT+CTRL=rotate:**". To specify the ***new copy*** location, pick a point or type numeric coordinates.

   FastCAD calculates the ***distance*** and ***direction*** between the points and ***copies*** the selected entities accordingly.
4. The prompt still reads "**SHIFT=move CTRL=scale SHIFT+CTRL=rotate:**". You can make ***multiple copies,*** selecting a new location for each, until you right-click to end the command.

Text equivalent: ***DRAGCPY***

Copy to Layer

COPY TO LAYER ***copies*** selected entities to a ***different layer***, leaving the originals in place.

To ***copy entities to a layer***, select **[Copy > Copy to Layer]**:

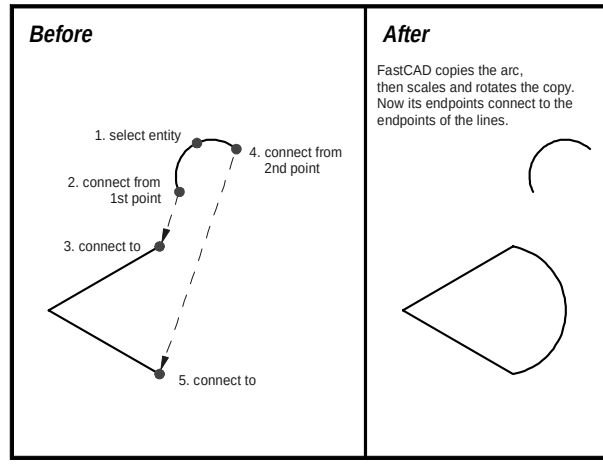
1. Select entities to ***copy***.
2. The prompt reads "**Layer name or # [dialog]:**". **FastCAD** is asking for the ***layer*** to which the selected entities will be copied. To specify a t***arget layer***:
   - Type in an existing layer name or ID # and press **ENTER**.
   - Right-click to display the ***Select Layer*** *dialog*. Pick the ***target layer*** from the list.

FastCAD ***copies*** the entities to the selected ***layer*** and terminates the command. You can verify the command's proper operation with the **LIST** command **[Info > List]**.

To ***move*** the entities to a ***new layer***, use the **[Edit > Change > Layer]** command.

Text equivalent: ***LAYERCPY***

Copy Connect



COPY CONNECT ***copies*** an entity or entities, then ***scales*** and ***rotates*** the ***copy***, ***connecting*** it to other entities and designated points. You supply **FastCAD** with four points, two on the entity to be moved (**source entity**) and two on the target entity.

This command works exactly like the **CONNECT** command **[Edit > Connect]**, except that the entity being connected is not altered. Instead, a copy of that entity is created and transformed.

To ***connect*** entities, select **[Copy > Copy Connect]**:

1. Select entities to ***connect***. Select only the entity or entities that should be copied to make the connection. For example, to copy an arc so it connects to two lines that remain in place, select the arc, not the lines.
2. The prompt reads "**Connect from 1st point:**". Pick a point on the source entity, using modifiers or the **Attach mode** to assist in precision.
3. The prompt reads "**Connect it to point:**". Pick the first connection point on the target entity.
4. The prompt reads "**Connect from 2nd point:**" Pick the second connection point on the source entity.
5. The prompt reads "**Connect it to point:**". Pick the second connection point on the target entity.
6. The prompt reads "**Align meridian point [2d plane]:**". **FastCAD** is asking you how to spin the source entity after connecting it to the target entity.
   - Right-click or press ENTER to accept the ***2*D plane**.

     FastCAD calculates the changes needed to fit the "**connect from**" points to the "**connect to**" points and redraw the entity as desired. The command ends.
   - Pick a point on the axis. You'll get best results if you place the axis point farther away from you than the center of rotation.

FastCAD calculates the changes needed to fit the "**connect from**" points to the "**connect to**" points and redraws a transformed copy of the entity as desired.

Text equivalent: ***CONNECTC***

Dynamic Scale Copy

DYN SCALE COPY visually shrinks or enlarges a copy of the selected entities, using a dynamic cursor.

To ***dynamically scale copies*** of entities, select **[Copy > Dyn Scale Cpy]**:

1. Select entities to ***copy*** and ***scale***.
2. The prompt reads "**Origin**:". The drag from point is the point around which **FastCAD** should scale the entities (the "**scale center**"). To specify the scale center:
   - Pick a point in a drawing window.
   - Type numeric coordinates and press **ENTER**.
3. The prompt reads "**SHIFT=move CTRL=scale SHIFT+CTRL=rotate:**". Move the mouse on the screen to visually resize the selected entities. They will shrink or grow depending on the direction of mouse movement. When you are satisfied with the size of the entities, left-click to create the scaled copies. The original selected entities remain unchanged.

   FastCAD redraws the entities at the new scale. The ***scale center*** remains fixed.
4. The prompt continues to read "**SHIFT=move CTRL=scale SHIFT+CTRL=rotate:**". You can continue to create new scaled copies, or you can right-click to end the command

The "**SHIFT=move CTRL=scale SHIFT+CTRL=rotate**" prompt tells you that dynamic cursors are in effect. While creating the scaled copies, you can also move the scale center or rotate the copied entities. To move the scale center, hold down the **SHIFT** key while moving the mouse. To rotate about the scale center, hold down the **SHIFT+CTRL** keys while moving the mouse.

Text equivalent: ***DYNSCLC***

Scaled Copies

SCALED COPIES shrinks or enlarges ***copies*** of selected entities, based on a ***scale*** value you enter.

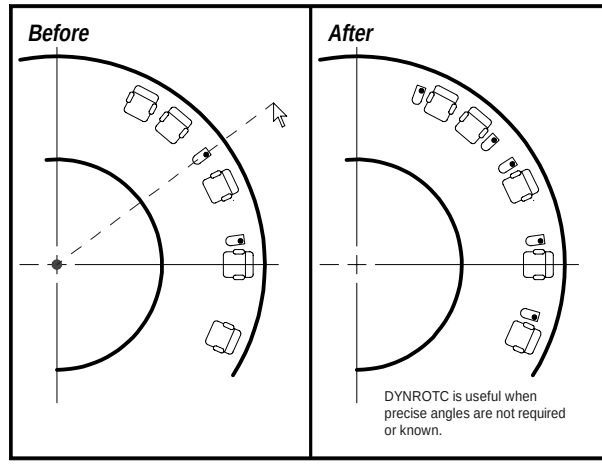
To ***scale copy*** entities, select **[Copy > Scaled Copies]**:

1. Select entities to ***scale***.
2. The prompt reads "**Scale by factor [n]:**". If you wish to accept the default, right-click or press ENTER. To enter a new scaling factor, type a scaling and press ENTER. Scaling factors **> 1** will make entities larger. Scaling factors **< 1** will make entities smaller. For example, to make the selected entities twice as large as they are now, type "**2**" and press ENTER. To make them half as large, type ".**5**" and press ENTER.
3. The prompt reads "**Origin point [x,y,z]:**". The ***scale center*** is the point around which **FastCAD** should scale the entities. To specify the scale center:
   - Right-click or press **ENTER** to accept the default or ***prior center.***
   - Pick a point in a drawing window.
   - Type numeric coordinates and press **ENTER**.

FastCAD draws new copies of the entities at the ***new scale***. The ***scale center*** remains fixed.

Text equivalent: ***SCLCPY***

Dynamic Rotate Copy



DYN ROTATE CPY works like **DYNAMIC EDIT**, except that, instead of rotating the selected entities, it makes rotated copies of the entities, leaving the originals in place.

To ***dynamically rotate copies*** of entities, select **[Copy > Dyn Rotate Cpy]**:

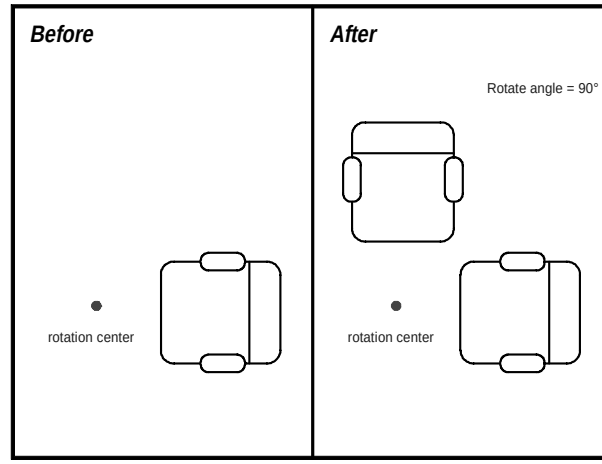
1. Select entities to ***copy*** and ***rotate***.
2. The prompt reads "**Origin:**". The drag from point is the point around which **FastCAD** should turn the entities. To specify the rotation center:
   - Pick a point in a drawing window.
   - Type numeric coordinates and press **ENTER**.
3. The prompt reads "**SHIFT=move CTRL=scale SHIFT+CTRL=rotate:**". Move the mouse on the screen to visually rotate the selected entities. They will rotate to the bearing defined by the cursor position to the rotation center. When you are satisfied with the rotation of the entities, left-click to create the rotated copies. The original selected entities remain unchanged.

   FastCAD creates ***copies*** and ***rotates*** them around the center, which remains fixed.
4. The prompt continues to read "**SHIFT=move CTRL=scale SHIFT+CTRL=rotate:**". You can continue creating new rotated copies, or you can right-click to end the command.

The "**SHIFT=move CTRL=scale SHIFT+CTRL=rotate**" prompt tells you that dynamic cursors are in effect. While creating the rotated copies, you can also move the rotation point or scale the copied entities. To move the rotation point, hold down the **SHIFT** key while moving the mouse. To *scale* hold the **CTRL** key while moving the mouse.

Text equivalent: ***DYNROTC***

Rotated Copies



ROTATED COPIES works like **ROTATE**, except that, instead of rotating the selected entities, it makes rotated copies of the entities, leaving the originals in place.

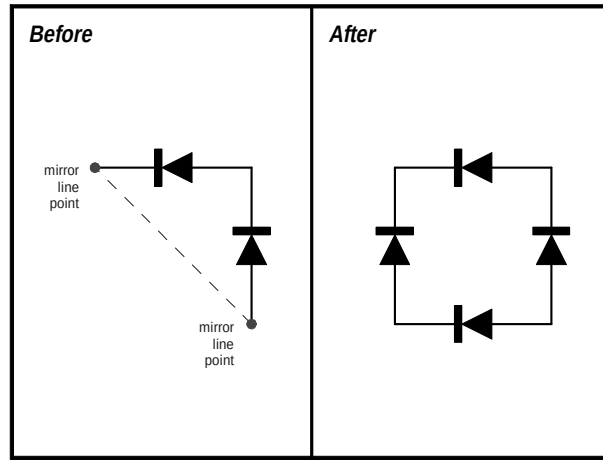
To place ***rotated copies*** of entities, select **[Copy > Rotated Copies]**:

1. Select entities to ***rotate***.
2. The prompt reads "**Rotate by Angle [n]:**", with the last angle you used suggested as a default or prior value. (The **default is 90°** when you start a new work session.) You can:
   - Right-click or press **ENTER** to accept the default or prior rotation.
   - Type a positive or negative angle, which can include values like **90.0005°** or **22+1/2°**. (Remember that positive angles turn counterclockwise.) For example, to turn the entities **45°** clockwise, type **"45"** and press **ENTER**.
3. The prompt reads "**Origin point [x,y,z]:**". This is the point which FastCAD should turn the entities. To specify the rotation center:
   - Right-click or press **ENTER** to accept the default or prior center.
   - Pick a point in a drawing window.
   - Type numeric coordinates and press **ENTER**.
4. The prompt reads **"Rotate Axis Point [XY plane]:". *Right-click to accept the*** XY plane.

FastCAD creates copies of the selected entities and rotates them around the center, which remains fixed.

Text equivalent: ***ROTCPY***

Mirrored Copies



Mirrored Copies icon

MIRRORED COPIES works like **MIRROR**, except that instead of flipping the selected entities, it makes ***mirror-image copies*** of them, leaving the originals in place. This is useful for drawing objects that are symmetrical about any axis. You can draw one side of the object, then use **MIRCPY** to create the other side.

The mirror line you specify is never actually drawn.

To ***mirrored copies*** of entities, select **[Copy > Mirrored Copies]**:

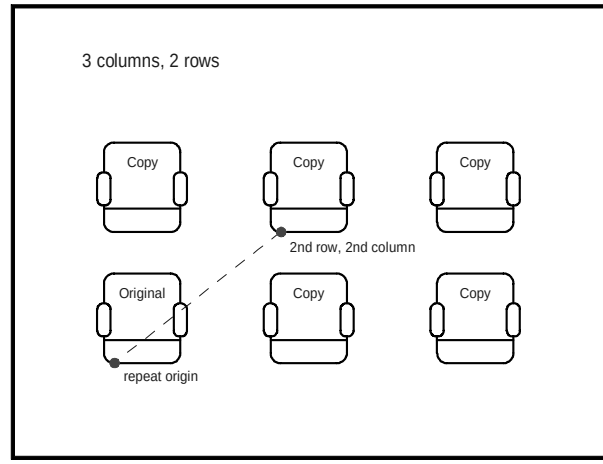
1. Select entities to ***mirror***.
2. The prompt reads "**Mirror line 1st point:**". Pick the mirror line's starting point in the drawing window, or type numeric coordinates.
3. The prompt reads "**Mirror line 2nd point:**". Pick the mirror line's ending point in the drawing window, or type numeric coordinates. The mirror line can be drawn at any angle.

   Hint: To lock the 2nd point precisely ***vertical*** or ***horizontal*** to the 1st point, use the **Ortho** *button*.

FastCAD draws ***new copies*** of the selected entities and ***flips*** them over the ***mirror line***. The command terminates.

Text equivalent: ***MIRCPY***

Rectangular Array



Tip:

You can force the mirror line to be perfectly horizontal or vertical turning on orthogonal locking. Turn on ortho by depressing the *Ortho button.*

This way, the second mirror line point can be chosen semi-aribtrarily (you just need to establish direction). This can often be faster than having to carefully choose both mirror line points.

Ortho button

RECTANGULAR ARRAY makes a ***rectangular array*** (regularly spaced ***rows*** and ***columns***) of copies. Build ***rectangular arrays*** using entities in your drawing or parts and symbols inserted from another file.

To build a ***rectangular array***, select **[Copy > Rectangular Array]**:

1. Select entities to ***array***.
2. The prompt reads "**# of columns [n]:**". **FastCAD** is asking for the number of ***horizontal copies*** to draw. Type the total number of objects in each ***row***, including the original. Right-click to accept the default or ***prior value***.
3. The next prompt reads "**# of rows [n]:**". **FastCAD** is asking for the number of ***vertical copies*** to draw. Type the total number of objects in each ***column***, including the original. Right-click to accept the default or ***prior value***.
4. The prompt reads "**Origin:**". This is a base point that **FastCAD** uses to determine spacing between copies. Pick a point on the object to be copied.

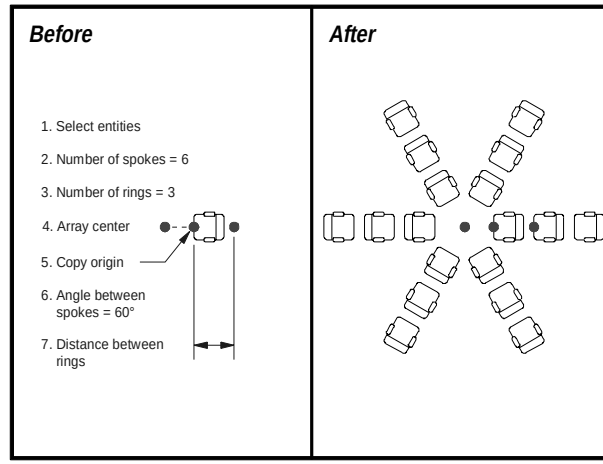
   FastCAD displays a rubber-band cursor, anchored at the "**Repeat origin**".
5. The prompt reads "**2nd col, 2nd row point:**". This offset from the ***origin point*** tells **FastCAD** both the ***horizontal*** and ***vertical*** distance you want between parts in the ***array***. You can use the rubber-band cursor to help you judge the spacing. Pick the point in a drawing window ***diagonally*** from the ***origin point***.

FastCAD draws the ***array*** and terminates the command.

NOTE: If you enter **quantity = 1** for the number of ***columns*** and ***rows***, the array command will ***cancel*** itself rather than (l**ikely inadvertently**) copy the selected entities on top of themselves.

Text equivalent: ***REPEAT***

Circular Array



CIRCULAR ARRAY creates ***multiple copies*** arranged in ***rings***. Build ***circular arrays*** using entities in your drawing or ***parts*** and ***symbols*** inserted from another file.

To build a ***circular array***, select **[Copy > Circular Array]**:

1. Select entities to ***array***.
2. The prompt reads "**# spokes [n]:**". **FastCAD** is asking for the number of copies to draw per revolution. Type the ***total number of spokes***, including the original. Right-click or press **ENTER** to accept the default or prior value.
3. The next prompt reads "**# rings [n]:**". **FastCAD** is asking for the ***number of concentric rings*** of copies to draw. Type the ***total number of rings***, including the original. Right-click or press **ENTER** to accept the default or ***prior value***.
4. The prompt reads "**Center point [x,y]:**". This is center point of the ***array***. To specify a ***center***:
   - Right-click or press **ENTER** to accept the default or ***prior center***.
   - Pick a point in a drawing window.
   - Type numeric coordinates and press **ENTER**.

   FastCAD displays a rubber-band cursor, anchored at the ***center point.***
5. The prompt reads "**Copy origin [center]:**". This ***origin point*** determines spacing between copies. Pick a point on the entity or entities to be copied.
6. The prompt reads "**Angle between spokes [n]:**". The default value spaces the spokes evenly to make a ***full circle***. Right-click or press ENTER to accept the default, or ***type in a different angle value*** if desired.

   If you did not specify more than one ring, **FastCAD** draws the ***array*** and terminates the command.

7. If you asked for more than one ring, the prompt reads "**Distance between rings:**" This is the distance from the ***origin point of the original to the corresponding point on the first copy in the next ring.*** Enter a measurement.

FastCAD draws the ***array*** and terminates the command.

Text equivalent: ***CARY***

Chapter 14

Insert Commands

Working with Symbols and Parts

One of the great advantages of **CAD** over pen & paper is its ability to readily duplicate drawings or components. After you've drawn something once, you can use it over and over again, even in other drawings. **FastCAD** uses two methods to accomplish this task: ***symbols*** and ***parts***.

Commands in the **INSERT** menu are used to manipulate ***symbols*** and ***parts*** (which are simply any **FastCAD** drawing). In the most general sense, ***symbols*** and ***parts*** are **FastCAD** drawings that are made "**portable**". Though they share similar usage traits, they differ in a number of important ways. Consider these differences when determining which method is best for a particular purpose.

Symbols and Parts Defined

Symbols are collections of entities that are stored internally in each drawing, hidden from view. These unseen, internal representations are called ***Symbol definitions***. In order to see a ***symbol*** on the drawing, you must draw a ***symbol reference.*** Each ***reference*** accesses a ***definition*** and ***cannot exist without that definition***. Because many ***references*** can point to the *same* ***definition***, ***symbol references*** can help keep your drawing small.

Symbols also posses an optional property called ***attributes***. ***Attributes*** are pieces of information that are attached to each instance of a ***symbol reference***. These are helpful when extracting drawing information for use in a database, bill of materials, etc.

Parts **also contain a collection of entities, but they are stored as** ***separate drawing files***. When you ***insert*** **a** ***part*** into your drawing, you are adding the ***entire part drawing file*** into your existing drawing—including any custom ***layers***, ***line styles***, ***fill styles***, etc., contained therein. When you ***insert*** **a** ***part***, you also import ***all symbol definitions*** in that ***part***.

FastCAD's Symbol Catalog Viewer

The ***Symbol Catalog*** *viewer* greatly enhances ***symbol*** usage.

Manage your ***symbols*** by using **FS7** ***symbol catalog files***. Entire sets of ***symbols*** can be loaded into your drawing for instant accessibility. **FS7** files can be created and customized like any standard **FastCAD** drawing. The **LOAD PART AS SYMDEF** command is available to ***convert entire folders of parts into a single*** **FS7** ***catalog file***.

Symbol insertion with the ***Symbol Catalog viewer*** is a drag-and-drop type operation. Use the thumbnail viewer to browse the catalog, then select, and insert. You can expand the single column thumbnail display into a full-page spread by right-clicking directly on the viewer.

Choosing Symbols or Parts Method

While you can ***scale*** or ***rotate*** both ***symbols*** and ***parts*** upon insertion, there are some distinctions that make one or the other a more appropriate choice for your current drawing.

If you are placing numerous copies of identical entities, symbols are your best choice. When you ***edit*** a ***symbol definition***, ***all symbol references in your drawing update accordingly.*** **FastCAD** also allows dynamic placement of ***symbols***, meaning that you can see a ghost cursor of the ***symbol*** before you actually insert it. ***Symbols*** can possess ***attributes***. ***Symbols*** can be efficiently and logically consolidated into ***catalog files for use with the Symbol Catalog viewer***.

Parts are more useful in situations where the source will be inserted into a drawing once, such as a drawing border. Also, use ***parts*** if you intend to ***edit*** the inserted entities. When ***parts*** are inserted, they are completely disassociated from the source.

Xrefs

External References files [**XREF**] are another method of combining different drawing files. Each **XREF** points to a separate drawing file, which then appears as part of the host drawing. There is no limit to the number of xrefs contained in a drawing.

When the **INSERT XREF** command is used to create a combined drawing file, the **XREF** file is actually inserted as a ***symbol reference***. In essence, an **XREF** is a ***symbol definition stored in another file***. The host drawing simply contains a ***reference*** that contains the **XREF** file's ***drive***, ***path***, and ***file name***. As long as that information is valid, The **XREF** files will appear automatically when the host file is loaded.

When a drawing exists as an **XREF**, ***it cannot be modified***. However, the xrefed file can be loaded as a drawing file and edited. Those changes are reflected wherever that drawing is xrefed.

Exploding an Xref

An **XREF** ***can be made a perminate part of the host drawing*** by using the **ECPLODE** command **[Edit > Explode]**. This works exactly like ***exploding a symbol reference***. The ghost image of the ***xfer*** file now become real entities in the host drawing and may be edited. The link between the ***xref*** file is broken and changes made to it will ***not*** be reflected in the host drawing.

Part commands

A ***part*** is any **FastCAD** drawing that you insert into another drawing. If your work uses repeated elements like furnishings, electronic components, or standard details, save each element as an individual drawing file. The **PART** commands let you use such elements over and over without ever having to construct them again.

Frequently used ***symbols*** can be imported into drawings via the ***parts*** method.

To *create* a *part* file:

- Load or create a **FastCAD** drawing in the usual way;
- Use the **CREATE PART** command **[Insert > Create Part]** to save selected entities from an existing drawing as a separate file.

When **FastCAD** inserts one file into another as a *part*, it puts the inserted file's ***origin*** or (**0,0**) coordinate on the "**insert at/scale factor [INS=specs]:**" point. When you start a new drawing, the (**0,0**) point is below and to the left of the screen. When you create a drawing to use as a ***part***, use **CHANGE ORIGIN** command **[View > Change Origin]** to set the (**0,0**) point to a logical point in the ***part***—the center or lower-left corner is usually a good choice.

Normally, entities that you insert as ***parts*** keep the **layer assignments** that they had in the original ***part*** file (so you can have multi-layer parts). If you want **FastCAD** to insert the ***part*** on the ***current work layer*** instead, draw the entities in the ***part*** file on the **MERGE layer**. If you forget to set up a ***part*** this way, it is easy to move the newly inserted ***part*** to the ***current laye***r—just select **[Edit > Change > Layer]**, select by **Prior**, then choose **Do it**. Right-click to display *the **Layer Management** dialog box*, then choose **OK** to accept the ***current layer***.

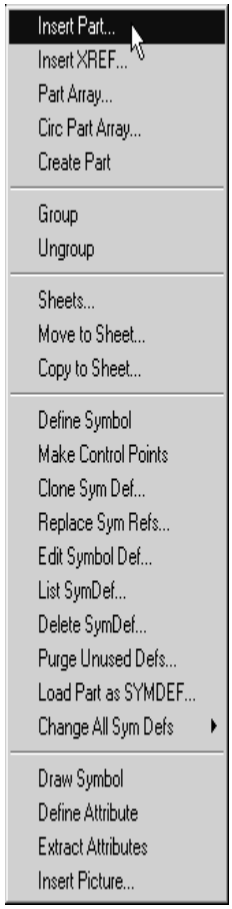
Commands for inserting parts:

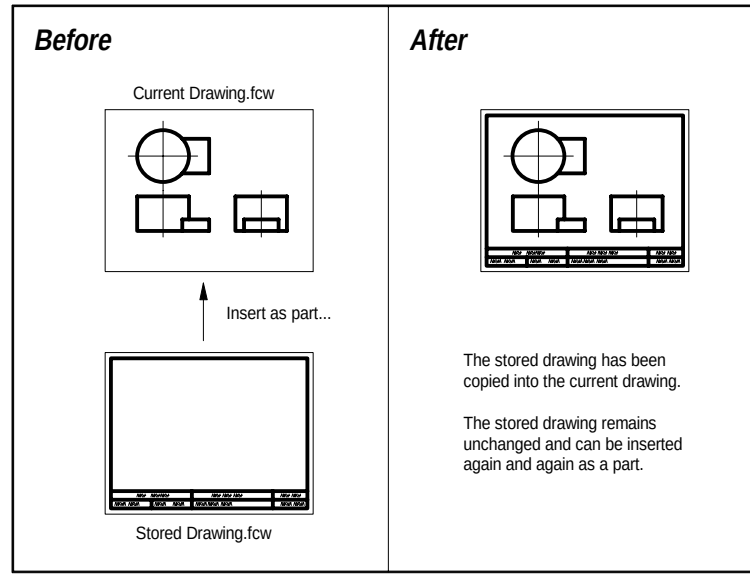
- **INSERT PART** adds one part at a time;
- **PART ARRAY** builds a rectangular array (**rows and columns**);
- **CIRC PART ARRAY** builds a circular array (**spokes and rings**).

Each time you insert a ***part***, you can ***scale*** and ***rotate*** it as needed. If units in the ***part*** file are different from the units in the current drawing, **FastCAD** does the conversion for you. For example, if you insert a 48" desk with units defined as Inches into a floor plan with units defined as Feet, the desk is 4 units long in the floor plan.

FastCAD inserts each ***part*** as a **GROUP** as it is added to your drawing, treating the ***entire part as a single entity***. To modify individual entities within inserted ***parts***, you can ***unlock group effects*** by clicking the ***Locked*** *button* so it reads ***Unlocked***, perform entity **edits**, and then ***restore group effects*** by clicking the ***Unlocked*** *button* so it reads ***Locked*** again. Use the **UNGROUP** command to permanently disassociate a grouped *part*.

Insert Part





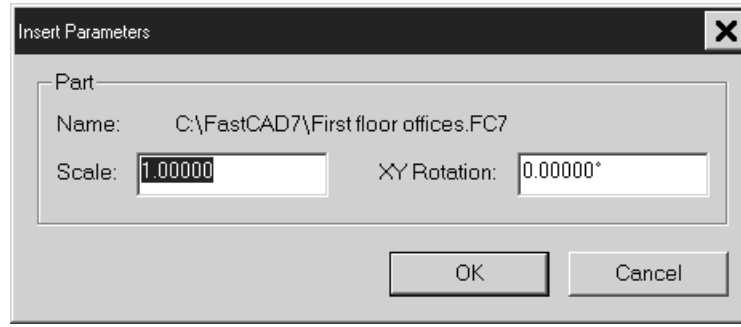
INSERT PART adds any other **FastCAD** drawing, as a ***part***, to the working drawing. One part at a time is inserted.

1. **FastCAD** displays the ***Part to Insert*** *dialog box*. This is the standard ***Load Drawing*** *dialog box*. Note that **FastCAD** displays the last inserted ***part*** as the default. To select a ***part* to *insert*:**
   - Double-click a filename;
   - Select a filename using the cursor and choose **Open**;
   - Type a drawing filename from the list of available drawings.

2. ***The prompt reads*** Insert part at/scale factor [INS=specs]:

   You should see a ***ghost image*** of the ***part*** attached to the moving cursor. If you want to change the ***scale*** of the original you may immediately t***ype a new scale factor*** such as "**2**" to make the ***part*** twice as large. If you press the "I**ns**" key on the keypad **FastCAD** displays the ***Insert Parameters*** *dialog box* with the ***default Scale factor*** and ***XY Rotation Angle*** of the ***part***. You may adjust both ***part scaling*** and ***rotation*** from this dialog box.. The ***name*** of the ***part drawing*** is also displayed in the dialog box.

- The ***default scaling is 1.0*** (**same size as the original part**). Type in a ***new scale factor*** if you wish to insert the ***part*** at a ***larger*** or ***smaller*** size.
- The ***default XY rotation angle is 0.0°.*** If you wish to **rotate** the ***part*** upon insertion, ***type in a new rotation between 0 degrees and 360 degrees***. For example, type "**45**" to ***rotate the part 45 degrees counter-clockwise***.



Insert Parameters dialog box

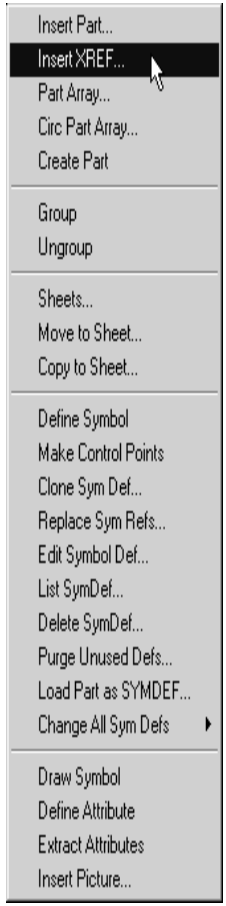
3. Choose the **OK** button to accept the ***scaling*** and ***rotation values***.
4. ***The prompt reads*** Insert part at/scale factor [INS=specs]:

   Enter an insertion point by using the crosshairs, or by typing numeric coordinates.

   FastCAD draws the contents of the ***part*** file at the ***insertion point***.
5. The prompt continues to read "**Insert part at/scale factor [INS=specs]:**". Continue inserting copies of the ***part*** if you wish. Right-click to end the command.

Text equivalent: ***PART***
Macro equivalent (bypass dialog box): ***PARTM***

Insert XREF



External references (**XREFs**) provide another method of working with multiple drawing files.

An* XREF *placed within a host drawing "**points**" ***to another physical drawing file***. ***The* XREF *then appears in the host drawing***. Because the **XREF** ***pointer*** is embedded in the ***host drawing,*** the **XREF** files automatically load with the ***host drawing***.

The easiest way to understand the benefits and limitations of **XREF**s is by thinking of them as how they are implemented: A visible **XREF** is like a ***symbol reference***, where the ***definition is an external file***. An **XREF** file itself is a valid and editable drawing, but ***it exists in the host drawing only as a file name*** (**along with its drive and path**).

By contrast, ***overlay*** and ***reference*** files never create an inherent association between themselves and any other drawing. They must always be loaded manually .

It is critical that the **XREF** file must exist on the ***same drive*** and ***directory path*** as when it was created in the ***host drawing***. **FastCAD** will not "**search**" for the file if it is not where the **XREF** ***pointer*** says it will be. This may create portability problems, especially if the ***host drawing*** contains **XREF**s to many differen drawing files in many different ***drives*** and ***directorues***.

If the host drawing is moved to another computer, the* XREF *files must be moved into the identical drive and directory path structure.

Like a ***symbol reference***, ***an* XREF *within a drawing cannot be directly edited within the host file***. You must instead load the **XREF** file as a ***separate*** drawing, then perform ***edits*** and ***save*** the file. When the ***host drawing*** is opened, the **XREF** ***will reflect the updates.***

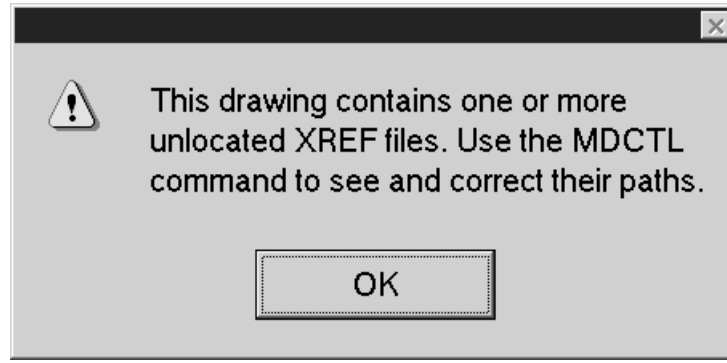
An **XREF** reference can be ***changed to point to a different file*** by clicking directly in the ***Work Dwg:*** *status window* to open the ***External Reference [XREF] Management*** *dialog box*.. Left-click the **XREF** file name and click the ***Change File*** *button*. **FastCAD** displays the standard ***Load Drawing*** dialog, from which you can choose a new **XREF** file. The new file will appear at the same ***Origin point*** as the original **XREF**.

Importing AutoCAD DWG and DXF drawings containing XREFs

You can import **AutoCAD files** with **XREF**s, but the exact original path (**DWG**) cannot be changed. If the ***host drawing*** is in **.DWG format**, then the **XREF** files must be **in .DWG** also. f the ***host drawing*** is in **.DXF format**, then the **XREF** files must be **in .DXF** format also.

If you open an **AutoCAD .DWG** or **.DXF** drawing with **XREFs** in them, you maus have the **XREF** drawings also or there will be missing information from the ***host drawing***. This missing information may be important to your project so **FastCAD** alerts .

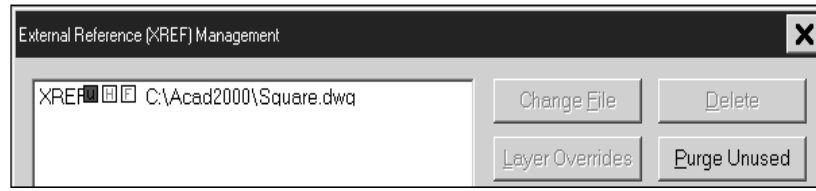
you to the fact that the drawing being imported contains **XREFs** that can not be located in the ***original drive*** and ***directory path*** pointed to by the ***host drawing***..



XREF Not Found Alert box

If you have the XREF files

If you were sent the **XREF** drawing files by the **AutoCAD** user, along with the ***host drawing, clcik directly in the Work Dwg:*** *status window* to open the ***External Reference [XREF] Management*** *dialog box*.. You will see the original **XREF *file name***, ***drive*** and ***directory path*** where it was originally stored. The left most checkbox will have a "**U**" in it signifing that it is ***Ulocated.***



External Reference [XREF] Management dialog box.

This checkbox normally has "**OK**" in it when the **XREF drawing** is located properly**.**

Left-click the **XREF** file name and click the ***Change File*** *button*. **FastCAD** displays the standard ***Load Drawing*** dialog, from which you can choose the ***drive*** and ***directory*** where you saved the **XREF** file**.** The **XREF** drawing will appear in th***e host drawing***.

If you do not have the XREF files

Contact the **AutoCAD** user that sent you the ***host drawing*** and have them send you the **XREF** drawings. Then use the method described above to locate them.

An alternative to this would be to Have the **AutoCAD** user use the **AutoCAD BIND** command which will convert **all XREF files referenced in the host drawing to *Symbols*** Then have them use a file compression progrm such as **WInZip** to compress it and send it to you again and the **XREFs** won't be an issue.

Inserting an XREF into a Fastcad Drawing

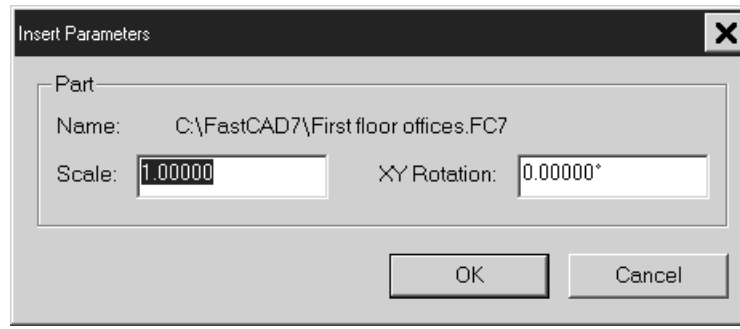
1. Click the **INSERT XREF** command from the **INSERT menu. FastCAD** displays the ***Load Drawing*** *dialog box*. This is the standard ***File Selection*** *dialog box*. Note that **FastCAD** displays the last inserted **XREF** as the default. To select an **XREF** to insert:
   - Double-click a filename;
   - Select a filename using the cursor and choose **Open**;
   - Type a drawing filename from the list of available drawings.

2. The prompt reades "**Insert part at/scale factor [Ins=specs]:**".

Insert part at/scale factor [INS=specs]:

You should see a ***ghost image*** of the **XREF** ***drawing*** attached to the moving cursor. If you want to change the ***scale*** of the original you may immediately ***type a new scale factor*** such as 2 to make the **XREF** twice as large. If you press the "I**ns**" key on the keyboard **FastCAD** displays the ***Insert Parameters*** *dialog box* with the ***default Scale factor*** and ***XY Rotation Angle*** of the **XREF**. You may adjust both ***scaling*** and ***rotation*** from this dialog box.. The **XREF** ***drawing name*** is also displayed in the dialog box.

- ***The default scaling is 1.0*** (**same size as the original drawing**). Type in a ***new scale factor*** if you wish to insert the **XREF** at a larger or smaller size.
- The default ***XY rotation angle*** **is 0.0°**. If you wish to ***rotate*** the **XREF** upon insertion, type in a new ***angle between 0 degrees and 360 degrees***. For example, type "**45**" to ***rotate*** the **XREF** ***45 degrees counter-clockwise***.

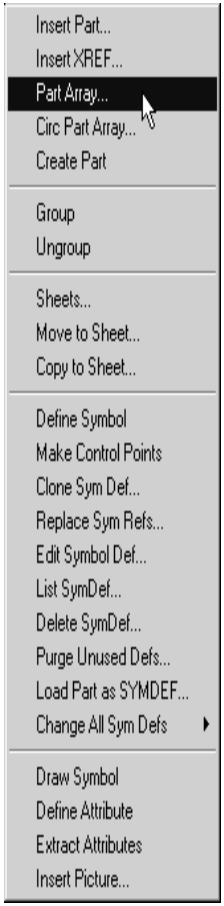


Insert Properties dialog box

3. Choose the **OK** button to accept the ***scaling*** and ***rotation*** values.
4. The prompt reads "**Insert part at/scale factor [INS=specs]**:". Enter an insertion point by using the crosshairs, or by typing numeric coordinates.
5. The prompt continues to read "**Insert part at/scale factor [Ins=specs]:**". Continue inserting copies of the **XREF** ***drawing***. Right-click to end the command.

Text equivalent: ***XREF***

Part Array

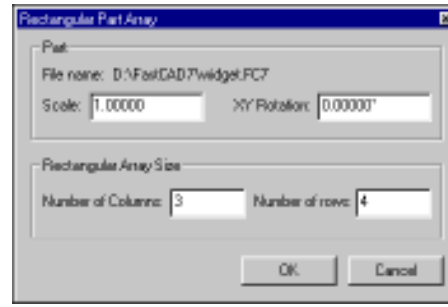


PART ARRAY builds a ***rectangular array*** (**rows and columns**) from ***multiple copies*** of a ***part*** file, arranging the parts in regularly spaced ***columns*** and ***rows***.

To create an ***array*** of **parts**, select **[Insert > Part Array]**:

1. **FastCAD** displays the ***Part to Insert*** *dialog*. This is a variant of the standard File Selection dialog. To specify a ***part*** to ***insert*** and ***array***:
   - Double-click a filename;
   - Pick a filename and click **Open**;
   - Type a drawing filename from the list of available drawings, and press ENTER.

2. **FastCAD** displays the ***Part to insert*** *dialog* and then the ***Rectangular Part Array*** *dialog*.from which you may adjust ***part scaling, rotation***, and ***array*** options.



Rectangular Array dialog

File name The file you specified in Step 1 is displayed here. To change the file, you must cancel the command and start over.

Scale Sets the size of all ***parts*** inserted into this ***array***. **The *default scaling is* 1.0** (**same size as the original *part***). Type in a new ***scale factor*** if you wish to insert the ***part*** at a larger or smaller size.

XY Rotation ***Part rotation in the XY plane*** (**rotation center is the part's origin**). The ***default XY rotation angle is* 0.0°.** If you wish to rotate each part by the same angle upon insertion, type in a new angle between 0 degrees and 360 degrees. For example, type "**45**" to ***rotate each part 45 degrees counter-clockwise***.

Number of columns The number of vertical columns in this *array*.

Number of rows The number of ***horizontal rows*** in this ***array***.

3. Click **OK** to confirm the parameters.

4. The prompt reads "**Insert 1st part at:**". To specify the first ***insertion point***, pick a point on the screen, or type coordinates.

   FastCAD displays a rubber-band cursor, anchored at the insertion point. If you are drawing in **3D depth mode**, see step 4a.

4a. If **3D Depth mode** is on, the prompt reads "**Depth from window pt/value [prior]:**". From the appropriate **3D** view, pick a ***depth point*** on the screen, or type coordinates.

FastCAD displays a rubber-band cursor, anchored at the ***insertion point***.

5. The prompt reads "**2nd column 2nd row at:**": This offset from the insertion point tells **FastCAD** both the horizontal and vertical distance between parts. Use the rubber-band cursor to help you judge the spacing. Pick a point when you are satisfied with the distance.

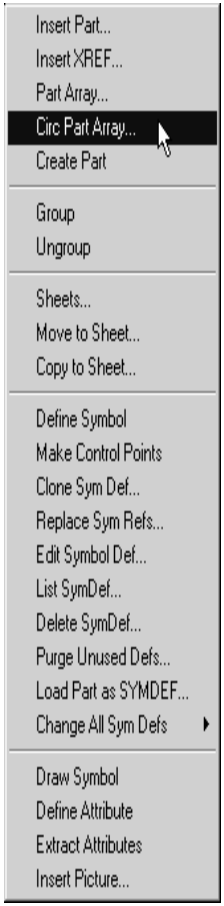
FastCAD draws the array and terminates the command. If you are drawing in **3D** depth mode, see step 5a.

5a. If **3D Depth mode** is on, the prompt reads "**Depth from window pt/value [prior]:**". From the appropriate **3D** view, pick a ***depth point*** on the screen, or type coordinates.

FastCAD draws the ***array*** and terminates the command.

Text equivalent: ***PARY***

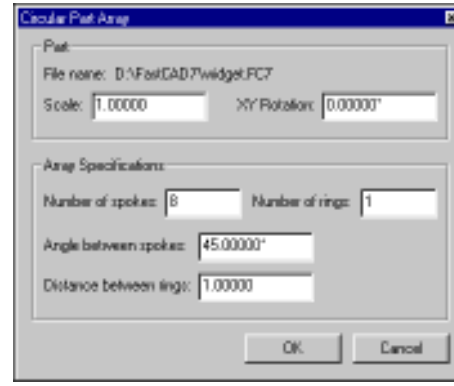
Circ Part Array



CIRC PART ARRAY (**Circular Part Array**) ***inserts multiple parts arranged in rings***. It's like a combination of **CIRCULAR ARRAY** and **PART ARRAY**. The angle of the first ***spoke*** (from the center to the "**copy origin**" or insertion point) determines the ***angle of each part in the array***.

To draw a ***circular part array***, select **[Insert> Circ Part Array]:**

1. **FastCAD** displays the ***Part to Insert*** *dialog*. This is a variant of the standard ***File Selection*** *dialog*. To specify a ***part to insert and array***:
   - Double-click a filename;
   - Pick a filename and click **Open**;
   - Type a drawing filename from the list of available drawings, and press **ENTER**.
2. **FastCAD** displays the ***Circular Part Array*** *dialog box*. You may adjust ***part scaling***, ***rotation***, and ***circular array*** options.



Circular Part Array dialog

File name The file you specified in Step 1 is displayed here. To change the file, you must cancel the command and start over.

Scale The size of all ***parts*** inserted into this ***array***. The ***default scaling is*** **1.0** (**same size as the original part**). Type in a ***new scale facto***r if you wish to insert the ***part*** at a larger or smaller size.

XY Rotation Each ***part's rotation relative to the*** **"0°"** ***axis*** defined by each ***spoke*** (**rotation center is the** ***part's origin***). The ***default XY rotation angle is*** **0.0°**. If you wish to ***rotate*** each ***part*** by the same angle upon insertion, type in a ***new angle between 0 degrees and 360 degrees***. For example, type "**45**" to ***rotate each part 45 degrees counter-clockwise, relative to its spoke.***

Number of spokes The ***number of spokes*** in the ***array***. Parts are radially aligned with each ***spoke*** upon insertion. Enter an integer value of at least 1 (though in most cases, 1 spoke is a trivial case). ***Each spoke in a circular array is evenly spaced from the others.***

Number of rings The ***number of ring***s in the ***array***. The default and minimum (integer) value is 1. ***Each ring is a circular array is evenly spaced from the others***.

Angle between spokes The ***angle between spokes***. ***The default value of the angle is always pre-calculated to space the parts evenly around a full circle***. You may override this value with a different angle to evenly arrange the spokes around a partial circle.

Distance between rings The ***distance between rings***. Value ignored if **Number of rings** = 1.

3. Click **OK** to confirm the parameters.

4. The prompt reads "**Array Center:**". To specify the ***center point***, pick a point on the screen , or type coordinates.

FastCAD displays a rubber-band cursor, anchored at the ***center point***. If you are drawing in **3D depth mode**, see step 4a.

4a. If **3D Depth mode is on**, the prompt reads "**Depth from window pt/value [prior]:**". From the appropriate **3D** view, pick a ***depth point*** on the screen, or type coordinates.

FastCAD displays a rubber-band cursor, anchored at the ***center point***.

5. The prompt reads "**Insert 1st part at:**". Pick the ***insertion point for the first part*** (**on the inner ring of the first spoke**).

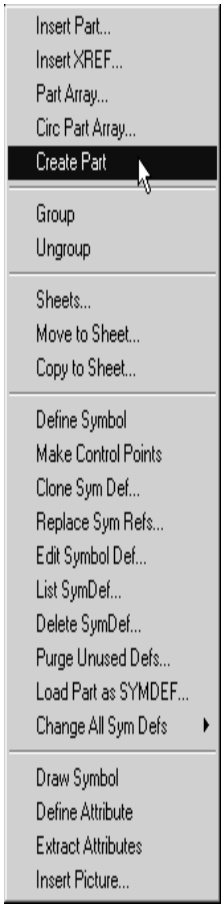
FastCAD draws the ***circular array*** and terminates the command. If you are drawing in **3D depth mode,** see step 5a.

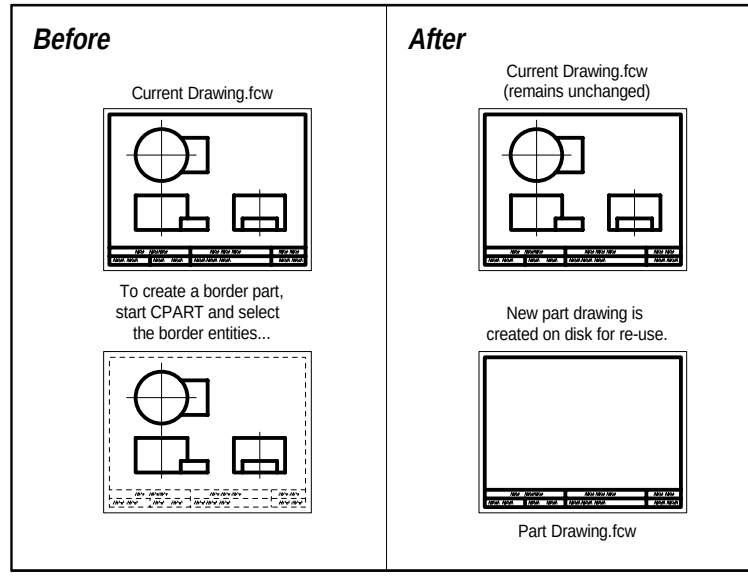
5a. If **3D Depth mode is on**, the prompt reads "**Depth from window pt/value [prior]:**". From the appropriate **3D** view, pick a ***depth point*** on the screen, or type coordinates.

FastCAD draws the ***circular array*** and terminates the command.

Text equivalent: ***CPARY***

Create Part





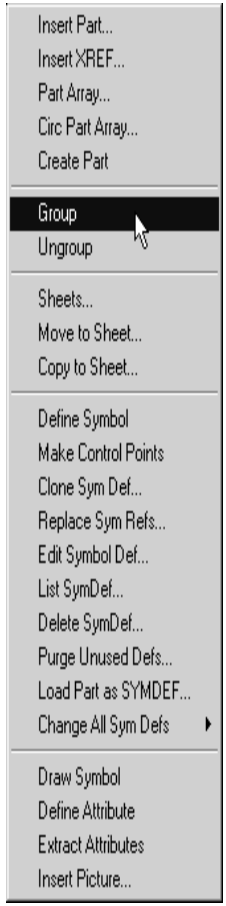
CREATE PART ***saves selected entities from the active drawing as a separate drawing. You can use this command to create a number of parts to use over and over.***

To ***create a part*** from the current drawing, select **[Insert > Create Part]**:

1. **FastCAD** displays the ***Create Part Name*** *dialog box*, which is a variant of the standard ***Load Drawing*** *dialog*, allowing you to ***select*** or ***create a filename*** to which the ***part*** will be written. Remember that the ***part file*** will be a standard **FC7** type file that can be ***opened by itself or inserted as a part***.
2. Select entities to be included in the ***part***.
3. The prompt reads "**Part origin:**". Specify the ***part origin*** by picking a point or typing numeric coordinates. This is the (**0,0,0**) point for the ***new part file***. **FastCAD** ***will align the new part's origin with the insertion point when you insert the part into another file***. Usually, you will select a point on the lower-left corner or center of the entities to be included in the part.
4. **FastCAD** copies the selected entities into the specified file. ***If entities you select for the part are already part of a group, they will be assimilated into a new part group when the new part is inserted into another drawing.***

Text equivalent: ***CPART***

Group



GROUP ***combines multiple entities so they act like a single entity***. Use **GROUP** to create structures composed of multiple entities that you want to keep together.

Groups maintain a hierarchical (or "**nested**") structure. As groups are combined, they retain their original identities and member entities (each group is internally assigned a unique number that identifies itself). For example, if two existing ***groups*** were grouped together and then **UNGROUP**ed, the two original groups would still exist.

Individual entities in a group cannot be edited. Single-entity **EDIT** commands such as **BREAK** or **TRIM** will not operate on entities in a ***group***. ***Multiple-entity*** **EDIT** ***commands will operate on the entire group, even if some entities in the group are on hidden layers.***

When you ***insert parts*** with **INSERT PART**, **PART ARRAY**, or **CIRC PART ARRAY**, ***they come in the drawing as groups***. You can toggle this behavior using the **OPTIONS** command **[Spec > Options]**. Remove the check from the box labeled "**Group parts on insertion**" to prevent this ***automatic grouping***

To break a group temporarily in your drawing Click the ***Unlock*** *button* on the ***Button*** *bar* so that its label reads "**Unlocked**". ***In this state, all group effects are turned off***. ***Entities behave as if they were never grouped***. Full group effects, including all nested levels, can be restored by clicking on the ***Unlocked*** *button* so that its label reads "**Locked**".

To break a group permanently in your drawing Use the **UNGROUP** command. **UNGROUP** removes one level of grouping.

To remove all levels of grouping from a grouped entity Use the **EXPLODE** command **[Edit > Explode]**. Make sure that you select by **"Entity Type"** ***Part Groups*** and not and not by ***All***. When you use the **EXPLODE** comman never select entities by ***All*** unless you really mean to do this. Entities will revert to more base structures. ***Polygons*** will become ***lines***, ***dimensions*** will become ***lines***, ***text*** and ***polygons*** for ***arrows*** etc.

To determine the hierarchical structure of a nested group Use the **LIST** command **[Info > List]** and select the ***group(s)*** you wish to inspect.

Text equivalent: ***GROUP***

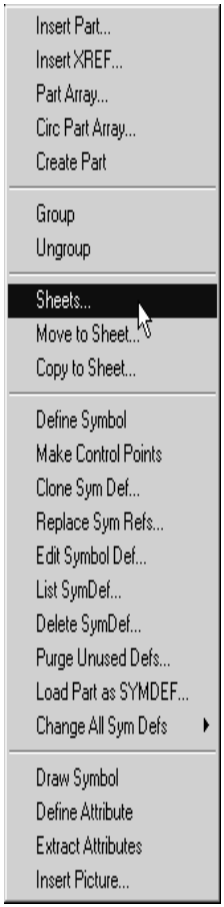
Ungroup

UNGROUP ***breaks up a selected group into its base entities***. This effect of **UNGROUP** will be permanent unless it is part of a valid **UNDO** sequence.

If you only wish to ***edit an entity in a group***, you should click the ***Locked*** *button* on the ***Button*** *bar* to ***temporarily suspend group effects***.

When using this command on a ***nested group,*** **UNGROUP** ***removes one level of grouping.*** To remove all levels of grouping, use the **EXPLODE** command **[Edit > Explode]**. Make sure that you select by **"Entity Type"** ***Part Groups*** and not and not by ***All***. When you use the **EXPLODE** comman never select entities by ***All*** unless you really mean to do this. Entities will revert to more base structures. ***Polygons*** will become ***lines***, ***dimensions*** will become ***lines***, ***text*** and ***polygons*** for ***arrows*** etc.

Text equivalent: ***UNGROUP***



Sheet commands

Sheets let you store multiple-page drawings in a single **FC7** ***file***. ***All sheets share common layer/style settings and symbol definitions.*** Multiple ***sheets*** can be printed in a single ***batch operation***. If you create several pages of drawings and keep them at different locations in one file, ***sheets*** will significantly speed up your work.

You can also use ***sheets*** as "**super layers**", since there is a similarity in concept. Like layers, ***sheets*** can be ***hidden*** or ***shown***. All entities on each ***sheet*** are sorted into order by sheet name, back to front.

Because ***sheets*** contain an entity sub-list that can be ***hidden*** or ***shown***, ***sheets*** can greatly speed up the handling of large drawings. Entities on ***hidden sheets*** are ***disregarded during entity selections and redraws***. By contrast, layer information is stored as an entity property, so each entity must be processed before it is hidden, shown, or selected. Note that the **LIST** command will display ***sheet*** membership information.

All **FastCAD** drawings use the "**common**" ***sheet***, as it is the standard, non-sheet drawing. The **COMMON** ***sheet*** may be selected, but ***never hidden as a sheet.***

FastCAD provides commands to ***create***, ***rename***, and ***delete sheets***.

Managing Sheets

Use the **SHEET** command to display the ***Drawing Sheets*** *dialog*. This dialog allows you to *create*, *rename*, and *delete sheets*. Use the checkbox controls to make any one *sheet current*, or to *hide/show* any *sheet*. The **COMMON** ***sheet*** **cannot be deleted.**

There is no practical limit to the number of sheets a drawing can contain (A single drawing can accommodate over 30,000).

Drawing and Editing on Sheets

All new and copy entities are placed on the current sheet.

The special commands **MOVE TO SHEET** and **COPY TO SHEET** let you select entities on any ***visible sheet*** and either ***move*** or ***copy*** them to a ***selected sheet***. The ***destination sheet*** is the one you select in the ***Drawing Sheets*** *dialog* (**entities can be moved/copied to sheets that are currently hidden**).

If you ***delete*** a ***sheet***, then all entities on that sheet are also ***deleted***. Be careful; **UNDO** will not restore a ***sheet*** that has been ***deleted***!

Printing Sheets

The ***Print Drawing*** *dialog* contains ***sheet*** commands in the ***View to Print*** *options*. The ***Sheet*** drop doen menu lets you specify one of five standard ***sheet printing*** options:

- **COMMON only** (**standard drawing**)
- **All visible sheets as overlays** (**single page**)
- **All visible sheets as separate pages**
- **All sheets as separate pages**
- **The currently selected sheet**

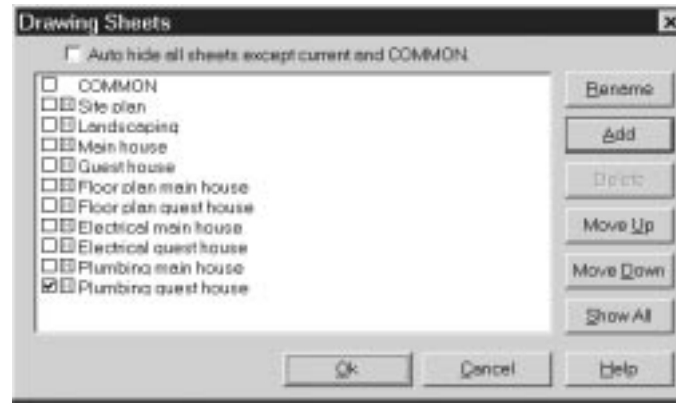
You can also choose the check box that allows "**common**" entities (those not assigned to any ***sheet***) to be printed on each separate page of a multi-page print.

Sheet


Sheets icon

Use the **SHEET** command to display the ***Drawing Sheets*** *dialog*. This dialog allows you to ***create***, ***rename***, and ***delete sheets***. Use the checkbox controls to make any one s***heet current***, or to ***hide/show*** any ***sheet***. Note that the **COMMON** ***sheet*** cannot be deleted.

In the illustration below, the ***sheets*** **COMMON**, "**Project A Block Text**", and "**A Electrical**" are visible. The ***current sheet*** is "**A Electrical**", so any new entities will be created on this ***sheet***.



Drawing Sheets dialog box

Auto hide all sheets except current and COMMON When checked, ***only the current and*** **COMMON** ***sheets are visible***. Whenever you choose a ***new current sheet***, all others (**including the previous current**) ***are automatically hidden***. If you wish to have ***more than one sheet visible*** (not including **COMMON**, **which is always visible**), ***this option must be unchecked***.

List Control All ***sheets*** contained in the ***active drawing*** are listed here. The **COMMON** ***sheet is always listed first***. Additional ***sheets*** are listed in the order created, but you may reorder them using the ***Move Up*** and ***Move Down*** *buttons*. ***Sheets*** will both ***draw*** and ***print*** in this order.

The ***first column*** checkbox indicates the ***current sheet***. Choose the ***current shee*t by clicking directly on the ***first column checkbox*** by the ***sheet name***. ***Only one sheet can be current at any time. The current sheet is always visible.***

The ***second column checkbox*** indicates the ***visible/hidde*n state of the ***sheet***. If ***Auto hide all sheets…*** is ***ON***, then *only the* ***current sheet and COMMON will be made visible***. If ***Auto hide all sheets…*** is ***OFF***, then ***other sheets can be made visible*** by clicking directly on the ***second column checkbox***. Clicking on the checkbox toggles ***hidden*** "**H**" (**dark background**) ***or visible*** (**white background**) ***states***.

Rename button Choose to ***rename*** the ***selected sheet***. Any ***sheet*** except **COMMON** can be ***renamed***.

Add button Choose to ***create a new sheet*** in the ***active drawing***. The ***new sheet*** is added to the bottom of the page. **FastCAD** will not let you duplicate existing ***sheet names***. ***Sheet names*** are not case sensitive, so "abc" is the same as "**ABC**". ***Maximum length is 63 characters***.

Delete button Choose to ***delete*** the ***selected sheet***. The ***sheet name*** and all entities on the ***sheet*** will be erased from the ***active drawing***. Use with care, as:

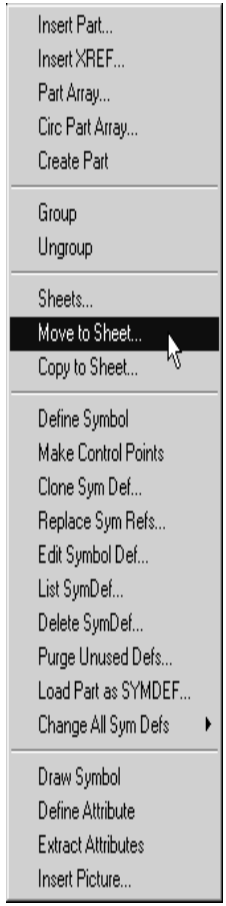
- **UNDO** will **NOT** reverse this operation, and
- Choosing the **Cancel** button will not restore deleted sheets.

Move Up button Choose to ***move*** the ***selected sheet name up one positio****n* in the list. **COMMON** is permanently listed first. **Note:** ***Move Up*** and ***Move Down*** are not undone by **Cancel**.

Move Down button Choose to ***move the selected sheet down one position*** in the list. **Note:** ***Move Up*** and ***Move Down*** are not undone by **Cancel**.

Text equivalent: ***SHEET***

Move to Sheet



MOVE TO SHEET ***lets you move selected entities from one sheet to another***. The t***arget sheet does not need to be current or visible. The target sheet can be created during the operation.***

Like the **CHANGE . LAYER** command, the selected entities a***re moved to the target sheet without being displaced.*** That is, the entities remain in the same coordinate space, but they are now members of a ***different sheet***.

To ***move entities to another sheet***, select **[Insert > Move to Sheet…]**:

1. Select entities to ***move***.
2. **FastCAD** displays the ***Drawing Sheets*** *dialog*. Select the ***target sheet*** and click **OK**.

   You can create the target sheet at this time by clicking **Add**.

   FastCAD then ***moves*** the entity to the ***target sheet,*** and the command terminates.

Text equivalent: ***MOVESHT***

Copy to Sheet

COPY TO SHEET ***lets you duplicate selected entities to another sheet***. ***The selected entities remain unchanged. The target sheet does not need to be current or visible. The target sheet can be created during the operation.***

Like the **COPY TO LAYER** command, the selected entities are ***copied to the target sheet. That is, the new entities are duplicated and reside to the same coordinate space, but they are members of a different sheet***. You could ***hide*** one ***sheet*** and use the **DRAG** command and ***move the copy to a different location*** then ***show*** the h***idden sheet*** and ***the original entity would remail in the original location***. These entities may be on the ***same layer*** but on different ***sheets.*** Using ***sheets*** is the only way to have entities residing on the **same** ***layer*** and by assigning them to different ***sheets***, selectively ***hide*** and ***show these sheets***.

To ***copy entities to another shee***t, select **[Insert > Copy to Sheet…]**:

1. Select entities to ***copy***.
2. **FastCAD** displays the ***Drawing Sheets*** *dialog*. Select the ***target sheet*** and click **OK**.

   You can create the target sheet at this time by clicking **Add**.

   FastCAD then ***copies the entities to the target sheet***, and the command terminates.

Text equivalent: ***COPYSHT***

Symbol commands

A symbol is a named collection of entities that defines a complex object—each symbol acts like a single entity. Like ***parts***, ***symbols*** provide a mechanism for handling repeated elements. To use ***symbols***, ***create a symbol definition*** that **FastCAD** stores in an invisible section of the ***active drawing*** file. Then you insert copies of the ***symbol***, called ***symbol references***, throughout the drawing. ***Because each reference is only an ghost image of the symbol definition, you can add hundreds of symbols without excessive memory usage.***

Symbol Definition

- The mathematical representation of the entities comprising the ***symbol***.
- Are never visible on the drawing screen.
- Can only be ***created*** using the **DEFINE SYMBOL** command.
- Can only be ***edited*** using the **EDIT SYMBOL DEF** command.
- Can be introduced into a drawing by ***inserting a part with symbols***. The use of such ***symbol libraries*** makes ***symbols portable*** from one drawing to another.
- Can be singularly introduced into a drawing by inserting the ***symbol reference*** via the ***Symbol Catalog viewer***. The use of ***Symbol Catalog file***s makes symbols efficiently portable from one drawing to another. You only insert ***symbols*** you want from the ***Symbol Catalog viewer***
- Should never be duplicated in a single drawing. Duplications wastes memory and may cause problems when drawings are converted to **DWG** or **DXF** format for use in other **CAD** programs.
- Deleting the ***definition will delete all references!*** The ***symbol references*** will no longer appear anywhere on your drawing.

Symbol Reference

- The on-screen "**visible**" representation(s) of the **symbol definition**. The ***reference*** gets its drawing information ***from the definition***.
- Can be inserted by using the **DRAW SYMBOL** command.
- Can be inserted by using the ***Symbol Catalog viewer***.
- Always has ***an associated symbol definition***. ***References*** cannot exist without the ***definition***.
- There can be more than one ***reference*** for any ***definition***.
- Editing or erasing the ***symbol reference*** does not affect its corresponding ***definition*** in any way.
- Can be ***rotated***, ***scaled***, or ***moved***; but you cannot make internal changes without ***exploding*** the ***symbol reference***.
- ***Exploding*** a ***symbol reference*** breaks it down into its component entities. Once a ***symbol reference*** has been ***exploded***, it is no longer dependent upon the ***definition***.

Note: ***Symbols*** in drawing files imported from other application programs may appear to be ill-behaved. If possible, have the exporting program "**explode**" its ***symbols*** or "**blocks**" before importing or updating the drawing.

Smart Symbols

When working with **wall networks**, you can use a special type of "**smart**" ***symbol***. ***Smart symbols*** add a level of automated intelligence to ***symbol*** placement.

Smart symbols, when inserted, ***automatically align themselves with the span into which they are inserted.*** Also, they can automatically "**cut**" and trim off the ***wall upon*** insertion.

Smart symbols can be easily created with the simple addition of ***control points*** to the ***symbol definition***. Not only does that permit easy Icreation, but existing ***symbols*** can also be rapidly converted into ready-to-use ***smart symbols***.

Xrefs

When the **XREF** command is used to create a combined drawing file, the ***xref file*** is actually inserted as a ***symbol reference.*** In essence, an **XREF** is a ***symbol definition stored in another file***. The host drawing simply contains a ***reference*** that contains the **XREF** file's ***drive***, ***path***, and ***file name***. As long as that information is valid, The **XREF** files will appear automatically when the host file is loaded.

Commands for managing symbols

- **DEFINE SYMBOL** ***creates a named symbol definition***;
- **CLONE SYMBOL** lets you ***copy an existing symbol definition to a new name***;
- **REPLACE SYM REFS** instructs a ***symbol reference*** to use a different ***definition***;
- **EDIT SYMDEF** permits ***modification of a referenced definition***;
- **LIST SYM DEF** ***displays or prints a*** l***ist*** of ***symbol definitions***;
- **DELETE SYMDEF** ***removes a symbol definition*** and ***references*** to it;
- **PURGE UNUSED DEFS** ***removes unreferenced definitions***;
- **DRAW SYMBOL** ***inserts symbol references*** at any angle or scale.

To ***create a symbol definition***, draw the individual entities, then use **DEFINE SYMBOL** to give it a ***name*** (**up to 31 characters**) and an ***origin point***. **FastCAD** moves the entities that make up the ***symbol definition*** to the drawing's invisible ***symbol area***, ready to be referenced by **DRAW SYMBOL**.

Symbol definitions are part of the current drawing only. To ***share symbols*** among several drawings, create a drawing file that contains nothing but ***symbol definitions*** and save it as a **FastCAD** ***Symbol Catalog*** (**.FS7**) ***file***. You can then use the ***Symbol Catalog viewer*** to add the ***symbol definitions*** and ***references*** to any drawing.

To modify symbol references**, **create the new symbol definition, then use the **REPLACE SYM REFS** ***to direct the references to the new definition.***

If ***symbols*** appear to behave erratically, **EXPLODE** them.

Colors and Symbols

By default, symbols will not change colors when you insert them into a drawing**. **That is, all symbol references will display the same color properties as the definition, regardless of the current color.

However, ***symbols*** can be defined so that they will appear in the ***current color*** when inserted. The recommend method is the ***Color by Block*** method. Before defining a ***symbol***, apply the ***Color by Block*** property to those entities you wish to color update. Do this by using the **CHANGE > TO COLOR OF STMREF** command [**Edit > Change > to Color of SymRef]**. Alternatively, you can use the **PROPERTIES** command **[Edit > Properties…]** and toggle the **Change > to color of SymRef** option. Any number of entities in the ***definition*** (none, some, or all) can use the ***Color By Block*** property. Those that do not are "**color locked**". You can use the **EDIT SYMBOL DEF** command to toggle ***Color by Block*** in existing ***symbols***. ***Redefined symbol references will change to the color that was current when inserted***.

The Symbol Catalog Viewer

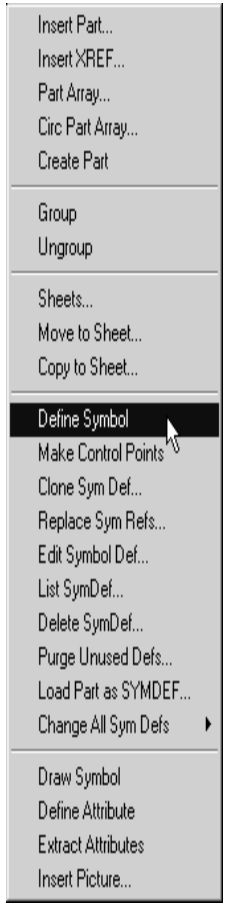
FastCAD provides you with a unique graphical management system for your ***symbols***. The ***Symbol Catalog*** *viewer* allows you to:

- Access separately maintained ***symbol catalog files*** (**.FC7** files);
- Select and place ***symbols*** into your drawing with a simple point and click system.

Toggle the ***Symbol Catalog viewer*** on or off using the **SCREEN TOOLS** command **[Specs > Screen Tools]** or its icon equivalent.

FastCAD has two ***Symbol Catalog viewers*** that can be displayed by clicking the the left or right arrow position box in the ***Screen Tools*** *dialog box***. Y**ou can point each ***viewer*** to a different **.FS7** ***Symbol Catalog*** file**.** More ***Symbol Catalog viewers*** may be added by modifying the **FCW7.CFG** file to define them.

Define Symbol



DEINE SYMBOL collects entities to make a ***symbol definition***.

To create a ***symbol definition***, select **[Insert > Define Symbol]**:

1. The prompt reads "**New Symbol Name:**". Type a name then press **ENTER**.
2. The prompt reads "**Origin At:**". Pick a point that will be used as the ***symbol's insertion point***, usually at the lower-left or center of the new ***symbol***.
3. Select the entities to be included in the ***symbol definition***. After you **Do it,** the newly created ***symbol definition will disappear***.

Use **DRAW SYMBOL** to create visible ***references*** to the ***symbol definition***.

UNDO brings back the entities that went into the ***symbol definition***, but does not un-define the ***symbol***! Use **LIST SYM DEF** to see ***names*** or ***pictures*** of ***symbol definitions*** or use **DELETE SYM DEF** to erase a definition.

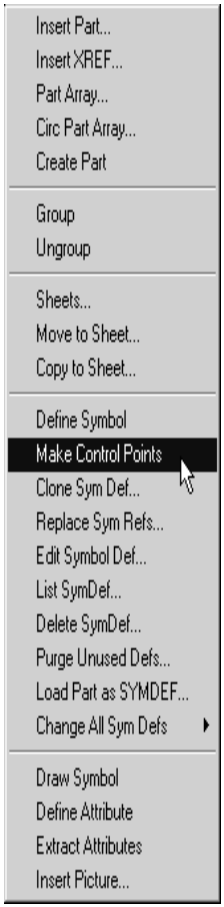
You can create ***symbol catalog* FS7** files for use in **FastCAD**'s ***Symbol Catalog viewer***. A ***symbol catalog*** is a drawing file that contains only ***symbol definitions***. You can create a ***symbol catalog*** by creating the ***symbol definitions*** in a drawing and saving it as an **.FS7** file. ***Symbol catalogs*** should use ***no grouping and contain noting but symbol definitions***. Entries for "**FastCAD FC7** ***Symbol Catalogs***" are available in the standard ***Load Drawing*** *dialogs*.

Note: When you select entities for a ***new symbol definition***, you can include ***references*** to other ***symbols***. Keep in mind that ***nested symbols*** slow **FastCAD** down, especially if the ***references*** are scaled or rotated.

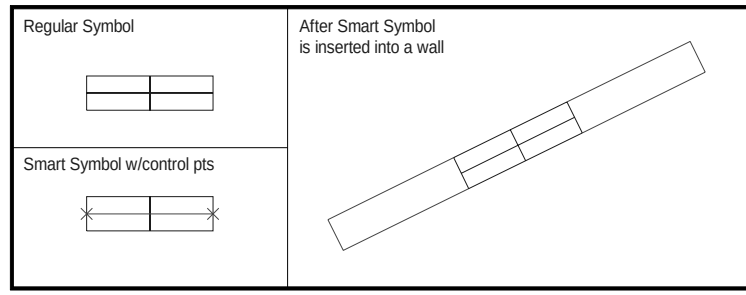
For more help understanding **symbol definitions** and ***references***, please see **Symbol** commands.

Text equivalent: ***DEFSYM***

Make Control Points



Control points are added to regular symbols to create "**smart symbols**". ***Smart symbols*** work intelligently with ***wall networks***, in that both the **symbol** and ***wall network are aware of each other during insertion***. This awareness allows the ***wall network*** to automatically create *cuts* to accommodate the ***symbol***, or have the s***ymbol align or scale itself to fit cleanly***. Typical uses for ***smart symbols*** would be doors, windows, etc.



Note that ***control points*** are invisible after a ***symbol*** is inserted.

Control points are added to ***symbols before they are defined***. When creating new ***symbols***, just add ***control points*** as entities and be sure to include them when you select entities during the **DEFINE SYMBOL** phase. If you are working with ***existing symbols***, you can use the **EDIT SYMBOL DEFINTION** command to add control points.

To add ***control points***, select **[Insert > Make Control Points]:**

The prompt reads "**1st point:**" and then "**Next point [done]:**". The positioning of these two points is important.

Firstly, the two points define a vector that is used to determine *smart symbol* alignment with the ***wall***. During placement, ***this vector remains parallel to the direction of the wall span***, reorienting the ***symbol*** accordingly. This allows a single ***smart symbol*** to be used for ***vertical walls***, ***horizontal walls***, or ***walls along any bearing***! The alignment vector also tells **FastCAD** how to "**flip**" the ***smart symbol*** during insertion.

The two ***control points also define the wall*** "**cut points**" ***when the symbol is inserted into a wall. The smart symbol will cut the wall just as the*** **CUT FROM** command would. ***The wall cuts are capped perpendicularly to the span direction.***

Control points are necessary for any ***smart symbol***, but you can turn off their ability to align or cut upon insertion using the ***Control Points Effects*** *dialog*. This dialog appears automatically after you draw the two ***control points***.

Control Points Effects dialog box

Control point effects can only be toggled at this time, before the ***symbol*** is defined. If you need to change the ***control point*** effects for an already defined ***symbol***, you need to use **EDIT SYMBOL DEF** to remove the existing ***control points***, and add new ones.

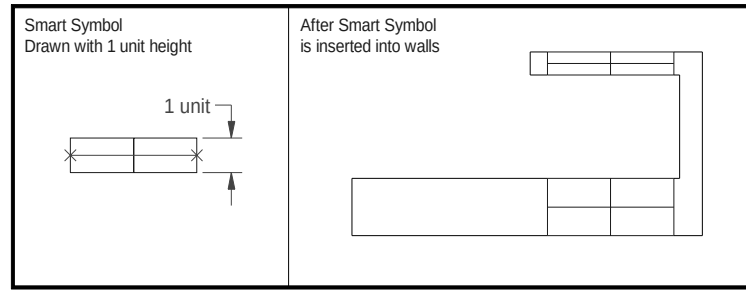
A symbol definition can only contain one pair of valid control points.

Align on Insertion ***Enable this option to auto-rotate the reference upon insertion.*** The smart symbol will only align when placed on a linear entity. The two control points define the direction of the ***smart symbol*** alignment.

Cut on Insertion Enable this option to automatically break the existing linear entity to accommodate the smart symbol. The break will occur at the *control points*.

Scale Y to Fit Enable this option to force the *smart symbol* to resize in the "vertical" direction upon placement. This resizing is relative to the width of the wall into which the *smart symbol* is inserted. This option only takes effect when inserting into entities with width > 0.

You can define a smart symbol so that it automatically snaps to the same width as any wall. To do this, the ends of the symbol that abut the wall cut (where the control points go) must be one unit long. Symbols constructed with **Scale Y to Fit** will scale in the ***Y-direction*** *only*. The length in the ***X-direction*** (the vector defined by the control points) remains constant. This can cause circular entities to distort into ellipses upon insertion.



Keep DynTrack Scale If checked, causes ***symbol*** to honor ***dynamic scaling/ rotation*** performed during insertion into linear entities. Hold down the **CTRL** key

during insertion to ***dynamically scale***. If unchecked, the ***symbol*** will insert using its defined size. Leave this option unchecked to "**lock**" insertion ***size*** and ***alignment***.

Offset from Place Point Enable this option to insert the symbol parallel to the initial placement point and orientation. The exact placement is visually determined from the dynamic cursor. This overrides the ***Cut on Insertion*** *check box* option (i.e., offset symbols will not cut the locating entity).

Text equivalent: ***CTRLP***

Clone Symbol

Use the **CLONE SYM DEF** command to create and edit a copy of an existing symbol definition. For instance, you may require a symbol definition that is slightly different than one already in your drawing. Instead of going through the process of inserting the similar symbol, exploding it, modifying it, and then redefining it, you can do everything in one step.

To ***clone a symbol,*** type **CLONESYM** at the command promps and press **ENTER**, or choose **[Insert > Clone Sym Def…]** from the menu:

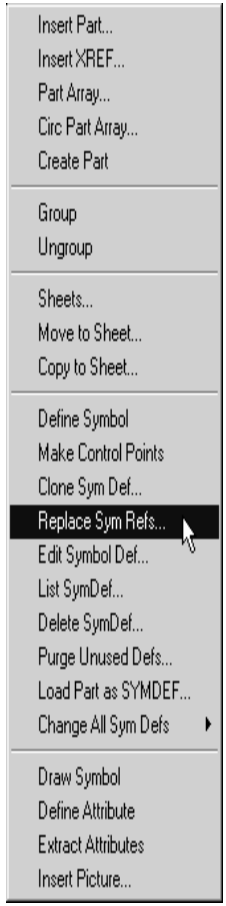
1. **FastCAD** displays the ***Select Symbol*** *dialog*. Select the ***definition*** you wish to ***clone***.
2. The prompt reads "**New Symbol Name:**". Type in a name for the ***new symbol*** and press **ENTER**.
3. **FastCAD** displays the *symbol* in a ***full size window*** on top of the drawing window with the new symbol name in its title bar.
4. You can now perform regular ***draw*** and ***edit*** operations as if this were a normal drawing. When you are done modifying the ***symbol definition***, choose the ***CLOSE*** *icon* or select **[File > Close]** command from the menu..

FastCAD displays an alert box as seen above that asks if you want to keep the changes to you made. If you choose the **Yes** button, the new symbol definition will be saved within the drawing. You can now draw symbol references using the **DRAW SYMBOL** command. If you choose the **No** button, then the clone will still be saved, but any edits you may have performed will be ignored. Either choice will close the symbol window.

Note that **UNDO** will not remove a symbol clone. If you wish to remove a symbol clone, use the **DELETE SYMBOL DEF** command.

Text equivalent: ***CLONESYM***

Replace Symbol



The* REPLACE SYM REF *command redirects all symbol references pointing to one definition to use a new definition instead. This command is particularly helpful when ***symbol references*** must be updated to reflect changes to the ***symbol definition***.

Before using the **REPLACE SYM REF** command, the target symbol definition must already exist in the drawing.

To use the ***REPLACE SYM REF*** command, select **[Insert > Replace Sym Refs..]**:

1. The ***Symbol to be Replaced*** *dialog box* appears. Select the ***symbol references*** you want to change by selecting the corresponding ***symbol definition***. (**Remember that you are not actually changing the *symbol definition;* you are simply redirecting the *symbol reference* from one *symbol definition* to another**. In this step, you are choosing the "**from**" ***symbol definition***.)
2. The dislog box title now reads ***Replace with Symbol***. Select the ***new symbol definition*** (the "**to**" ***definition***).
3. **FastCAD** redraws the ***old symbol references*** using the ***new symbol definition***. Use the **UNDO** command t***o revert back to the original symbol definition***.

FastCAD updates all ***symbol references*** with the ***new symbol definition***.

EXAMPLE: Suppose you have a ***symbol*** of an office chair in your drawing named **CHAIR01**, and that there are twelve ***references*** in your drawing (meaning you can see twelve chairs). Now you want to change these chairs to a new style, keeping their positions and orientations as they are. The new style is defined in the ***symbol*** **CHAIR04**. Use the **REPLACE SYM REF** command, selecting **CHAIR01** in the ***Symbol to be Replaced*** *dialog box*, then selecting **CHAIR04** in the ***Replace with Symbol*** *dialog box.* All twelve ***symbol references*** that previously showed **CHAIR01** now display **CHAIR04**.

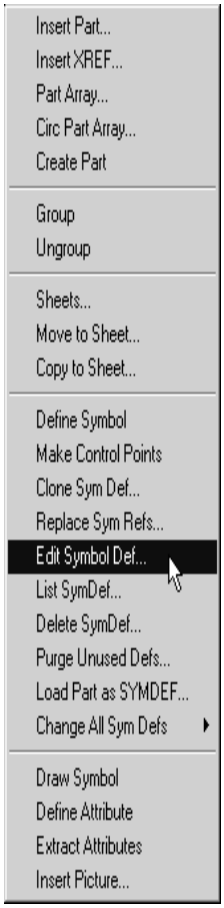
In the above example, **CHAIR04** must have already been defined in the drawing. This could have been done by previously inserting a **CHAIR04** ***symbol*** from the *S**ymbol Catalog** viewer*. Or you could have drawn the **CHAIR04** from scratch. You might have also **EXPLODE**d a **CHAIR01** ***reference***, edited it (e.g., by adding a chair back), then used the **DEFINE SYMBOL** command to create the **CHAIR04** ***symbol***, taking care to select the same ***symbol origin***.

Note: Any ***symbol references*** you previously **EXPLODE**d to their base entities will not be updated. ***Exploding*** disassociates the entities from the ***original symbol definition***.

***Note: References to symbol definitions that are nested under a different definition will not be affected by* REPLACE SYM REF**.

Text equivalent: ***REPSYM***

Edit Symbol Definition

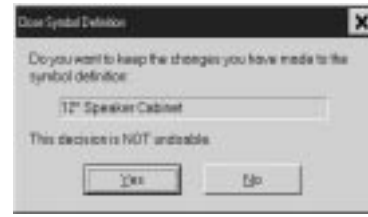


The* EDIT SYMBOL DEF *command lets you select any symbol defined* in *your drawing and edit it. Once the ***symbol definition*** has been edited, ***all references*** pointing to it use the updated ***definition*** and redraw accordingly.

When you invoke this command, **FastCAD** displays the ***Select Symbol*** *dialog box*. Select the ***symbol definition*** you wish to modify.

FastCAD displays the ***symbol in a full size window*** on top of the drawing window with the ***symbol name*** in its title bar.

You can now perform regular ***draw*** and ***edit*** operations as if this were a normal drawing. When you are done modifying the ***symbol definition***, choose the ***Close*** *icon* or select **[File > Close]** command from the menu..



FastCAD displays an alert box as seen above that asks if you want to keep the changes you made. If you click **Yes**, the new ***symbol definition*** will be saved within the drawing, and ***symbol references*** pointing to it will be updated. If you click **No**, then your edits will be ignored, and the ***symbol*** window will ***close***.

Note that **UNDO** will **NOT**work on this command. If you accept a modified definition, you cannot take it back.

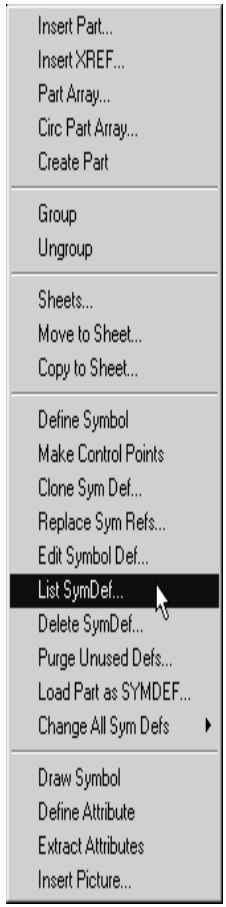
Editing Symbol Definitions Color

You can use the **EDIT SYMBOL DEF** to change colors of a specific ***symbol definition.*** However, **FastCAD** ***provides commands to change the color for all symbol definitions in a drawing:***

- **CHANGE ALL SYMDEFS > CHANGE ALL COLOR** asks for a **color**, and ***changes all entities in all symbol definitions to that color***. ***All symbols in the active drawing will now have a single color.***
- **CHANGE ALL SYMDEFS > CHANGE ALL COLOR By BLOCK** ***changes all entities in all symbol definitions to*** **Color by Block**. Every ***symbol*** will be drawn with the color assigned to its ***symbol reference***. For new ***symbol*** insertions, this would be the current color. For existing ***symbol references***, this would be the color current when the ***symbol*** was inserted.

These commands can be big timesavers when modifying an entire ***symbol catalog***. Without them, you would have to use **EDIT SYMBOL DEF** on each ***symbol definition*** and then apply the **EDIT > CHANGE > COLOR** command.

Text equivalent: ***EDSYMDEF***



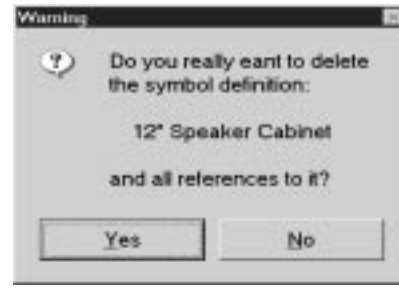
List SymDef

LIST SYMDEF ***provides information about symbol definitions available in the current drawing.*** A dialog box identical to the ***Select Symbol*** *dialog box* is displayed. Select a ***symbol*** and the component entity information is displayed in a text window.

Text equivalent: ***LISTSYM***

Delete SymDef

DELETE SYMDEF ***deletes an individual symbol definition***. If the ***active drawing*** contains ***symbol references*** to the ***deleted definition***, the ***references*** will also be deleted.



FastCAD will warn you before proceeding with the alert box as seen above.

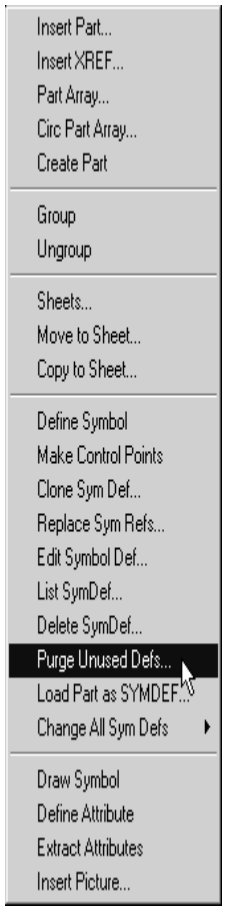
This command displays the ***Select Symbol*** *dialog box*. It is identical to the ***Select Symbol*** *dialog box* except that selection ***deletes the symbol definition*** and ***all references*** to it.

Use **UNDO** to reverse the effects of this command.

For more help understanding s***ymbol definitions*** and ***symbol references***, please see **Symbol** commands.

Text equivalent: ***DELSYM***

Purge Unused Symbol Definitions



The* PURGE UNUSED DEFS *command removes all unused symbol definitions from the current drawing. An unused symbol definition is one that does not have any corresponding symbol references.

PURGE UNUSED DEFS ***does not affect symbol references or referenced definitions in any way.***

This command is helpful for maintaining an efficient drawing file when ***symbols*** are frequently used. Unused ***symbol definitions*** sometimes propagate and accumulate, often inadvertently, possibly hindering **FastCAD**'s performance and inflating file sizes. For instance, if you insert a ***part*** that contains ***500 symbols and make use of only one of those symbols***, ***the other 499 definitions remain as*** "**excess baggage**" in the drawing file! The **PURGE UNUSED DEFS** command will easily remove them from the drawing in one step.

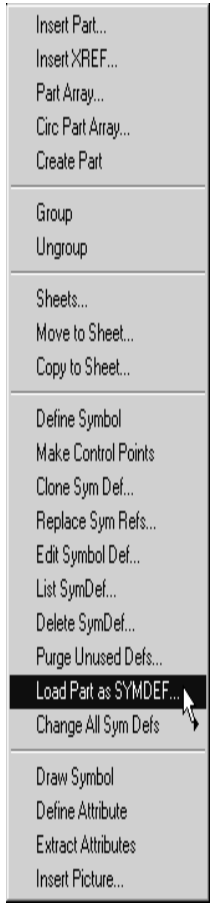
To use the **PURGE UNUSED DEFS** command, select **[Insert > Purge Unused Defs]**, or type **PURGESYM** at the command line.

Note: **UNDO** will **NOT** work with this command; so be sure that you can reintroduce or re-create any un-referenced symbol definitions that you might again need in the future.

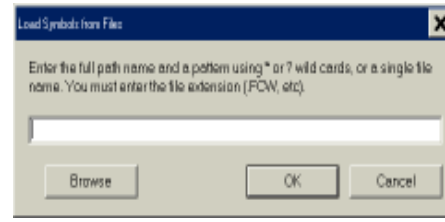
For more help understanding ***symbol definitions*** and ***references***, please see ***Symbol*** commands.

Text equivalent: ***PURGESYM***

Load Part as SYMDEF..



The* LOAD PART AS SYMDEF *command lets you load a drawing or part file as a symbol definition. Use this command to quickly create **FS7** ***symbol catalog files*** from stored ***part*** drawings.



Load Symbols from Files dialog box

When you start the command, **FastCAD** prompts you from the ***Load Symbols from Files*** dialog to enter a full path name and filename pattern, or a single filename. The use of the ***** or **?** ***wildcards*** are supported. For instance, "**\mylib*.fc7**" will load all files in the \mylib directory. The appropriate filename extension must be entered if you are loading non-**FC7** files (**FastCAD** will convert the files "**on the fly**"). ***If no extension is specified*, FC7 *is assumed*. *If no path is specified, then the* FastCAD *working directory is assumed.***

Brouse button

If you click the ***Brouse*** button in the ***Load Symbols from Files*** dialog, the ***Brouse for Folder*** dialog will open alowing you to visually select a folder that contains the drawings you want to load as ***symbols***.When you click a fonder it will appear in the ***Load Symbols from Files*** dialog with the default ***.FC7** file extension.If you click **OK** the entire folder of **.FC7** files will be loaded and converted to ***symbols*** in the ***active drawing*** file.You can use wildcard characters instead or the ***** character if desired.

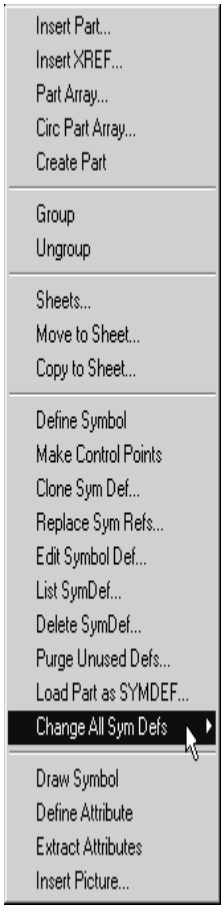
When a ***part file*** is loaded, it is loaded as a ***symbol definition***. It is not loaded as a visible ***reference***. ***Each part's filename becomes the symbol name, and its*** (**0,0**) ***point is the symbol origin or insertion point handle***.

To see or verify the loaded ***symdef***s, use the **LIST SYMBOL DEF** command **[Insert > List Symdef]**. Iif the ***Symbol Catalog*** viewer is on, click the ***Dwg Syms*** *button* below the ***Catalog*** *button* to view the ***Symbols*** you just loaded into the ***active drawing***.

Wildcards can only be used for the filename, not the directory name. If you are loading multiple parts from different directories then **LOAD PART AS SYMDEF** must be run once for each directory.

This command can be run directly from a ***macro*** or ***script***. The ***path\name*** is taken directly from the code; the dialog display is suppressed.

Text equivalent: ***LDSYMDEF***



Insert > Change All SymDefs > Scale All Defs

INSERT > CHANGE ALL SYMDEFS > SCALE ALL DEFS ***allows you to scale all symbol definitions in the current drawing by a specified factor. The definitions are scaled about their origin.***

Existing ***symbol references*** that point to the ***definition*** are not rescaled, but they will reflect the scale changes made to their respective ***definitions***. Existing attributes are not scaled; that is, the existing ***symbol reference*** attribute text height and location do not readjust. ***References*** inserted after **CHANGESS** ***will properly locate and scale***.

This command can be helpful in symbol catalog maintenance. For example, you may have a ***symbol catalog*** containing 150 schematic symbols. However, in order to be useable in the units you draw, they need to be scaled up 2x. Rather than using **EDIT SYMBOL DEF** 150 times for each symbol, apply the **CHANGESS** scale once (**using a factor of 2.0**), and **SAVE AS** the catalog file.

To change the ***symbol scale***, type **CHANGESS** at the command line or select **[Insert . Change All SymDefs > Scale All Defs]**:

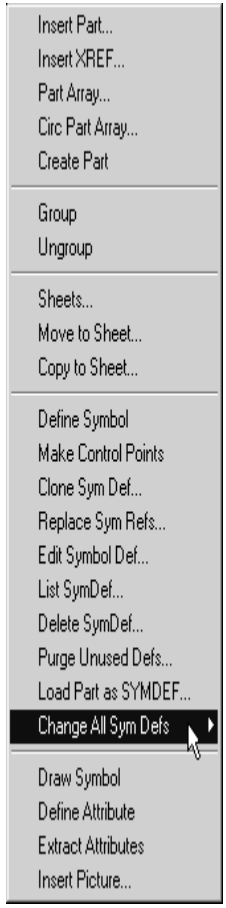
. ***The symbol definitions update and the command terminates.***

This command is useful for managing **FS7** ***symbol catalog files***. To change the colors in a catalog, you must first load the catalog file itself as a drawing, then apply the command. **CHANGESCBB** only affects the ***symbol definitions*** contained in a drawing, not those in the catalog displayed in the viewer.

Use **UNDO** to reverse this command.

Text equivalent: ***CHANGESS***

Insert > Change All SymDefs > Change All Color



INSERT > CHANGE ALL SYMDEFS > CHANGE ALL COLOR ***allows you to change all symbol definitions in a drawing to a selected color. Any references in the drawing will automatically update to reflect the new color.***

To change all symbol definitions color, type "**CHANGESC**" at the command line or select **[Insert > Change All SymDefs > Change All Color]**

> Select a ***new color*** by choosing a ***Color*** *bar* or palette color, or typing in a color number.

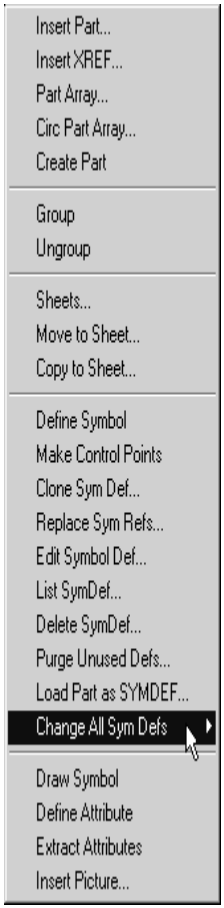
This command is useful for managing **FS7** ***symbol catalog files***. To change the colors in a catalog, you must first load the catalog file itself as a drawing, then apply the command. **CHANGESC** only affects the symbol definitions contained in a drawing, not those in the catalog displayed in the viewer.

To see which symbols are contained in the drawing, and which are in the catalog, press the **Catalog...** or ***Dwg Syms*** *button* at the top of the viewer.

- When **Catalog…** is on, the catalog symbols are displayed. These are not affected by **CHANGESC**.
- When **Dwg Syms** is on (**depressed**), the symbol definitions in the drawing are displayed. These will be affected by **CHANGESC**.

Use **UNDO** to reverse this command.

Text equivalent: ***CHANGESC***



Insert > Change All SymDefs > Change All Color by Block

INSERT > CHANGE ALL SYMDEFS > CHANGE ALL COLOR BY BLOCK ***allows you to change all symbol definitions in a drawing to Color by Block. Any references in the drawing will automatically update to reflect the*** **By Block color**. ***References already in the drawing will change to the color current during insertion*** (this can be determined by using the **LIST** command). ***Symbol references that you add will use the current color.***

If you want to change only a single definition to use By Block color, use the **CHANGECBB** command **[Edit > Change > to Color of SymRef]**.

Note that this command is not in any standard menu. You must type "**CHANGESCBB**" at the command line prompt to invoke the command.

To change all drawing symbol definitions to color by block, type "**CHANGESCBB**" at the command line or select **[Insert > Change All SymDefs > Change All Color by Block]:**

. ***The symbol definitions update and the command terminates.***

This command is useful for managing **FS7** ***symbol catalog files***. To change the colors in a catalog, ***you must first load the catalog file itself as a drawing***, then apply the command. **CHANGESCBB** ***only affects the symbol definitions contained in a drawing, not those in the catalog displayed in the viewer.***

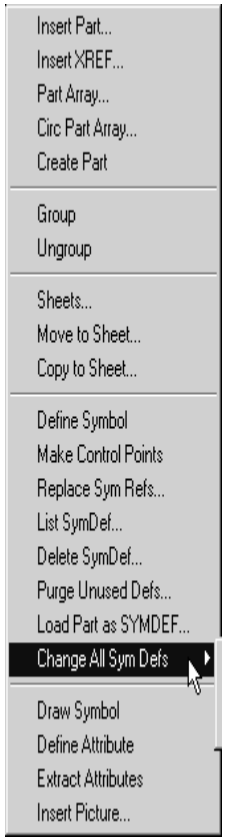
To see which ***symbols*** are contained in the drawing, and which are in the catalog, press the ***Catalog*** or ***Dwg Syms*** *button* at the top of the ***viewer***.

- When ***Catalog…*** is on (**depressed**), the ***catalog symbols*** in the external **.FS7** file are displayed. These are not affected by **CHANGE ALL COLOR BY BLOCK.**.
- When ***Dwg Syms*** is on (**depressed**), the ***symbol definitions*** in the ***active drawing*** are displayed. These will be affected by **CHANGE ALL COLOR BY BLOCK**.

Use **UNDO** reverse this command.

Text equivalent: ***CHANGESCBB***

Insert > Change All SymDefs > Change All Layers



The* CHANGE ALL LAYER *command allows you to change the* LAYER *of* ALL ENTITIES IN ALL SYMBOL DEFINITIONS *in the active drawing to a* DIFFERENT LAYER *that you select.

ALL ENTITIES *in* ALL SYMBOL REFERENCES *in the drawing will* AUTOMATICALLY UPDATE *to reflect the* NEW LAYER *assignments.*

This means that if you **HIDE** *the* **LAYER** *that you select with this command,* **ALL SYMBOL REFERENCES *will disappear from the drawing screen.***

The previously inserted **SYMBOL REFERENCES *will remain on the* LAYER *that was current at the time that they were inserted.***

Using this command on your **.FS7 SYMBOL CATALOG FILES i**s a good way to keep the **SYMBOL REFERENCES** behaving in a consistent manner after inserting them into a drawing from the **SYMBOL CATALOG VIEWER.**

NOTE: You should NOT use this command if you purposely have entities on DIFFERENT LAYERS in the SYMBOL DEFINITIONS that you want to selectively HIDE and SHOW.

1. To change the **LAYER of ALL ENTITIES in ALL SYMBOL DEFINITIONS in the current drawing on your screen,** select the **[Insert** > **Change All Sym Defs** > **Change All Layer]** command**.**

   The prompt reads: **'Layer name or # [dialog]:'**

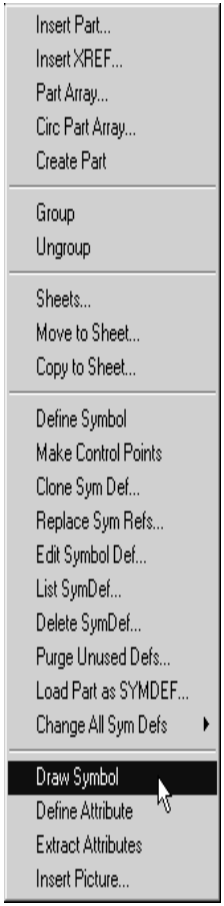
2. Right-click to display the ***'Layer Management' dialog box*** and left-click on the desired **LAYER NAME.**

Now **ALL of the ENTITIES in ALL SYMBOL DEFINITIONS** that you have inserted into the drawing on your screen will be associated with the **LAYER** you select.

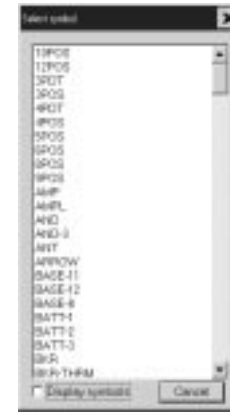
As long as that **LAYER** is **VISIBLE,** as well as the **LAYER** each **SYMBOL REFERENCE** resides on, the entities in the **SYMBOL REFERENCES** will be **VISIBLE.**

Text equivalent: ***CHANGESLYR***

Draw Symbol

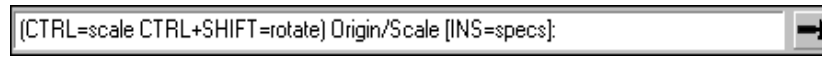


DRAW SYMBOL ***adds a symbol reference to your drawing. A symbol reference is a ghost image of a symbol definition that you have stored in the invisible symbol area of your drawing.*** Invoking the command displays the ***Select Symbol*** *dialog box*:



The illustrated ***Select Symbol*** *dialogs* display identical information in different ways. The dialog on the left has the **Display symbols** option switched **ON** (**located at the bottom of the dialog**). This graphical display mode can only display seven ***symbol*** thumbnails per page (**use the vertical scroll bar to see more or the scroll whele on a mouse**). The dialog on the right has **Display symbols** switched **OFF**. Only ***symbol names*** are displayed. In list mode, twenty-four names can be displayed per page.

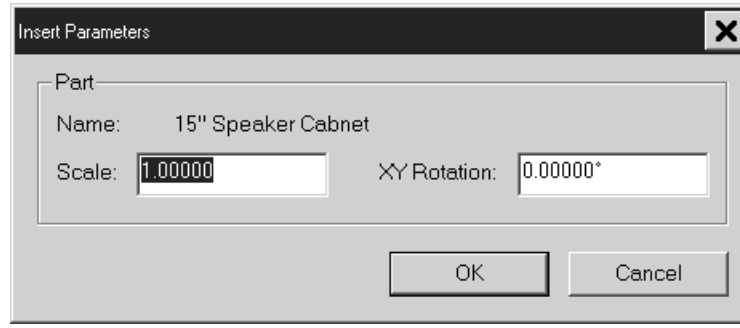
To ***insert*** a ***symbol***, select a ***symbol*** from the list. **FastCAD** displays a ***ghost image*** of the ***symbol***. You can ***dynamically scale*** or ***resize*** the image before positioning when the prompt reads "**(CTRL=scale CTRL+SHIFT=rotate) Origin/Scale [INS=specs]**".



- To ***scale*** the ***symbol***, type a scale factor like "**2**" or "**.5**".
- To ***dynamically scale*** the ***symbol***, press **CTRL** and move the mouse..
- To ***dynamically rotate*** the ***symbol***, press **CTRL+SHIFT** and move the mouse
- Press the **INS** key on the keypad the ***Insert Parameters*** *dialog box.* Set the **Scale and rotation angle and click OK.** The ***symbols*** inserted from this point will use these values**.** The ***symbol name*** is also dispplayed in dialog box**.**

If you want to change the ***scale*** of the original you may immediately t*ype* ***a new scale factor*** such as "**2**" to make the ***symbol*** twice as large. If you press the **"Ins"** key on the keypad, **FastCAD** displays the ***Insert Parameters*** *dialog box* with the ***default Scale factor*** and ***XY Rotation Angle*** of the ***symbol.*** You may adjust both ***symbol scaling*** and ***rotation*** from this dialog box. All ***symbols*** inserted from this point will use these default values. The ***symbol name*** being inserted is also displayed in this dialog.

- The ***default scaling is 1.0*** (**same size as the original symbol**). Type in a ***new scale factor*** if you wish to insert the ***symbol*** at a ***larger*** or ***smaller*** size.
- The ***default XY rotation angle is 0.0°.*** If you wish to **rotate** the ***symbol*** upon insertion, ***type in a new rotation between 0 degrees and 360 degrees***. For example, type "**45**" to ***rotate the symbol 45 degrees counter-lockwise***.



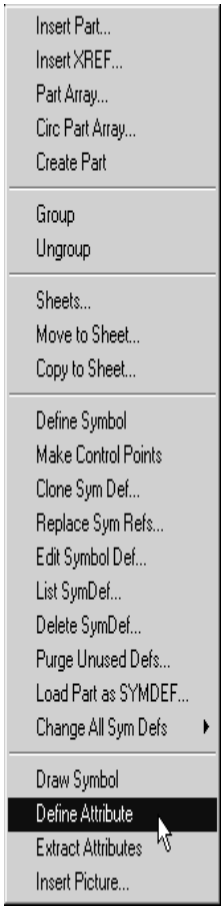
Insert Parameters dialog box

To insert the ***symbol reference***, pick a point in a drawing window. The ***insertion point*** of the ***symbol*** (**its 0,0 Origin**) aligns exactly with the point you pick. You may continue to insert the same ***symbol reference*** by picking new points, or right-click to end insertion.

Note*:** Although **FastCAD** doesn't have specific commands for drawing ***symbols in rectangular or circular arrays, you can do this very easily—just insert one ***symbol***, then select **Prior** for the **COPY** or **ARRAY** commands.

Text equivalent: ***INSSYMD***

Define Attribute



Attributes are pieces of information* (text labels) *you attach to symbols for later use. They act as ***symbol*** parameters or ***references*** that you might find useful for times when you are working with a database. ***Attribute*** definitions are contained in every ***symbol***.

Once you've drawn the entities that will eventually be your ***symbol***, you'll need to define the symbols and attributes to recall it for later use.

1. Select **[Insert > Define Attribute]**. The "**Text Attribute**" dialog box appears with three options to define the attribute:
   - Tag text is the attribute definition or the per instance name of your ***symbol***. You can change the tag or use its default before placing each symbol.
   - Prompt text is the constant name or ***definition*** of your ***symbol***. Once you enter prompt text, you cannot change it.
   - Default value is the value you give your ***symbols***. You can change the value or accept the default before placing each ***symbol***. Values can be the same for all of your ***symbol references***.
2. Review the three check boxes for defining attribute specifications. The checkboxes are:
   - **Hidden**: Checking this hides the text.
   - **Constant**: If this is checked, it means the values are unchanged.
   - **Display Tag:** Gives you the option to display the tag or value in your drawing.
3. Enter the **Tag text**, **Prompt text** and **Default value** for your symbol.
4. Select **Text Properties** if you want to change ***text attributes*** such as height, justification, etc.
5. Now select **[Insert > Define Symbol]** and type in a name for your ***symbol***.
6. Select an insertion point with the crosshairs and left button or by typing in coordinates.
7. Select **DRAW SYMBOL** and the ***Attribute Reference*** *dialog box* appears. Select the ***symbol*** you want to place. (**You can change the tag or value before placing each *symbol***)
8. Drag the ***symbol*** dynamically to place it.
9. Continue placing as many ***symbols*** as you like. Right-click to stop placing ***symbols***.

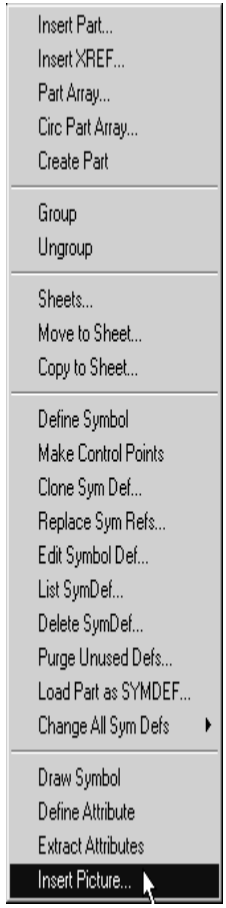
Text equivalent: ***ATTRIB***

Extract Attributes

EXTRACT ATTRIBUTES, like **LIST** in the ***Info*** *menu*, acts as a "**reveal codes**" function by displaying a list of all of your ***symbols*** and their attribute values.

Text equivalent: ***EXTRACT***

Insert Picture

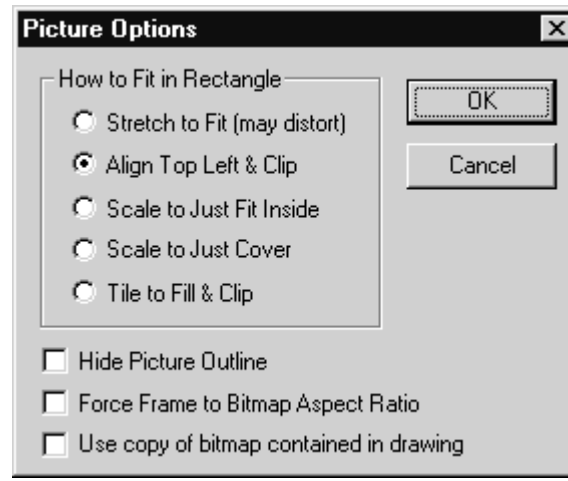


Use **INSERT PICTURE command** to display a ***bitmap image* .BMP file** in your drawing. The image will appear in the rectangle you draw on the screen (**just like using the BOX command**). To draw the box, when prompted left-click to pick each corner point in the drawing window, or type **numeric coordinates**.

> The ***current fill style*** has no bearing on the appearance of the ***inserted picture***. However, the image entity does acquire the current drawing properties. This may be a factor when selecting entities by ***color***, ***layer***, ***fill style***, etc.
>
> **FastCAD *does not embed the image into the drawing.*** Instead, a ***dynamic link to the actual* .BMP *file is maintained***. If you intend to ***copy*** or ***move*** an **.FC7 drawing containing images*, remember to also copy or move the* .BMP *files as well***. You can minimize conflict by ***copying*** or ***moving*** the **.BMP** files into the same folder as the drawing that will use them before you even insert it intoit**.** If a drawing is loaded and the **.BMP *image file does not exist*** or ***cannot be found,* FastCAD *displays an empty box*.**
>
> When you select this command, **FastCAD** displays the ***Select bitmap for picture*** *dialog box*, allowing you to select the desired **.BMP** file. **FastCAD** then displays the ***Picture Options dialog box***.

To **Insert a Picture** into your drawing type the **PICT** command **[Insert > Insert Picture]** or select it from the **INSERT** menu**.**

1. Select a **.BMP** file from the ***Select bitmap for picture*** *dialog box.* After you select the **.BMP** file, the ***Picture Options*** *dialog box* opens**.**



Picture Options dialog box

Stretch to Fit (may distort))

Frame drawn at exact image aspect ratio (not distorted)

Frame not drawn at exact image aspict ratio (distorted)

The option ***squezes*** the **.BMP** ***image vertically*** and ***horizontally*** so that it ***fills the picture frame.*** The image conforms to ***aspict ratio of the picture frame you draw.*** If you know the **exacr dimensions of the .BMP file, x" by y" inches,** you can draw the ***picture frame*** this ***exact size*** and the **image will not distort.** ***The picture frame is the proper aspect ratio***. Load the **.BMP** image into a program such as **Coral Photo Paint** and select the **PROPERTIES** command to see what the ***dimensions*** of the ***scanned image*** are**.**

After you have drawn the ***picture frame the exact size of the scanned image,*** you can use one of the **SCALE** commands to enlarge it. Remember you are displaying a ***bitmap image composed of dots*** so at some point it will look very blocky and be unusable. Experiment to see what works best for you. The images above were scaned at **72dpi** and **250 lpi**.

Align Top Left & Clip

The option ***aligns the*** **.BMP images upper left corner** to that of the ***picture frame*** and retains its original size**.** If the image is smaller than the ***picture frame***, the blank area will opaquely fill with the background color.

Using this option, you will draw the ***picture frame to fit the scaned image*** since the pixels are not being scaled to fit inside the ***frame.*** The higher the resolution the larger the image will be when viewed on the drawing screen.

EXAMPLE: If you scaned an area that is **4"x 4"** at **75 dpi**, the image would be **75 x 4 = 300 pixels by 300 pixels**. If your video resolution is **1024 pixels horizontally**, the image would ***cover about 1/3 of this horizontal screen area***. If the screen resolution is **768 pixes vertically,** the image would cover almost ***1/2 of the vertical screen area***. If you scanned the **4" x 4"** area at **300dpi,** the image would be **300 x 4 = 1200 pixels by 1200 pixels**. This means that a **1024x768 resolution screen** would be totally covered. The ***Align Top Left & Clip*** method would not work. ***The picture frame would be blank.*** Usually scanning at **75dpi** is sufficient for this type of application**.**

Scale to Just Fit Inside

The image ***scales*** itself to fit entirely within the ***picture frame*** without clipping. The image will maintain its original aspect ratio, regardless of that of the ***picture frame***. Any blank area in the box will appear transparent, and is only visible when **Hide Picture Outline** is **off**.

Scale to Just Cover

The option ***scales*** the image so there is no blank area in the ***picture frame.*** The image ***scales itself just large or small enough so that the entire picture frame*** is filled, clipping as needed. The image is centered in the box.

Tile to Fit and Clip

The image ***maintains its original size and aspect ratio***, and will repeat itself within the ***picture frame.*** as to fill it. The "**start**" image aligns to the top left corner. Bottom and right border images will clip as needed.

Hide Picture Outline

When you draw the ***picture frame*** it uses the current color in the ***active drawing***. If you check this box the ***picture frame*** will not appear on the screen.

2. Select a method for fitting the **.BMP** image into the ***picture frame*** and the prompt reads **"First corner:".** Left-click to place the first corner of the ***frame***

   The prompt reads **"Opposite corner:".** Left-click again to place the opposite corner of the ***picture frame***

Force Frame to Bitmap Aspect Ratio

Checking this box ensures that the image will not distort but ***it retains its original size as scanned*** and may be to large to view on screen. If the ***picture framw*** is blank, this is the case. You must use one of the methods that ***scales*** th image to the ***picture frame***.

Use copy of bitmap contained in drawing

The data contained in the **external .BMP file will become part of the drawings data**. This will increase the file size accordingly. If the **.BMP file is 200kb** and there is more than one instance of this same **.BMP** placed in the drawing, each placement **increases the drawing file size buy 200kb**.

Advantages of including a copy of the .BMP data in the drawing

The **ADVANTAGE** of storing the **.BMP** data in the drawing file makes it portable. No matter where the drawing gets moved to or copied to, the **.BMP image** will always appear in the Picture Frame.

Disadvantages of including a copy of the .BMP in the drawing

The **DISADVANTAGE** is that the **drawing file size is much larger when the .BMP data is stored in it** instead of just pointing to the **external .BMP file** and displaying the image in the **Picture Frame**. Another method is to always store the .**BMP files** on the ***same drive*** and ***in the same folder*** and the link to the **.BMP** can always be found as long as the drawing is on a **LAN** or on the same computer as the **.BMP** that it references. If you email a drawing with a .BMP image you should always use this method unless you also email the **.BMP file** separately and both it and the drawing that references it are stored on the ***same drive*** and ***the same folder name*** as the originals were.

Email attached file size limitations

If you email a drawing with a **.BMP** ***file as part of the drawing file data***, the file may be to large to attach to the email. Many internet providers have a **maximum email file attachment size limit of less than 3MB.** If the file you attach to the email is to large the email will be rejected. You should contact the person you are emailing the drawing to and determine what the **email file attachment size limit is** for their internet provider.

NOTE: If the **.BMP files** are **ERASED** or **MOVED** to another **DRIVE** or **FOLDER** only the **PICTURE FRAME** will be displayed in the drawing unless you use the '**Use copy of bitmap contained in drawing'** option.

Use the **[Edit>Edit]** command and select the edge of the **PICTURE FRAME** and select the **DRIVE** and **FOLDER** where the **.BMP file** is now located and it will be displayed again. If you send another **EasyCAD7/FastCADv7** user a drawing with a **.BMP** in it you will need to send them the **.BMP file** also. These **.BMP PICTURES DO NOT** convert to **AutoCAD .DWG** or **.DXF** files.

Text equivalent: ***PICT***

Chapter 15

Viewport Menu

Viewports are Inset views of specific areas in the drawing

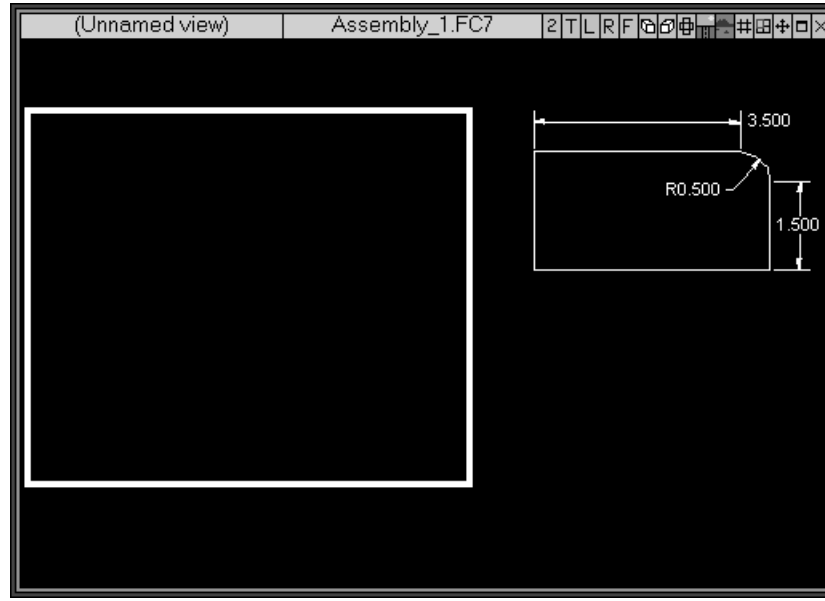
A Viewport is a **2d** entity that you draw (define) on the **2d** ***workplane.*** Its interior displays a specified ***view*** in the ***active drawing.*** A ***viewport*** allows you to show inset detail of **2d** and **3d** entities in the drawing. This ***view*** may be scaled up (magnified) without changing the displayed values of ***associative dimensions*** that are in the ***view***.

The ***viewport*** entity must be a **2d** ***closed*** entity such as a ***box, polygon, circle or spoly.***

Viewports > Create

To **create** a **viewport;**

1. Draw the **2d** ***box, polygon, circle or spoly*** that will be the ***viewport entity***.



In the illustration above, the large ***polygon*** with the thicher outline is the ***viewport entity***. Its outline has a ***pen thickness*** applied to it so it is easily distinguishable from the other entities in the drawing.

2. Select the **[Viewports > Create]** command.

Tthe prompt reads Select entities (H=help) [0 selected]:

Left-click on the ***viewport entity***.and the ***Viewport Specification*** *dialog* opens.

Click the **OK** button to close the dialog box.

The ***Viewport Specification*** *dialog* is discused later in this chapter.

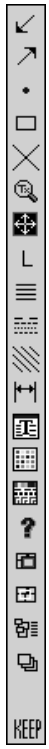
Text equivalent: ***VPORT***

Viewports > Copy window view

COPY WINDOW VIEW copies the selected view, set up in a ***new drawing window,*** to the ***viewport*** entity. To set the desired view you must select the ***Make New Drawing Window*** *icon* and pick two diagonal corners for the ***new drawing window***.

You then zoom in that window to set up the view you want to be displayed in the ***viewport entity.*** Then use the **COPY WINDOW VIEW** command to select the ***viewport entity*** that will display the active drawing window's view.

After you have used this command you may close the ***drawing window*** you created by clicking the **X** in its upper-right corner. If you want to change the view for that ***viewport*** later, add another ***new drawing window***, adjust the desired view, and use this command again



New Drawing Window

Standard Icons bar

To **copy** a **window view** to a ***viewport entity***;

1. Select the the ***New Drawing Window*** *icon* from the ***Standard Icons*** *bar.*

The prompt reads First corner:

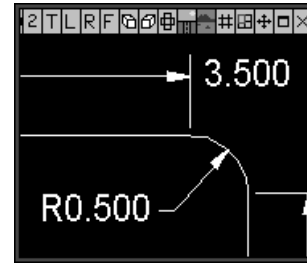
2. Left-click to place the ***1st corner*** of the ***new drawing window***

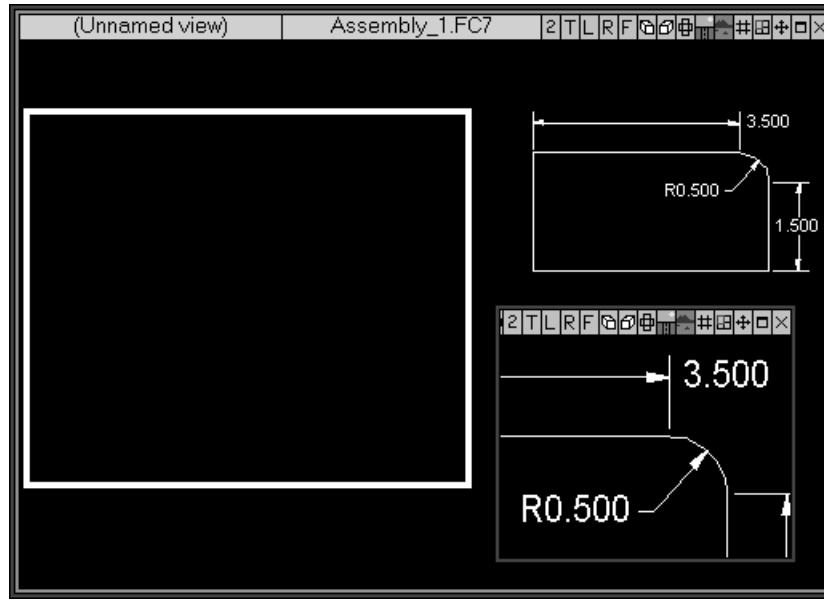
The prompt reads Opposite corner:

3. Left-click to place the ***opposite corner*** of the ***new drawing window***

You now have a ***new drawing window*** open on top of your drawing The view in this window will initally be zoomed to extents. You may maximize this window and use any zoom method to get the view exactly as you want, then minimize it.

The ***new drawing window*** as seen below was drawn in the lower-left corner of the screen as seen in the illustration on the next page. ***Dynamic Pan and Zoom*** was used to get the ***view*** as you see it.

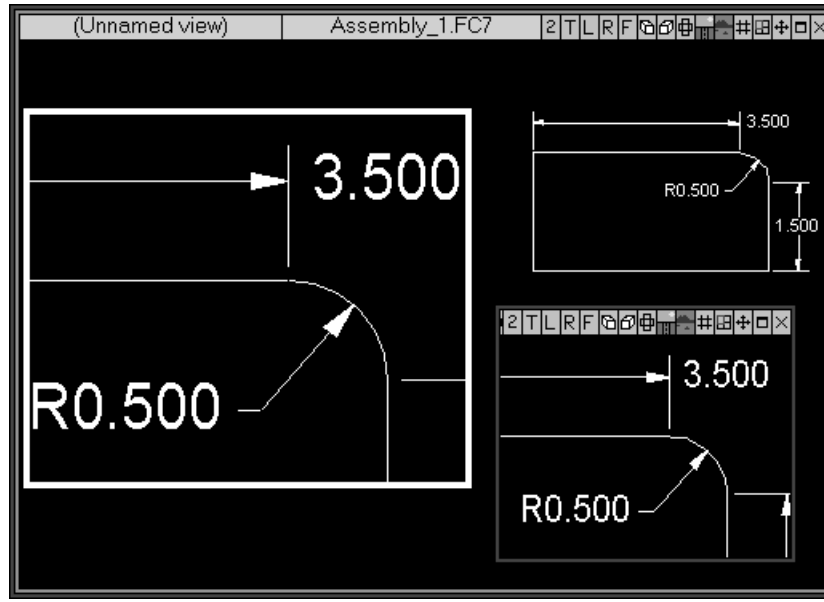




4. Select the **[Viewports > Copy window view]** command.

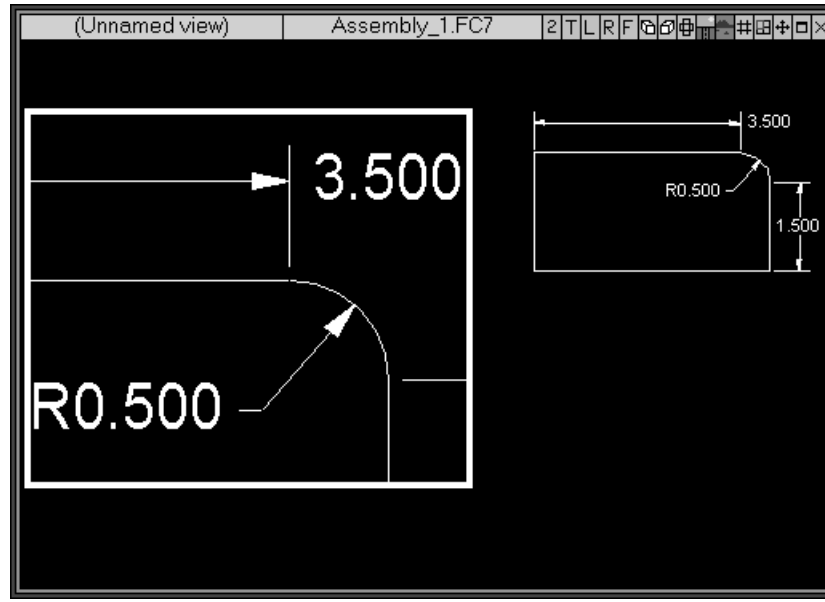
The prompt reads Viewport entity:

Left-click on the previously ***created viewport entity*** and the ***view*** is displayed inside. The ***view*** in the large ***polygon*** is the ***view*** in the ***window*** at the lower-left.corner.



After the ***view*** has been ***copied*** to the ***viewport*** from the ***drawing window*** used to set it, you may close the ***drawing window*** by clicking the **X** icon in its upper-left corner.

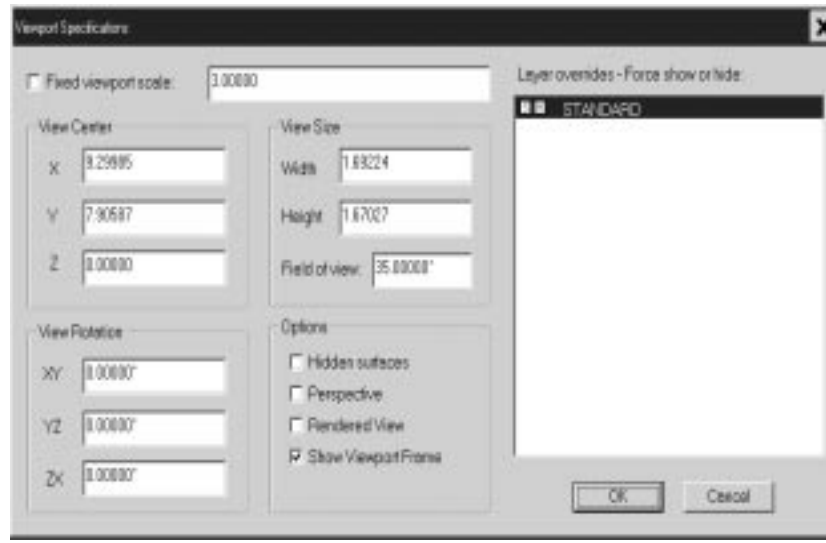
The illustration below shows the drawing screen after the ***drawing window*** used to set the desired ***view*** was closed.



Text equivelant ***VPCOPYW***

Viewport Specifications Dialog

The ***Viewport Specification*** *dialog* allows you to set various options for each ***viewport*** you ***create***.



Fixed Viewport Scale

This box must be checked if you want the entities in the ***viewport*** to ***scale*** by the value displayed in the window to its right. If this box is not checked the entities displayed in the ***viewport*** will use the view ***center***, ***width*** and ***hieght***. set from the ***drawing window*** you added earlier.

The Scale Value window

If the ***Fixed Viewport Scale*** *box* is checked, the value in this window will determine the sixe of the entities in the ***viewport***. If the value is **0.25**, the entities will be one fourth the actual size when viewed and ***printed***. The entities will ***scale*** about the original ***view center*** when it was copied from the ***drawing window*** using the **[Viewports > Copy window view]** command.

View Center and View Size

The **XYZ** coordinates of the ***view center*** may be adjusted in these windows If you use the **[info > Coordinate]** command and pick an exact point in the drawing using a ***modifier*** such as ***Enpoint of seg.***, you can type these coordinates into this area to change the center the ***viewport*** precisely to that location.

View Rotation

The **XY, YZ,** and **ZX** ***views rotation*** angles in **3d** space may be adjusted in this area.

View Size

This area allows the ***view height*** and ***width*** to be adjusted as well as the ***Field of view angle*** in **3d** space.

Hidden Surfaces, Perspective, Rendered View and Show Viewport Frame checkboxes

These four checkboxes ***enable*** or ***disable*** the display of ***Hidden Surfaces, Perspective, Rendered view****i* and ***Show Viewport Frame***.

Layer Overrides - Force Show or Hide

This window displays a list of ***layers*** *in the* ***active drawing***. You may **override** the ***hidding*** or ***showing*** os entities in the ***viewport.*** Click the "**S**" icon on a ***layer*** to make the entities visible in the ***viewport*** even if that ***layer*** is ***hidden*** in the ***active drawing***. Clicking the "**H**" icon ***hides*** the entities in the ***viewport*** even if they are visible in the ***active drawing.***

Practice by creating a few new ***layers*** with entities on each. Draw a ***polygon*** *or* ***circle*** and ***create*** some ***viewports. Hide*** **and** ***show*** each ***layer*** to see how this feature works..

NOTE: ***Viewport entities must be 2d closed entities residing on the 2d XY plane but they may show 3d entities in them of a view in 3d space. The viewport entity may have a different color background*** **(interior)** ***color. Use the*** **[Edit > Change > Fill (2nd) color]** ***command to change it Use a colot that makes the detail stound out on the drawing.***

Text equivalent: ***CPCOPYW***

Viewport Background Color

In a new drawing the **background color** is **Black (color #0)**. The drawing's background color never prints but the ***viewport's background color always prints*** unless you select the "**Print everything black**" checkbox in the ***Print Drawing*** *dialog box.* With this box checked the ***viewport*** **background** is not printed while the entities displayed in it are printed **black.**

But the ***viewport*** entities **background color** is independent of the bachground color of the rest of the drawing and can be set for the desired printing results.

Changing a Viewport's Background Color

Use the **[Edit > Change > Fill (2nd) color]** command to change the interior **background color** of a ***viewport*** **entity**. Choose the command and select one or more ***viewport*** entities, right-click for the popup menu and left-click the "**Do it"** command and left-click the color block of the desired color you wish to change it to.

NOTE: To print all of the entities in the drawing and in the *viewport* black, check the "Print everything black"checkbox in the *Print Drawing dialog box.* With this box checked the *viewport* background is not printed while the entities displayed in it are printed black. If this box is NOT checked, color printing will be performed and the color of the interior of the *viewport* will be printed as shown.

Click the "Preview" button before printing the drawing to see how the *viewports* will be printed. This will save you time, ink and paper.

Viewport Tips

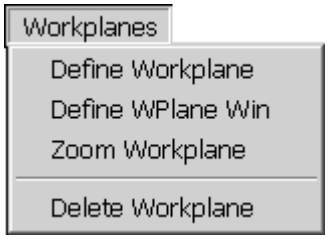
1. Create a separate **Layer** and draw your ***viewport*** entities on it. This way you can keep them separated from the other entities in the drawing.This allows you to **show** or **hide** the **layer** that contains them.

2. Always click the "**Preview**" button in the ***Print Drawing dialog box*** before printing the drawing to see how the ***viewports*** will be printed. This will save you time, ink and paper.

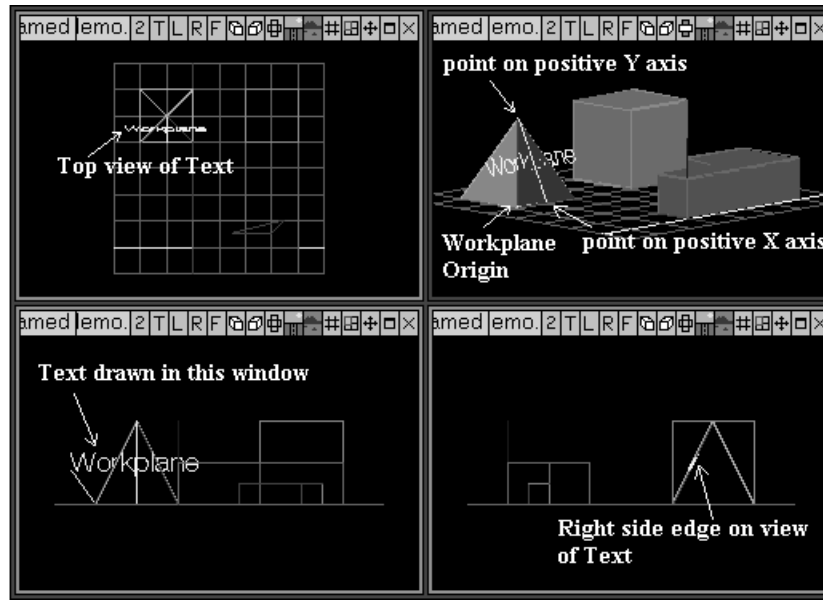
3. You can keep the same ***drawing window*** you used when setting the view you copied to the ***viewport*** using the **[Viewports > Copy window view]** command. The smaller drawing window will be on top of the larger main drawing window. When you click the "**Make active window"** icon of the larger drawing window, the smaller window will seem to disappear but it is now behind the main drawing window. To use it again hold the **CTRL** key and at the same time press the **TAB** key and the smaller window will appear on top of the larger window. You may zoom to reset the view in this smaller window and use the **[Viewports > Copy window view]** command.and click a ***viewport*** entity. Click the "**Make active window"** icon of the larger drawing window again and the smaller window will disappear behind the main drawing window for use later.

Workplanes Menu

Create, Zoom To and Delete Workplanes (work surfaces) in 3d space to draw new entities on.



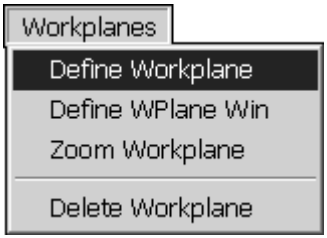
WORKPLANES are invisible flat areas in 3d space that you define by picking an Origin Point, a Point in Positive X from that Origin point and a third point in the Positive Y from this Origin Point. Drawing on a *workplane* makes it much easier to add entities to your drawing in 3d space. Use the ZOOM WORKPLANE command to place surface of the active drawing window flat against a Workplane. You can draw 2d or 3d entities on the *workplanes*.Open the sample 3d drawing WPDEMO.FC7 from the \FastCAD7\DWGS\ folder to see the *workplane* with a piece of Text on the face of the pyramid.



IMPORTANT: Becoming proficient at defining *workplanes* and workplane windows is essential for drawing in 3d space.

Note: Only vector font, such as FastCAD .FNT or AutoCAD® .SHX, may be used when drawing Text or Dimensions on 3d *workplanes*. Windows® TrueType® fonts may not be used.

Define Workplane



The **DEFINE WORKPLINE c**ommand asks you to name a workplane on the surface of a 3d entity that you have drawn. The ***active drawing window*** may then be placed flat against this ***workplane***, using the ***Zoom 3d Workplane icon,*** **a**nd drawn on as if it was a **2d** drawing. This greatly simplifies drawing in **3d** space.

Both 2d and 3d entities may be drawn on a Workplane.

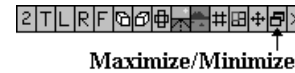
After zooming to the surface of the ***named workplane***, you may draw both **2d** and **3d** entities.

For instance, you may draw a **2d** polygon on the surface and use the **EXTRUDE** command to give it ***depth*** in the ***Z axis.*** You may do the same with a **2d** circle and **EXTRUDE** it into a ***cylinder.***

Define the Workplane by picking three points.

Three points must be picked in **3d** space to determine a planer surface ***(*****Workplane).** **FastCAD** prompts for the ***Workplane origin point,*** a ***second point in the*** **positive X direction from this origin** and a ***third point in the*** **positive Y direction** ***from this origin point.*** This method allows it to be treated as a **2d** drawing even thought it is in **3d space.**

Tip: Picking the ***workplane origin point***, ***point in positive* X *axis*** and ***point in positive* Y *axis***, *is* easiest in the upper-right view window in **Std. 3d Views** screen configuration. Click its ***Maximize/Minimize icon*** to make the view full screen. Pick the points using ***modifiers*** such as ***endpoint of seg***. Then click the ***Maximize/Minimize icon*** again to return to the four window screen.



To define a **3d workplane** select the **[Workplanes > Define Workplane]** command.

1. The prompt reads "**New workplane name [done]**.".

   Type a ***descriptive name*** for the ***workplane,*** or right-click for ***no name.***

2. The prompt now reads "**Workplane origin**".

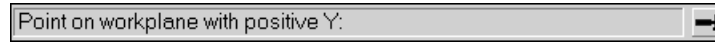
   If you were ***defining one surface of a cube pick it's lower-left corner***, using the '**Endpoint of seg**.' (**F5**) ***modifier*** for accuracy, as if this surface was a ***2d drawing***. This will be the workplane's (**0,0**) ***origin point***. You may use any appropriate ***modifier*** to pick these points.

   You should pick this point in ***the upper-right obligue view window***

3. The prompt now reads "**Point on workplane positive X**".

Pick this point ***to the right from the origin point*** (**positive X direction from 0,0**) ***of the workplane***. You may use a ***modifier*** such as ***endpoint of seg*** **(F5)** or ***midpoint of seg*. (F3)** to pick these points.

4. The prompt now reads "**Point on workplane with positive Y**".



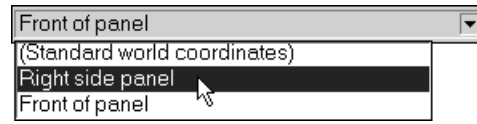
Pick this point ***up vertically from the origin point*** (**positive Y direction from 0,0**) ***of the workplane.***. You may use a ***modifier*** such as ***endpoint of seg*** **(F5)** or ***midpoint of seg.*** **(F3)** to pick these points

Zooming to a Workplane Surface

The current workplane name is displayed in the workplane status window on the status bar. Click the drop-arrow at the right end of the status window to select a different workplane from the drop-down menu.

Current Wirkplane Status window

Workplane drop-down menu



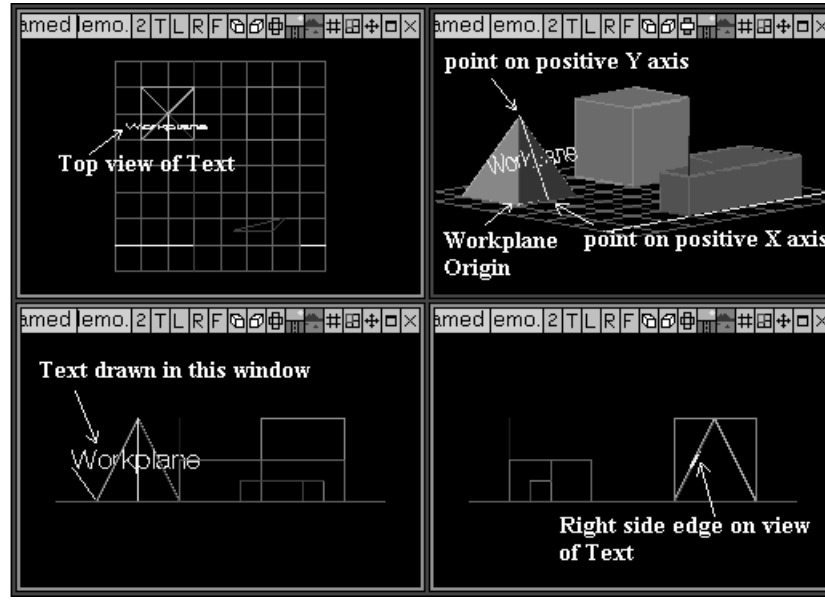
Example:

In the drawing below, the ***lower-left view window*** is the ***active workplane*** making it the drawing surface ***as if it was a 2d drawing***. Notice the **2d** text **"Workplane Window"** is drawn flat on the ***lower-left window surface***. It appears in the ***upper-right overall view*** angled in ***3d space on the front surface of the pyromid.***. The ***lower-right window*** shows it ***edge-on*** viewed from the ***right side.*** The text is angled in the ***upper-left top view window 3d space***.

After the ***workplane*** was defined, the ***Make active window icon,*** as seen below, was clicked in the ***lower-left view window making it active for drawing***.

Next the ***Zoom 3d Workplane icon*** was clicked to place the ***workplane*** (**the front face of the pyramid**) ***flat against the lower-left drawing window surface***. The text was then drawn and is displayed in all four view windows simultaneously

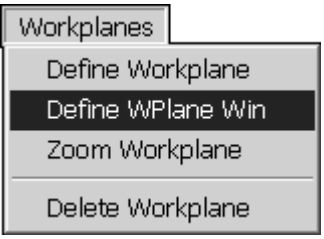
I**MPORTANT:** Becoming proficient at ***defining workplanes*** and ***workplane windows*** is essential for drawing in **3d** space.



Note: Only ***vector fonts***, such as **FastCAD** .**FNT** or **AutoCAD® .SHX**, may be use when drawing ***Text*** or ***Dimensions*** on 3d workplanes. **Windows® TrueType®** fonts may not be used.

Text equivalent: ***WPDEF***

Define WPlane Win



The **DEFINE WPLINE WIN** (**Define Workplane Window**) *c*ommand asks you to name a ***workplane.*** It then prompts you for the ***Workplane Origin***. You then click on a drawing window and that window is now the active workplane. Any entities you draw will be on that ***workplane***. Make sure that the window you click in is at the location that you want to draw on in **3d** space. This is a quick way to set up a ***workplane*** if you know the a particular drawing window is where you want to draw.

This command differs from **DEFINE WORK PLANE** in that you are not prompted for the ***Point on positive X axis*** and ***Poinr on positive Y axis*** to define the ***workplane***. **DEFINE WPlPLANE WIN** automatically uses the window you click in for these **X,Y** directions relative to the ***workplane origin***.

Both 2d and 3d entities may be drawn on a Workplane.

After zooming to the surface of the named workplane, you may draw both **2d** and **3d** entities.

For instance, you may draw a **2d** polygon on the surface and use the **EXTRUDE** command to give it depth in the **Z *axis***. You may do the same with a **2d** circle and **EXTRUDE** it into a cylinder.

To **define a 3d workplane window,** select the **[Specs > Define WPlane WIn]** command.

1. The prompt reads "**New workplane name [done]**.".

Type a ***descriptive name*** for the ***workplane,*** or right-click for ***no name*.**

2. The prompt now reads "**Workplane origin**".

Click in a drawing window that you want to be the workplane.

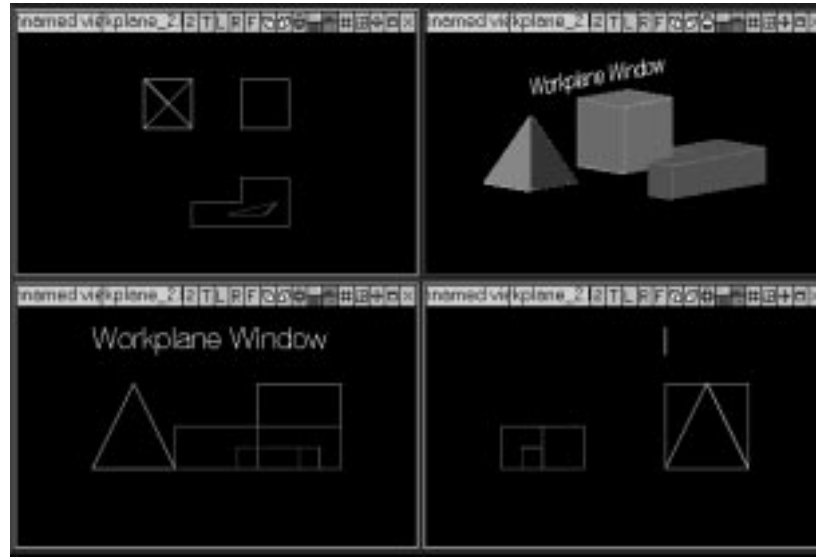
If you were defining one surface of a cube pick it's lower-left corner as if this surface was a **2d** drawing. This would be the **workplane's (0,0) origin** If it was **2d**. You may use a ***modifier*** such as ***endpoint of seg*** **(F5)** or ***midpoint of seg*. (F3)** to pick this ***origin point***.

Example:

In the drawing below, the ***lower-left view window*** was picked as the ***workplane origin immediately making it the drawing surface as if it was a 2d drawing***. This eliminates the step of clicking the ***Zoom 3d Workplane icon*** to move the ***defined workplane*** to the surface of the drawing window.

Notice the **2d** text **"Workplane Window"** is drawn flat on the ***lower-left window surface***. It appears in the ***upper-right overall view*** angled in ***3d space***. The ***lower-right window*** shows it edge-on viewed from the ***right side.*** The text is not visible in the ***upper-left top view window*** because it is ***above the plane*** of that window in ***3d space***.

IMPORTANT: Becoming proficient at defining workplanes and workplane windows is essential for drawing in 3d space.



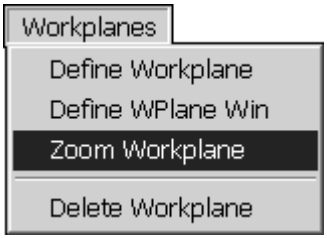
Note: Only ***vector font,*** such as **FastCAD** .**FNT** or **AutoCAD® .SHX**, may be use when drawing ***Text*** or ***Dimensions*** on **3d** ***workplanes***. **Windows®** T**rueType®** fonts may not be used.

Text equivalent: ***WPDEFW***

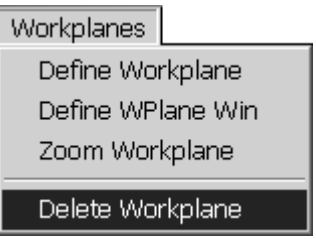
Zoom Workplane Icon

Zoom Workplane

The **ZOOM WORKPLANE command** moves the *active drawing window surface* flat against the selected ***workplane***. All new entities are drawn on this ***workplane*** surface.



Text equivalent: ***ZWPLANE***



Delete Workplane

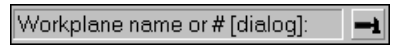
The **DELETE WORKPLANE** command allows you to type a ***Workplane name*** to remove from the active 3d drawing or right-click to open the ***Select Worlplane dialog box*** and click on the workplane name that you want to remove as seen below.

NOTE: Deleting a Workplane does NOT remove entities drawn on that Workplane.

To **Delete** a **Workplane** select the **[Workplanes > Delete Workplane]** command.

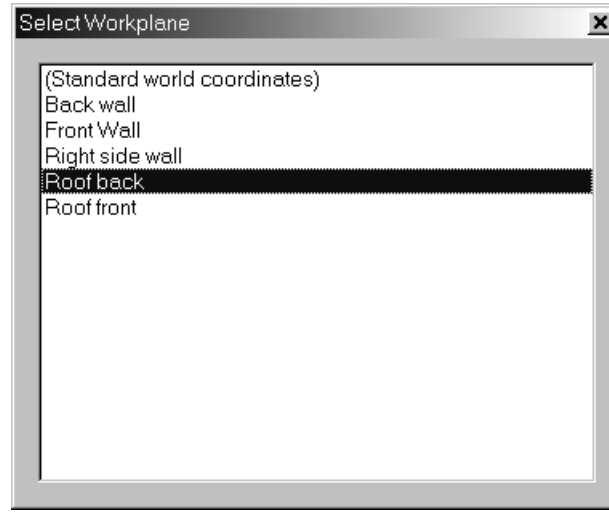
1. The prompt reads "**Workplane name or # [dialog]**".

.



Type a ***Workplane name*** and press **ENTER** or;

2 Right-click to display the ***Select Workplane dialog box*** and left-click on the ***Workplane name you want to delete.***



FastCAD removes the ***Workplane name*** from the available list. The entities that were drawn on this ***Workplane*** remain in the drawing.

Text equivalent: ***WPDEL***

Chapter 17

The Render Menu

Rendering: Working with FastCAD to create photorealistic images of 3D CAD models

Commands in the **Render menu** prepare your **3D** model for rendering. A rendered **3D** image is one that shows solid surfaces with light sources and shadows.

FastCAD v7 gives you the capability to specify the rendering program you wish to use. You can use the supplied **OpenGL** or **LightWorks** renderers, or a supported separate program. Rendering programs are supported through **XP** modules created by the rendering program vendor, or other third party developer.

Before using **Render menu** commands, you must use **FastCAD v7** to build a **3D** model using entities that have **enclosed surfaces**, like **polygons**, **spheres** and **solids**. Entities with no surface (like **points**, **lines**, **text**, and **splines**) don't appear in a rendered image.

Lighting is an important factor in the quality of **rendered images**. Carefully consider the type, placement, color, and intensity of light sources that illuminate your **3D** models. Start with just one or two light sources, and avoid over-lighting your models, as this will have a "flattening" effect, and diminish the **3D** appearance of objects in the scene.

When using the **OpenGL** renderer, ***wireframe object color controls the rendered surface color***. Objects like **spheres** and **cylinders** ***will appear solid***, ***smooth***, and ***shaded*** according to the **light placement**. **OpenGL** works very fast, and so offers a quick way of displaying **3D FastCAD v7** files to visualize shapes and relationships between objects.

In the future updates additional **Render applications** may be added to **FastCAD v7** for ***photorealistic textures*** but currently none are available.

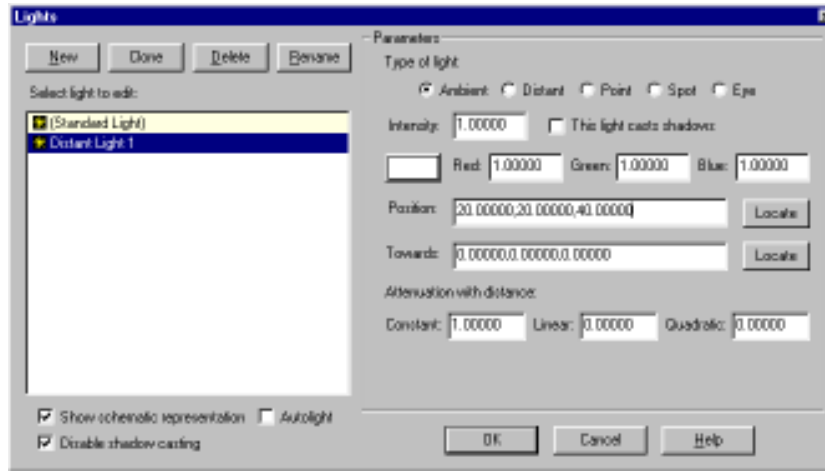
Lights...

LIGHTS illuminates your **3D** model for shading with the specified rendering program. **FastCAD v7** displays a dialog box with four kinds of light source and their relevant parameters.

Light sources have these properties:

- **Location**
- **Direction** (for distant and spot lights only)
- **RGBA color values**—three values representing the relative proportion of red, green, and blue in the light source. Each value can range from 0.0 to 10. For white light (the default) all three values equal 1.0. Color values less than 1.0 also affect the light's intensity. The alpha channel determines the lights opacity.
- **Intensity** (how bright the light is)
- **On or Off status**
- **Additional properties** for spot lights.

When you invoke the **LIGHTS** command, **FastCAD v7** displays the **Lights** dialog. Use this dialog box to manage all individual light sources in your drawing. The dialog lists all defined light sources. Each light source can be edited, copied, or deleted.



New - Creates a new light source. When a new light is created, **FastCAD** displays the ***Rename Light*** *dialog*. Type in a new name, otherwise **FastCAD v7** assigns the default name "(**Standard Light**)". The new light source is added to the bottom of the list.

Clone - Creates a copy of the selected light source. When the new light is created, **FastCAD** displays the ***Rename Light*** *dialog*. Type in a new name, otherwise **FastCAD** assigns the source name to the new light.

Delete - Deletes the selected light source.

Rename - Renames the selected light source. Type a new name in the ***Rename Light*** *dialog* and click **OK**.

Type of light

Ambient - Ambient light shines from all directions and casts no shadows. Its intensity doesn't vary with distance. You do not need to specify a location for ambient light.

Distant - Distant lights shine in one direction, from infinitely far away (like the sun). The rays of light are parallel, and light intensity doesn't decrease with distance.

Point - Point lights shine in all directions from a specific location (like a light bulb without the shade). The light's intensity decreases with distance.

Spot - Spot lights shine from a specific location in a specific direction, illuminating only objects that fall within a cone (like a stage light). The light's intensity decreases with distance.

Eye - Acts as a distant light, lighting the scene from the position of the current screen camera view, the viewers "eye."

Options

Show schematic representation - Lights display in the drawing windows as editable entities. If unchecked, actual light source positions are not displayed.

Autolight - Sets an automatic general distant light source.

Disable shadow casting - Turns off shadows cast by lights globally.

Parameters

Intensity: - The strength of the selected light, with a basic value of 1.0. Lessen or increase the intensity value of a light source to achieve different lighting effects, or as a parameter of intensity falloff over distance.

This light casts shadows - Specifies that a new light will cast shadows.

Position: - The **3D** location of the selected light, represented in **3D** numeric coordinates. To visually specify the position, click the ***Locate*** *button* and pick a point on any drawing window.

Towards: - A **3D** location that specifies the direction of the light source, represented in **3D** numeric coordinates. The direction vector is determined from the Position point to the Towards point. To visually specify the position, click the ***Locate*** *button* and pick a point on any drawing window.

Attenuation with distance: - Specify the light intensity falloff over distance.

Constant: - Specify the light intensity falloff over distance as a constant value.

Linear: - Specify the light intensity falloff over distance according to a linear scale.

Quadratic: - Specify the light intensity falloff over distance as a square of the distance from the light source.

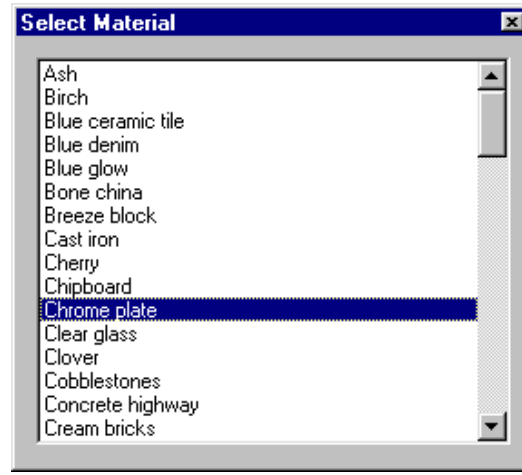
Spotlight Angle: - Angle width of a spotlight cone (Spot-type only).

Exponent: - Intensity of beam (Spot-type only).

Text equivalent: ***LIGHTS***

Material...

MATERIAL is a property of each entity and specifies the current surface material for new **3D** objects, for use with a renderer that supplies shaders for these material types. Currently **NONE ARE AVAILABLE**. The **OpenGL renderer** will **NOT** display any materials for surfaces.



NOTE: Currently there are NO photorealistic renderer applications available for FaastCAD v7. Only SOLID surfaces will be displayed on 3d objects

Text equivalent: ***MATRL***

Texture Scale

TEXTURE SCALE *scales* the *texture size* of a *selected material* used with the **LightWorks** renderer within the *active* drawing. **1.0** is the *default scale* value. Change the value to change the relative size of the texture in relation to the dimensions of 3D objects. A value of **2.0** has the effect of *doubling* the *size* of the texture, a value of **0.5** *halves* the texture *size*, etc.

Text equivalent: ***MATRLSCL***

Scale All Textures

SCALE ALL TEXTURES *scales* the *texture size* of *all materials* used with the **LightWorks** renderer within the *active* drawing. **1.0** is the *default scale* value. Change the value to change the relative size of all textures in relation to the dimensions of **3D** objects. A value of **2.0** has the effect of *doubling* the *size* of all the textures within a drawing, a value of **0.5** *halves* the textures *size*, etc.

Text equivalent: ***MATRLSCLA***

Current Mode

RMODE chooses as the default rendering mode one of the rendering modes available with the **LightWorks** renderer: It has no effect when using the **OpenGL** renderer.

 # 1 Wireframe - monochrome transparent wireframe image of the current view.
#2 Hidden Line - monochrome hidden line image of the current view.
#3 Gouraud - smooth shaded image of the current view using the Gouraud algorithm.
#4 Phong - smooth shaded image of the current view using the Phong algorithm.
#5 Preview - quick rendered image of the current view displaying materials.
#6 Normal - fully rendered and raytraced image with LightWorks materials and shadows.
#7 High Quality Anti-aliased - fully rendered and raytraced image of the current view with highest level of anti-aliasing.

Text equivalent: ***RMODE # (1, 2, 3, 4, 5, 6, 7)***

Select Renderer

SELECT RENDERER selects the rendering program **FastCAD v7** uses. **FastCAD v7** currently supports two **OpenGL renderers**.

To select the **OpenGL** render choose either the file **RENDOGLM.DLL** (**Microsoft**) or **RENDGLS.DLL** (**Silicon Graphics**) from the ***Select Render (Starts on next load)*** *dialog* that appears when select the **SELECT RENDERER** command **[Render > Select Renderer...]**.

RENDOGLM.DLL (Microsoft)

RENDGLS.DLL (Silicon Graphics)

RENDER.DLL (contains the currently selected renderer)

The **Microsoft renderer** is supplied with **Windows 98, 98Me, NT, Windows 2000 and Windows XP**. You can download the **Silicon Graphics** version on the Internet, follow the link provided at our website - **www.fastcad.com**. Other renderers can be supported by data interfaces developed with the **XP Toolkit**. All available rendering support **DLLs** are located in the **\RENDER** folder of **FastCAD**'s working directory.

This selection is copied to the file **RENDER.DLL**, which is the render file used by **FastCAD v7**.

If the ***Select Render (Starts on next load)*** *dialog* displays a blank screen (where not even **render.dll** is displaying), it's probably because your **Windows** system is configured to **'Hide protected system files'** from view. Consult your **Windows help file** for instructions on reconfiguring this option.

Text equivalent: ***SELRENDER***

Chapter 18

The Animate Menu

Animation: Creating moving "fly-through" views of 3D models with FastCAD v7

FastCAD has a series of commands that provide the ability to set up and view "**fly-through**" animations of 3D **FastCAD** drawings on screen in real time. The user can create, name, and select multiple paths for the viewing camera to follow. This allows the user to view a moving tour through an environment, model, or architectural structure; or to view **3D** objects or assemblages of objects from a camera path that extends through, along, or around those objects. Using the **OpenGL** rendering options, these "**fly throughs**" can be viewed in perspective, with shaded and rendered surfaces in full color.

The Animation menu commands are particularly useful in visualization of architectural space; to aid in the design process or to allow the user or clients to see and tour designs in real time. Animation can also be used to display mechanical or other **3D** models, viewing them in ways that might be difficult if not impossible with real objects, such as passing through them or into the interior of an entity or entities; or around them so as to make it appear that they are rotating in space. This usage can aid in visualization and communication of intricate relationships between elements in a **3D CAD** model, or the overall concept behind a **3D** design.

These powerful features can aid the user during the design process, offering the dimensions of time and motion as tools to refine and prove the efficacy of **3D** designs. Creative planning and judicious use of Animation in **FastCAD v7** can heighten the impact of a design presentation, or convey nuances of form that might be missed with a static **CAD** drawing alone.

Camera Paths

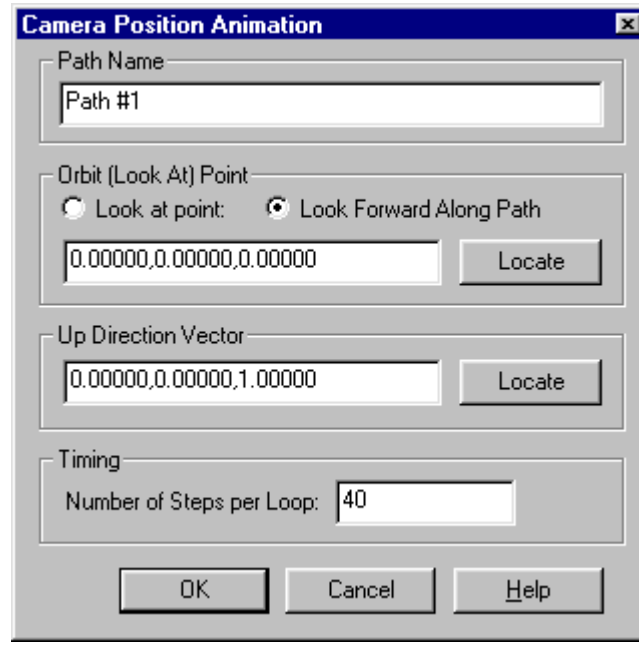
CAMERA PATH lets you define a ***camera motion path***, or "**point of view,**" for **3D** animations. The active window viewpoint is moved along the motion path in small increments through your model. Think of the ***camera path*** as a "**dolly track**" the camera follows as your animation plays out.

The ***motion path*** can be defined using any conical entity such as a **line, circle**, **ellipse**, or **spline**. Draw the ***motion path*** before using this command.

During animation playback, the selected ***camera path*** entity is always hidden from view. Other unused ***animation paths*** are not automatically hidden from view. They should be created on discrete layers that you can hide.

To set parameters for a motion path, select **[Animate > Camera Path]**:

1. The prompt reads "**Camera path entity:**". Pick a valid entity in the ***active drawing*** window. If you fail to pick a valid entity, the command terminates.
2. **FastCAD** displays the ***Camera Position Animation*** *dialog* box. Use this dialog box to define the various parameters of your animation:



Path Name Type a unique name for the selected path. This name may be referenced later by the **SHOW NAMED LOOP** and **SHOW NAMED ONCE** commands.

Orbit (Look At) Point The ***orbit point*** defines how the camera is oriented as it travels along the path.

Look at point Forces the camera to look towards a ***fixed center*** you specify. As the camera travels along the path, it continuously "**pans**" to always maintain its view on this ***orbit point***. To specify the ***orbit point***, type **3D** numeric coordinates in the text box immediately below, or click the ***Locate*** *button* and pick a point on any drawing window.

Look ForwardAlong Path Forces the camera to *always look tangentially along the* ***specified motion path***. This renders a "**pilot's eye**" point of view as the camera moves along the path.

Up Direction Vector Specifies the "**up**" direction in the animation view. You can think of this as ***how the camera is tilted as it travels along the path***. For instance, the default up direction is (**0,0,1**), or ***along the z axis***. However, if the ***up direction*** is set to (**0,0,-1**), the camera will travel the same path "**upside down**."To specify a new direction vector, type in **3D** numeric coordinates. The vector starting at (**0,0,0**) to the specified coordinates defines the ***direction***.You can also click the ***Locate*** *button* and pick two points on any drawing window to visually specify the ***direction vecto*r**.

Number of Steps per Loop Specifies the ***number of frames*** that will be rendered for each path revolution. This is loosely analogous to the ***frame rate*** (**frames per second**), except that the ***length of the loop is the governing measure***. As such, the number of steps also ***controls the velocity of the camera motion along the path***. To ***slow down*** the camera velocity, ***increase the number of steps***. You can also slow down the camera by leaving the number of steps unchanged, but selecting a ***motion path with more length***.In extremely complex animations, a higher number of steps can take more time to process, affecting rendering performance. In these cases, try lowering the number of steps.

3. After you have entered the desired parameters into the dialog, click **OK** to accept the settings and close the dialog. The command terminates.

Once you have defined parameters for a ***camera path***, those settings act as session defaults (**except for the Path Name, which must be entered anew each time this command is invoked**).

If you have given your ***camera path* a *name***, you can use the **SHOW NAMED LOOP** or **SHOW NAMED ONCE** commands and use a ***named path***.

Text equivalent: ***APOV***

Show Loop

SHOW LOOP selects a ***camera path*** defined by the **CAMERA PATH** command with the crosshair and selection box cursor, and loops repeatedly as it moves the screen camera along the defined animation path with the specified number of steps, either tangent to the path or aimed at an orbit point until the **ESCAPE** key is pressed.

To loop an animated view, select **[Animate > Show Loop]:**

The prompt reads ”**Animate camera path:**”. Pick a valid entity in the active drawing window. If you fail to pick a valid entity, the command terminates.

If the selected ***camera path*** has been defined using the **CAMERA PATH** command, those parameters are used. If the selected ***camera path*** has not been defined, then the current or default **CAMERA PATH** settings are used.

Upon selection of a valid ***camera path*** entity, **FastCAD** immediately renders and plays the animation in the active drawing window.

Text equivalent: ***ALOOP***

Show Once

SOW ONCE selects a ***camera path*** defined by the **CAMERA PATH** command with the crosshair and selection box cursor, and moves the screen camera along the defined ***animation path*** once with the specified number of steps, either ***tangent to the path*** or aimed at an ***orbit point***.

To play an animated view, select **[Animate > Show Once]**:

The prompt reads ”**Animate camera path**:”. Pick a valid entity in the ***active drawing*** window. If you fail to pick a valid entity, the command terminates.

If the selected path has been defined using the **CAMERA PATH** command, those parameters are used. If the selected path has not been defined, then the current or default **CAMERA PATH** settings are used.

Upon selection of a valid path entity, **FastCAD** immediately renders and plays the animation in the *active* drawing window.

Text equivalent: ***AONCE***

Show Named Loop

SHOW NAMED LOOP sselects a ***camera path*** defined by the **CAMERA PATH** command specified by ***name***, and ***loops repeatedly*** as it moves the screen camera along the defined ***animation path*** with the specified number of steps, either ***tangent to the path*** or aimed at an ***orbit point*** until the **ESCAPE** key is pressed.

To loop an animated view along a ***named path***, select **[Animate > Show Named Loop]**:

The prompt reads ”**Animate camera path name:**”. Type a ***camera path name*** and press **ENTER**. If you enter a non-***existent path name***, the command terminates without effect.

After a valid ***path name*** is entered, **FastCAD** immediately renders and plays the animation in the ***active drawing window***. The animation uses the parameters of the ***named path*** as defined using the **CAMERA PATH** command.

Text equivalent: ***ALOOPM***

Show Named Once

SHOW NAMED ONCE selects a ***camera path*** defined by the **CAMERA PATH** command specified by *name*, and moves the screen camera along the defined ***animation path*** once with the specified number of steps, either ***tangent to the path*** or aimed at an ***orbit point***.

To play an ***animated view*** **along a** ***named path***, select **[Animate > Show Named Once]**:

The prompt reads ”**Animate camera path name:**”. Type a ***camera path name*** and press **ENTER**. If you enter a non-***existent path name***, the command terminates without effect.

After a valid ***path name*** is entered, **FastCAD** immediately renders and plays the animation in the ***active drawing window***. The animation uses the parameters of the named path as defined using the **CAMERA PATH** command.

Text equivalent: ***AONCEM***

Chapter 19

Specs Menu

Set-up and configure* FastCAD v7 *to the way you work

The **Specs** ***(Specifications)*** *menu* controls some of **FastCAD**'s precision drawing features like **Grids**, **Frames**, and **Crosshairs**. Other commands set formats and styles for entities, text, and dimensions.

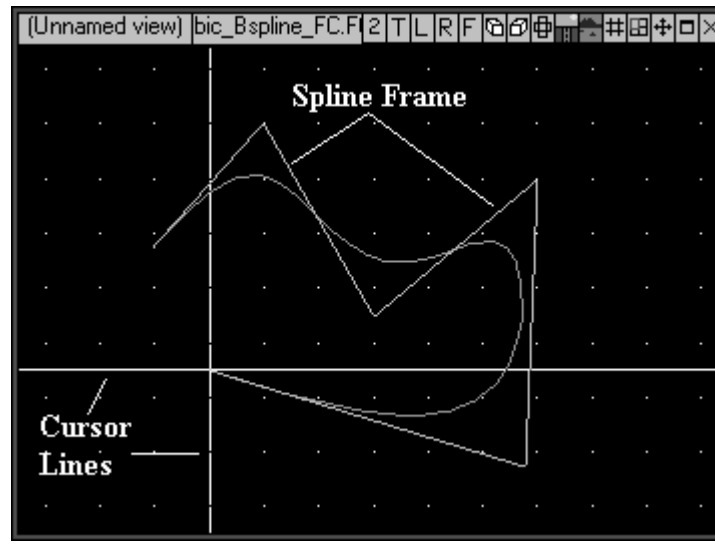
Grid button OFF

Grid button ON

Grids are drawing aids that help you visually scale, position, and orient entities in your drawing. They appear on your drawing screen as a series of evenly spaced white dots. Though you can edit their spacing properties, the grid dots themselves cannot be edited (they are *not* point entities). This "always frozen" property ensures that grids will not be inadvertently altered or moved. Toggle grid display using the ***Grid*** *button* on the ***Button*** *bar*.

Use **crosshairs** when you want more visual detail when you move the cursor. By default, **FastCAD** displays either a pointing cursor (arrow shaped) or a pick box, depending on the context. Crosshairs add horizontal and vertical tracking lines to both standard cursors.

Grids, **Cursor Crosshairs**, and **Spline Frames**, are all toggled on in the illustration below These and other options may be toggled or tailored through commands you find in the **Specs** *menu.*

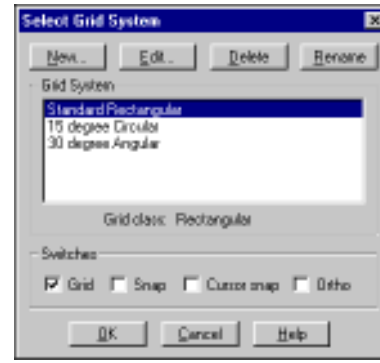


Select Grid

Selecting the **SELECT GRID** command from the **SPECS** menu displays the ***Select Grid System*** *dialog box* with options that control the ***grid system.*** You may also right-click over the ***Grid, Ortho***, ***CsrSnap*** or ***Snap*** *buttons* to open the ***Select Grid System dialog box***. ***Grids*** are classified as ***rectangular***, ***circular***, or ***angular*** (angular grids are rotated rectangular grids). Horizontal and vertical ***grid spacing*** may be set independently. You may edit existing grids to create new grids, allowing for even greater control of your drawing environment. **FastCAD** has one ***rectangular grid***, one ***15 degree circular grid*** and one ***30 degree angular grid*** already set up for you to use. They can be edited, modified, even renamed, but these two can not be deleted.

You can type ***grid names*** containing spaces on the command line when using the **SGRID** command (press ENTER to invoke the command). ***Grid*** definitions are saved with each drawing file.

Select Grid System dialog box



Dialog Options

List Window Pick one of the pre-defined grids or a grid that you have defined. If you click **OK**, the selected grid will become current.

Grid Class **FastCAD v7** currently supports three classes of grids:

- **Rectangular**
- **Circular**
- **Angular**
- **Hexagonal**

Buttons

New **Creates a new grid system**. You will first be asked to create either a **rectangular**, **circular**, **angular or hexagonal grid system**. The corresponding ***Edit Grid System dialog*** will then appear, allowing you to edit values.

Edit Allows you to modify the highlighted grid system. The appropriate ***Edit Grid System*** *dialog box* is displayed.

Switches

Delete Deletes the highlighted grid system. Note that the default grid system "**Standard Rectangular**" cannot be deleted.

Rename Renames the highlighted ***grid system.*** Type a new name in the ***Rename Grid System*** *dialog.*

Grid Displays a regular pattern of grid dots to help you keep track of distances and sizes. The dots are only a visual aid and are not part of your final drawing. Toggle the display mode with the ***Grid*** *button*.

Snap Forces point picks to regular intervals you specify. Toggle the snap mode with the ***Snap*** *button,* or by typing **SNAPON** and **SNAPOFF** at the prompt.

Cursor snap Forces cursor motion to ***snap points***. Applies only if *Snap* is on. When snap locking is on, cursor snap makes cursors "**jump**" to snap points instead of moving smoothly over the screen. Toggle the cursor snap mode with the ***CsrSnap*** *button*.

Ortho button
When you use ortho with an angular grid, "horizontal" and "vertical" are aligned with the grid.

Ortho Forces new lines or displacements to be horizontal or vertical. Toggle the ortho mode with the ***Ortho*** *button*.

Text equivalent: ***SGRID***

Align Grid

Use the **ALIGN GRID** command to reset visual alignment of the grid dots. The grid dots shift so that a grid dot coincides exactly with an alignment point you specify. This command does not reset the origin point (**0,0,0**) of the drawing. The drawing coordinates do not change; only the visual grid is shifted.

You can also use **EGRID [Specs > Edit Grid]** to enter a new alignment point. The coordinates are stored as the Grid center parameter in the ***Edit Grid*** *dialogs*.

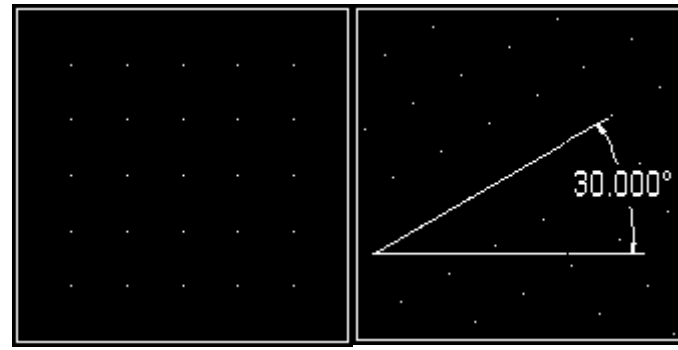
To *align* the grid, select **[Specs > Align Grid]**:

The prompt reads "**Align grid on point:**". To specify a new alignment point:

- Right-click or press ENTER to accept the default or prior value;
- Pick a point in a drawing window;
- Type numeric coordinates and press ENTER.

Text equivalent: ***AGRID***

Angle Grid



The grid on the left displays a typica l 0° square grid. The grid on the right is an Angled Rectangular class grid, set to 30°.

Use the **ANGLE GRID** command to realign the current angular grid. You can either input a value (in standard decimal or dms format) or visually specify an alignment, using the mouse. ***Ortho*** locks to the ***grid*** orientation.

ANGLE GRID ***only works if an angular-type grid is current***. If you attempt to rotate an i**nvalid grid** (**rectangular or circular**), **FastCAD** displays a message box stating, "**The currently selected grid is not an angular grid**". If the **grid** you are rotating is not visible (***Grid*** *button* **off**), it will be automatically shown after the rotation (***Grid*** *button* **on**).

To set the **grid angle**, select **[Specs > Angle Grid]**:

The prompt reads "**Bearing to +X**:". You can:

- Type in a **bearing angle**, then right-click or **press ENTER**;
- Define a bearing by clicking two points on the screen.

FastCAD ***angles*** the ***grid*** with the ***angle*** you specified.

Text equivalent: ***GRIDANG***

Edit Grid

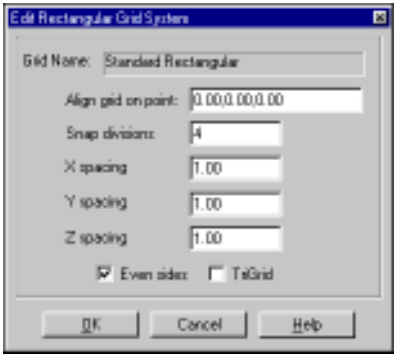
EDIT GRID allows you to re-specify any parameter for any grid system in the drawing.

FastCAD displays a different ***Edit Grid System*** *dialog box*, depending upon the type of Grid System being modified or created:

- **Rectangular**
- **Circular**
- **Angular**
- **Hexagonal**

Editing Rectangular Grids

Edit Rectangular Grid System dialog box



Dialog Options

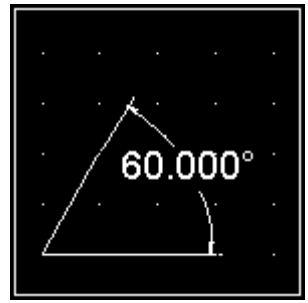
Align grid on point: The origin point (**0,0,0**) of the grid can be different than the (**0,0,0**) point set by the **CHANGE ORIGIN** command **[View > Change Origin]**]. Can be visually specified using the **ALIGN GRID** command.

Snap divisions Number of snap points per grid unit (**integer value**).

X/Y/Z Spacing Distance between grid dots in each axis direction. These values are displayed in the current distance display format.

Even sides **Forces all Spacing values equal**, **resulting in a square** (**2D**) or cubical (**3D**) *grid*. Click off **Even sides** to allow different values for **X**, **Y**, and **Z Spacing**.

TriGrid ***TriGrid*** is useful for creating isometric drawings. It creates a special grid that you can use to draw entities at **30°, 60°, 120° angles** for **ISOMETRICS**. **FastCAD** prompts you for horizontal spacing, which you can set with any of the usual methods. **FastCAD** then calculates the vertical spacing and displays the grid dots.



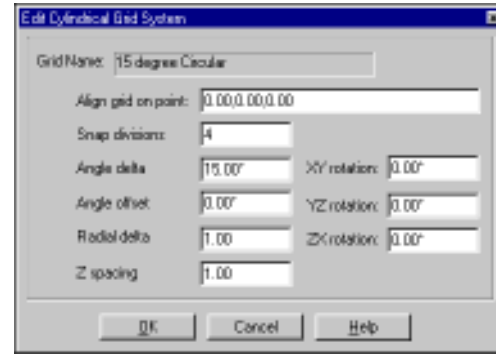
TriGrid

TriGrid creates a special grid that you can use to draw entities at **60°.** Enter the **horizontal spacing only (**into the **Spacing field). FastCAD** then calculates the vertical spacing.

Use **SNAP** to ensure that you are drawing in **angle increments of 30°** and **60°** for drawing **ISOMETRICS.**

Editing Circular Grids

The ***Edit Cylindrical Grid System dialog box*** is displayed when a **circular/ cylindrical grid** is selected for editing, or when creating a new **circular/cylindrica**l **grid**.



Edit Circular Grid System dialog box

Dialog Options

Align grid on point: The origin point (**0,0,0**) of the grid can be different than the (**0,0,0**) point set by the **CHANGE ORIGIN** command **[View > Change Origin]**. Can be visually specified using the **ALIGN GRID** command.

Snap divisions Number of snap points per grid unit (**integer value**).

Angle delta Angle (i**n degrees**) between spokes in the cylindrical grid pattern.

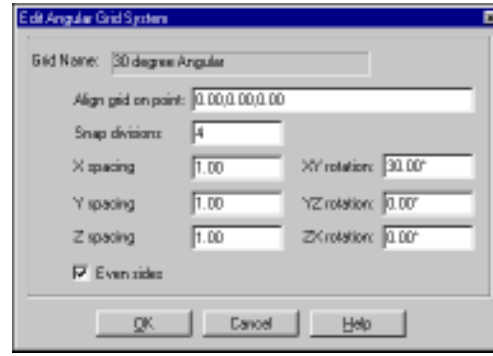
Angle offset Bearing of the first spoke in the grid pattern. The **Angle delta** is maintained for the remaining spokes.

Radial delta Distance between rings in the cylindrical grid pattern.

Z spacing Distance between "**layers**" in the cylindrical grid pattern. Note that cylindrical grids are always aligned along the **Z-axis**.

Editing Angular Grids

The ***Edit Angular Grid System*** *dialog box* is displayed when an angular grid is selected for editing, or when creating a new *angular grid*.



Edit Angular Grid System dialog box

Dialog Options

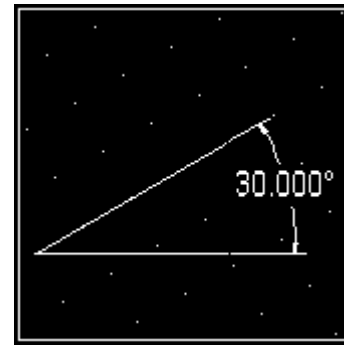
Align grid on point: The origin point (**0,0,0**) of the grid can be different than the (**0,0,0**) point set by the **CHANGE ORIGIN** command **[View > Change Origin]**. Can be visually specified using the **ALIGN GRID** command.

Snap divisions Number of snap points per grid unit (**integer value**).

X/Y/Z Spacing Distance between grid dots in each axis direction. These values are displayed in the current distance display format.

XY/YZ/ZX Rotation Enter the angle in which you wish the grid rotated in each plane. You can visually re-specify this value using the **ANGLE GRID** command **[Specs > Angle Grid]**.

Even sides Forces all Spacing values equal, resulting in a square (**2D**) or cubical (**3D**) *grid*. Click off **Even sides** to allow different values for **X**, **Y**, and **Z Spacing**.



The grid seen above is an Angled Rectangular class grid, set to

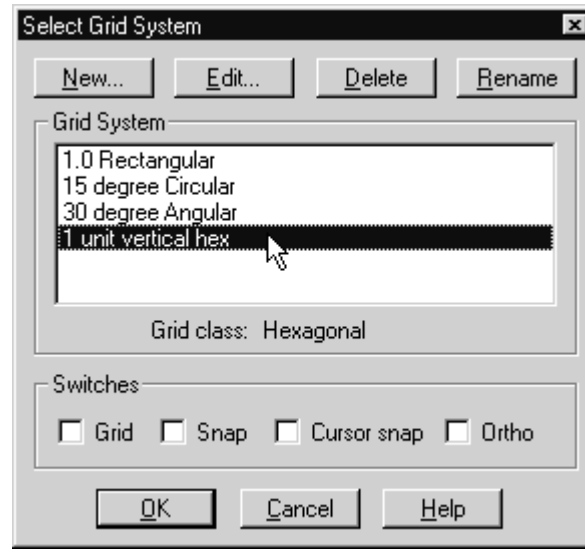
Text equivalent: ***EGRID***

Editing Hexagonal Grids

FastCAD v7 has a unique **Hexagonal Grid System**. The **HEX GRID CELL** allows you to draw **Lines at angle increments of 30 degrees**. This makes it easy to draw **isometrics**.

Selecting an Existing Hexagonal Grid System

To select a **HEXAGONAL GRID SYSTEM** place the mouse cursor over the **GRID** button and right-click to open the '**SELECT GRID SYSTEM'** dialog box as seen below and click the **'1 unit vertical hex grid'**.



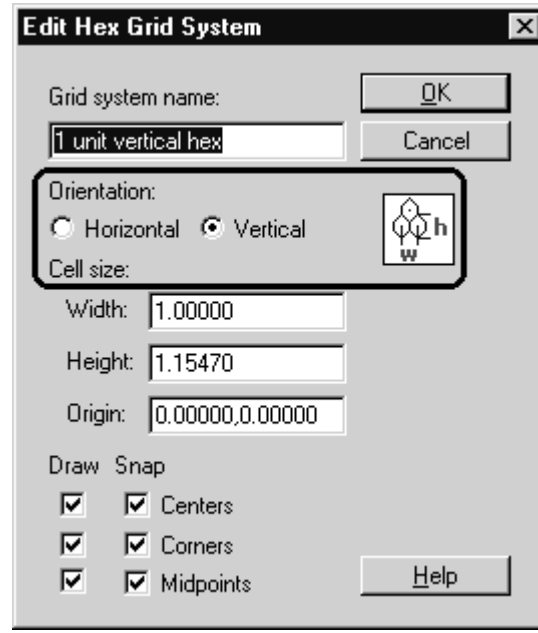
A **HEX GRID SYSTEM** is orientated either **VERTICALLY** or **HORIZONTALLY** depending on the setting in its **EDIT HEX GRIS SYSTEM** ***dialog box***.

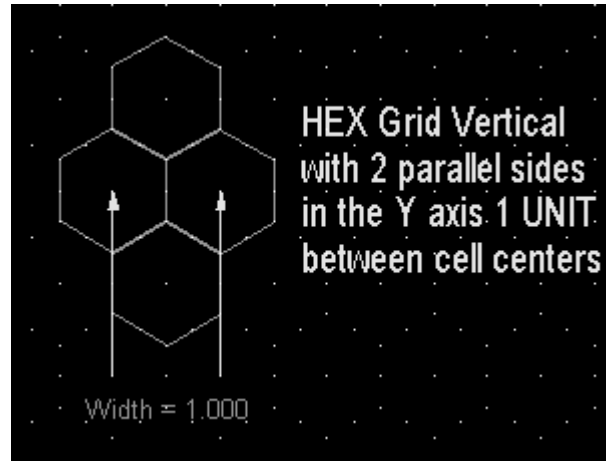
Vertical HEX Grid cell Orientation

VERTICAL cell orientation for a **HEX GRID system** means that **2 of the 6 parallel sides of the HEX GRID cell are VERTICAL in the Y drawing axis**. The **default GRID SPACING** for **VERTICAL cell orientation is set to a cell size WIDTH of 1.00000 and a HEIGHT of 1.15470 automatically**, which is the proper value to maintain the proper settings for the **HEX GRID to have six equal and parallel sides.**

The **WIDTH value is measured HORIZONTALLY between the mathematical centers of each adjacent GRID cell**. The **distance between cell centers** should be changed to make the **cell larger** or **smaller NOT** the **VERTICAL distance across the cell between two NODES**, which would be the **HEIGHT** of **1.15740** when the **HEX GRID spacing is set to VERTICAL**. See the example below for **VERTICAL** settings for the **HEX GRID cell**.

WIDTH and HEIGHT measurements for VERTICAL HEX Grid Orientation





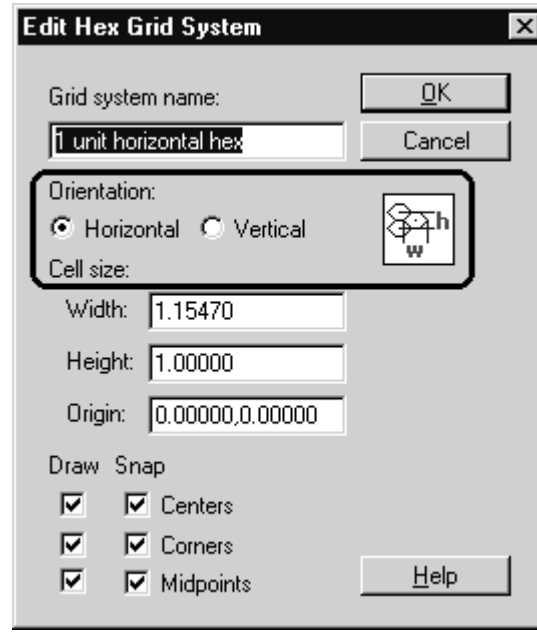
<u>Horizontal HEX Grid cell Orientation</u>

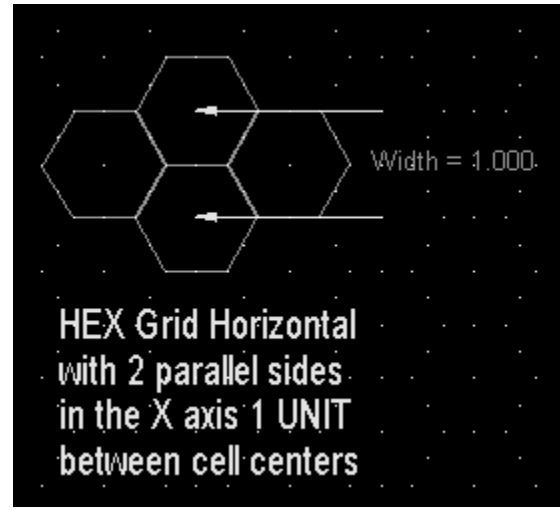
HORIZONTAL cell orientation means that **2 of the 6 parallel sides of the HEX GRID cell are HORIZONTAL in the X drawing axis**.

The **default GRID SPACING for HORIZONTAL cell orientation is set to a cell size HEIGHT of 1.00000 and a WIDTH of 1.15470 automatically**, which is the proper value to maintain the proper settings for the **HEX GRID to have six equal and parallel sides.**

The **HEIGHT value is measured VERTICALLY between the mathematical centers of each adjacent GRID cell**. The **distance between cell centers** should be changed to make the **cell larger** or **smaller NOT** the **HORIZONTAL distance across the cell between two NODES**, which would be the **WIDTH of 1.15740** when the **HEX GRID ORIENTATION** is set to **HORIZONTAL** See the example below for **HORIZONTAL settings for the HEX GRID cell**.

WIDTH and **HEIGHT** measurements for **HORIZONTAL HEX Grid Orientation**





Select Color

Select Color icon

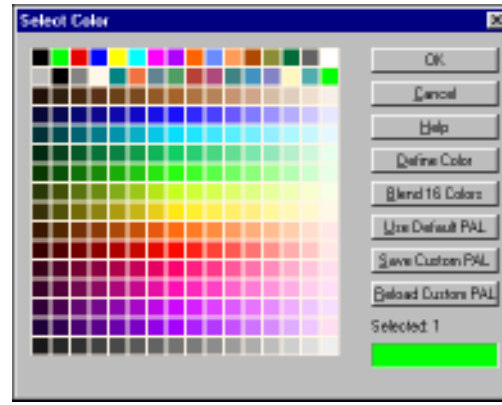
FastCAD provides you with a number of mechanisms for quickly changing the working (**active**) color. Usually, the quickest way to change the working color is to click directly on the ***Color*** *bar*. The working color is displayed on the ***Color*** *icon*. To quickly activate a color from the ***256-color palette***, you can click directly on the ***Color*** *icon*.

SELECT COLOR command **[Specs > Select Layer]** allows you to select the new active drawing color or modify the color palette. When selecting this command from the menu, you may elect to:

- Type a color number and press ENTER;
- Pick a color from the ***Color*** *bar*;
- Right-click to display the ***Select Color*** *dialog box*.

The quickest way to change your active drawing color is to select it from the ***Color*** *bar*. You can always change colors while in the middle of another command. Use the **SELECT COLOR** command only to adjust the palette, create new colors, or select a color that does not fit on the ***Color*** *bar*.

When you need to select a color from the larger *256-color palette*, choose it from the ***Select Color*** *dialog box*.



Select Color dialog box options

Dialog Options

Palette Click on any color to make it the active drawing color.

The ***Color*** *palette* is a collection of colors stored together for easy access. The palette can be customized using the other buttons on the dialog box.

Each color on the **palette** is assigned a number **0-255**. Redefining a color will change the appearance of all entities drawn in that color number.

Some colors numbers have special significance or uses:

0-23 appear on the color bar

16-20 are **system colors** extracted from the **Windows Display Properties**, Appearance settings. Modifying these colors in this dialog box will only affect the **FastCAD** display environment. They will *not* affect the appearance of Windows or other applications.

Specific color assignments:

16 button face color

17 button text color

18 button shadow color

19 list paper background

20 prompt text

31 is the "**no change**" color. Entities drawn using this color will be drawn in the same color as the prior entity. This can be useful when defining arrowhead symbols. If the arrowhead symbol is drawn, a line command immediately following will use the color of the arrowhead. When a symbol has more than one color, the color used to draw the last entity in the symbol will be used.

Define Color Display the ***Select Color*** *dialog box*, which allows you to pick you color from a much larger spectrum if your video card allows. Only the selected color is modified.

Blend 16 Colors Create a 16-color spectrum using the selected color as a base. This command is useful for creating several shades of the same hue. Seven colors to the left will gradient to black, and eight colors to the right will gradient to white. The new colors will replace any existing colors, so use this command carefully.

Use Default PAL Replace a user-customized palette with the ***Default Palette.*** The ***Default Palette*** contains the standard **FastCAD** color scheme. The ***Default Palette*** is hard-coded into **FastCAD** and cannot be modified or deleted.

Save Custom PAL Save your modified palette. **FastCAD** can save only one custom palette created with user defined colors. Individual palettes cannot be saved for each drawing. **FastCAD** saves the custom palette in a file named fcw32.pal.

Reload Custom PAL Load the ***user-defined palette*** stored in the file **fcw7.pal**.

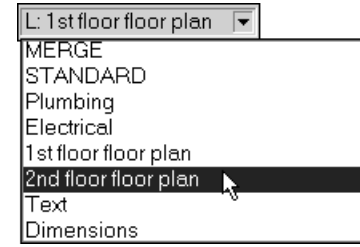
Text equivalent: ***COLOR***

Select Layer

Layer icon

The **SELECT LAYER** command **[Specs > Select Layer]** displays the ***Layer Management*** *dialog box* allowing you to select the current work layer for *new* entities or select layers to **HIDE** or **FREEZE**. .

Current Layer Status window



Available Layer Status drop=down menu

Layers & Order

When you add new layers, notice that they automically sort in alphabetical order.

There may be instances when you want layer names to appear in a specified order. You can do this if you precede the layer name with a number. Numerical digits always precede alphabetic characters.

For instance, if you want a layer named "waterways" to always appear at the top of your list, re-name it "1_waterways", or "001_waterways".

Using this method, you can precisely control the list position of your layer names.

When you import CAD drawings created in the DOS versions of EasyCAD or FastCAD, the layer numbers are translated to characters. Therfore, layer "15" appears in the list before layer "2". Layer "237" appears in the list before layer "3".

To set the current work layer:

. Click the drop-arrow at the right end of the ***Current Layer*** *status window* with the tag "**L:**" as seen below and a menu with all layers names in the *active* drawing will drop down. Click the layer name you want to make current.

FastCAD closes the menu. Note the *Layer indicator* on the ***Status*** *bar* shows the layer name you just selected. Any entities you draw from this point will acquire this layer property.

To create a new work layer:

1. Click in the ***Current Layer*** *status window* and the the ***Layer Management*** *dialog box* is displayed. Choose the **New** button.
2. **FastCAD** displays the ***Add New Layer*** *dialog box*. Type a new layer name (up to 32 characters) and click **OK**. **FastCAD** returns to the ***Layer Management*** *dialog box*. The new name is added to the list, sorted alphabetically and will automatically become the *active work layer*.
3. Click **OK** to confirm the changes and close the dialog box. If you click **Cancel**, FastCAD closes the dialog without adding the new layer(s) you just created.

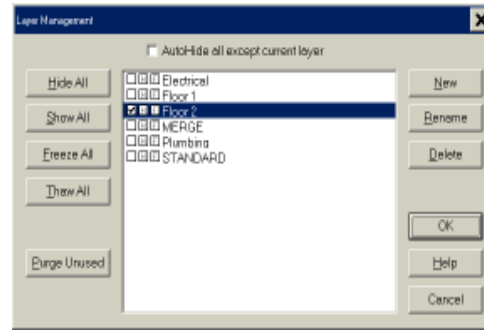
To delete an existing work layer:

1. In the ***Layer Management*** *dialog*, highlight the layer name you wish to *delete*.
2. Click **Delete**. The layer name will be deleted from the list.

Note that **FastCAD** will not allow you to delete a layer with entities on it. This is to prevent entities from becoming "**orphaned**" To remove entities from a layer, use the **ERASE** command and select entities by layer. To move entities to another layer, use the **EDIT > CHANGE > LAYER** command and select entities by layer.

FastCAD will **NOT** allow you to ***delete*** the ***current layer***

3. Click **OK** to confirm the changes and close the dialog box. If you click **Cancel**, **FastCAD** closes the dialog without deleting the layer(s) you just selected.



Layer Management dialog box

AutoHide all except current layer box If this box is checked when you pick a different layer as the *current work layer*, *all other layers are automatically hidden.* This is a quick way to see which entities are on a specific layer. Normally this box should not be checked.

Layer Status boxes Three columns of checkboxes display the status of the listed layers. The *left column* is the *Active Work Layer*; only one layer can be designated as "**active**" at a time. The *center column* indicates that the layer is **HIDDEN**. The *right column* indicates that the layer is **FROZEN**.

Layer Name All layers are listed in alphabetical order.

Hide box [H] Toggles the ***hide/show*** state of the highlighted layer. The current later cannot be hidden.

Freeze box [F] Toggles the ***freeze/thaw*** state of the highlighted layer. The current layer cannot be frozen.

Purge Unused button Automatically **deletes** any **layer** that contains no entities. This can be helpful when importing drawings that have many valid but unused layers.

Rename button Renames any layer except "**STANDARD**" and "**MERGE**".

New button Adds a *new* layer. Type a name (**up to 32 characters**) in the ***New Layer Name*** *dialog box* and click **OK**. Layer names can contain spaces. You can define a new layer with a name even in the middle of the **CHANGE-LAYER** or **COPY TO LAYER** commands.

Delete **Deletes** the highlighted **layer name**. The "**STANDARD**" and "**MERGE**" **layers cannot** be ***deleted***. **Layers** containing entities **cannot be deleted**.

Text equivalent: ***LAYERDLG***
*Macro equivalent (**bypass dialog box**):* ***LAYER***

Merge & StandardLayers

Merge and **Standard** are the two default layers that new FastCAD drawings start with. Is there anything special about them?

In the case of **Merge**, the answer is yes.

The **Merge** layer has special significance when used with parts and symbols. If the entities of a part are on the layer **Merge**, then they will import onto the current layer of the target drawing. If part entities are on any other layer, then the entities will import on those layers instead. If those layers don't exist in the target drawing, they are created.

Symbols import the same way. Draw them on layer **Merge** (or use the CHANGEL command after you've drawn them). When you insert those symbols, the symbol reference acquires the current layer property.

There is and can be only one **Merge** layer.

On the other hand, the **Standard** layer is not so special. It's simply there so you have a non-**Merge** layer to draw on.Neither **Merge** or **Standard** may be ***renamed*** or ***deleted.***

Select Layer

The **SELECT LAYER** command **[Specs > Select Layer]** displays the ***Layer Management*** *dialog box,* allowing you to select a *new current work layer* for new entities or **HIDE** nad **SHOW** other layers..

Text equivalent: ***LAYERDLG***

Hide

The **HIDE** command hides layers. Entities on hidden layers aren't affected by edit and copy operations. Hidden layers aren't included in printed output. After invoking the **HIDE** command, the prompt reads "**Layer name or # [dialog]:**". Type the layer name or press ENTER to see the ***Layer Management*** *dialog box,.*

Text equivalent: ***HIDE***

Show

The **SHOW** command displays a hidden layer. After invoking the **SHOW** command, the prompt reads "**Layer name or # [dialog]:**". Type the layer name or press ENTER to see the***Layer Management*** *dialog box,.*

Text equivalent: ***SHOW***

Hide All

The **HIDE ALL** command hides all but the current layer at once. You may find it more convenient than using the ***Layer Management*** *dialog box* and clicking the **HIDE ALL** button.

Text equivalent: ***HIDEA***

Show All

The **SHOW ALL** command displays all hidden layers at once. You may find it more convenient than using the ***Layer Management*** *dialog box* and clicking the **SHOW ALL** button. .

Text equivalent: ***SHOWA***

Freeze

Use the **FREEZE** command to freeze a layer or layers. Freezing a layer is the same as "**write-protecting**" the layer. You can see entities on *frozen layers* and refer to them with modifiers, but you can't select them for edit or copy operations. After selecting the **FREEZE** command, the prompt reads "**Layer name or # [dialog]:**". Type the layer name or press ENTER to see the ***Layer Management*** *dialog box*,.

Text equivalent: ***FREEZE***

Thaw

The **THAW** command *unfreezes* a layer. At the "**Layer name or # [dialog]:**" prompt, type the layer name or press **ENTER** to see the ***Layer Management*** *dialog box*,. You can use **FREEZE** to *freeze* any thawed layer (**except the current layer**).

Text equivalent: ***THAW***

Freeze All

Use the **FREEZE ALL** command to freeze all layers, except for the current layer, which must remain thawed. You may find **FREEZE ALL** more convenient than the clicking the **FREEZE ALL** button in the ***Layer Management*** *dialog box* if you need to *freeze all but the current layer* at once.

Text equivalent: ***FREEZEA***

Thaw All

Use the **THAW ALL** command to *unfreeze all frozen layers* at once. You may find it more convenient than clicking the **THAW ALL** *button* in the ***Layer Management*** *dialog box*.

Text equivalent: ***THAWA***

List Layers

The **LIST LAYERS** command **[Specs > Layer Options > List]** lists all drawing layers in a drawing file. The list is displayed in a text window. Note that the existence of a layer does not necessarily mean that it contains any entities.

Click the **PRINT** or **SAVE TEXT** icons at the right end of the **LIST** *dialog title bar* to print the list or save to a **.TXT** file.

If you wish to delete any layers that are unused, you can do so in one step by opening the ***Layer Management*** *dialog* and clicking the ***Purge Unused*** *button*.

Text equivalent: ***LISTLYR***

Line Style


Line Style icon

LINE STYLE allows you to ***change the current line style***. You can either type in a l*ine style* name or ID #, or you can right-click to display the ***Line Styles*** *dialog* for the *active* drawing. You can also use this dialog box to ***create***, ***edit***, and ***delete*** styles. When you choose a different style from the list, the style is applied only to new entities that are drawn.

Note that you can also click in the ***Line Style*** *status window* to choose from a list of used ***line styles***, or to display the ***Line Styles*** *dialog*.

To change the style of an individual entity, use the **EDIT** command. To change the style of a collection of entities already drawn, use **EDIT > CHANGE >-LINE STYLE.** Both of these commands can be found in the **EDIT** menu.

Understanding Line Style Patterns

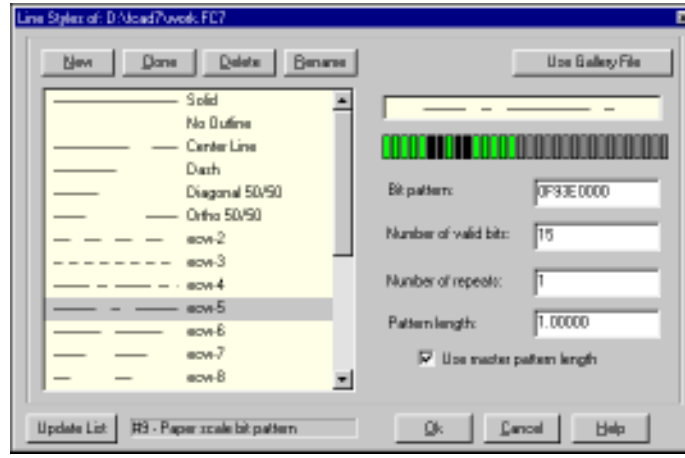
A **FastCAD** line entity as seen on the screen and printouts is actually a defined pattern drawn over a specified distance, then repeated as needed.

Each repeating pattern is comprised of a specific sequence of alternating line segments and spaces. Each sequence consists of up to 32 divisions, or "bits," that can be toggled **ON** to make segments, or **OFF** to make gaps. A basic dash sequence, where the line and gap sizes are equal, can be defined by two bits.

Sequences are repeated one or more times within their pattern length to achieve desired density. For example, a basic dash sequence can be made denser by increasing the number of repeats.

The pattern length is the paper scale length of the defined pattern. Because ***line styles*** are ***paper scale***, they print with the same spacing regardless of the drawing's print scale. ***Line styles*** are also displayed at the same resolution regardless of zoom factor.

Line style dialog



Line Style dialog box

Dialog Options

List window All available ***line styles*** are graphically sampled in a scrollable list. When you click on any pattern, its editable properties are displayed in the dialog's right pane, and the ***Line Style*** name updates accordingly.

New Creates a ***line style.***

Clone Creates a new ***line style*** by copying the currently selected ***line style***. The new style is added to the end of the list and is made current for editing or use.

Delete Deletes the selected *line style*. Any ***line style*** can be renamed except "**Solid**".

Rename Renames the selected ***line style.*** Any *line style* can be renamed except "**Solid**".

Use Gallery File Opens the file dialog, where you can specify an existing gallery file. **FastCAD** lets you import styles contained in gallery files.

Update List Resets the ***line style*** *status* to show only "**Solid**", "**No Outline**", and those styles used in the drawing. New ***line styles*** are added to this list, whether they are used or not.

Style status Displays the selected style's **ID#** and type.

Editing Options

Pattern Sample Displays a real-size, repeating sample of the ***line style*** as it would appear in actual use.

Pattern Edit Each ***line style*** is defined by a repeating pattern consisting of up to **32 on/off** "**bits**." The **Number of valid bits** determines how many of the bits represent the pattern. Click on each valid bit to switch it **on** or **off**. The **Pattern sample** updates to reflect the edit.

Bit pattern Hexadecimal value representing the on/off state of each of the **32 bits** in the displayed pattern. If known, you can enter the any bit pattern (in hexadecimal notation) to define a pattern.

Number of valid bits Number of bits used to define this pattern. The range of valid integer values is **1** through **32 bits**.

Number of repeats Number of times the bit pattern is repeated in the pattern. The higher the number, the denser the pattern. Decimal values can be entered.

Pattern length The length of each pattern, in ***paper scale*** units. If modified when **Use master pattern length** is checked, the value is used for all such patterns.

Use master pattern length Forces use of the master pattern length. All such patterns use the same pattern length. To set a new master pattern length, change the **Pattern length** while checked. To allow the selected pattern to use a unique length, uncheck this option and enter a new **Pattern length**.

Text equivalent: ***LSTYLEDLG***

Fill Style


Fill Style icon

Fills are repeating patterns you can assign to bounded entities such as polygons, circles, and ellipses. Fills add information to your drawings by visually or schematically representing different surfaces, materials, or textures. Fills are sometimes called "**cross hatches**" or "**hatches**".

All entities possess the ***fill style*** property, but the default setting is "**Hollow**". Entities with the "**Hollow**" ***fill style*** are *transparent* and exhibit no fill pattern.

FILL STYLE allows you to change the current ***fill style***. When a ***fill style*** is made current, all new valid entities will draw with that fill pattern. **FastCAD** provides you with many different pre-built fills and hatches you can immediately use. **FastCAD** also provides means for creating your own fills, so your selection is virtually unlimited.

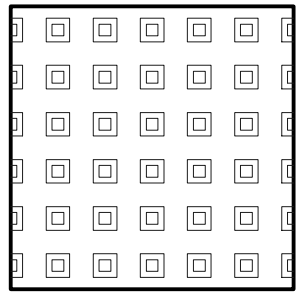
To set the current ***fill style***, you can invoke the **FILL STYLES** command and enter an existing ***fill style*** name (select from the dialog or type directly from the keyboard). To access a much broader set of ***fill style*** tools, right-click to display the ***Fill Style*** *dialog box*. Use this dialog box to ***apply***, ***create***, ***edit***, and ***delete*** styles. When you choose a different style from the list, the style will only be applied to new entities that are drawn.

To change the ***fill style*** of an individual entity, use the **PEDIT** command **[Edit > Properties…]**. To change the ***fill style*** of a collection of entities already drawn, use the **CHANGE-FILL STYLE** command **[Edit > Change > Fill Style]**.

The color of the ***fill style*** can be made different than that of the defining entities. Use the **CHANGE-2ND COLOR** command **[Edit > Change > Fill (2nd) Color]** to set the secondary color.

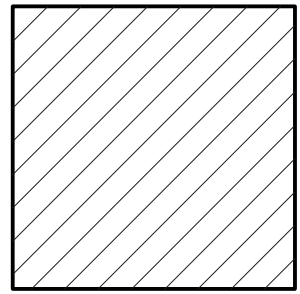
FastCAD provides four types of fill styles:

Brush Patterns



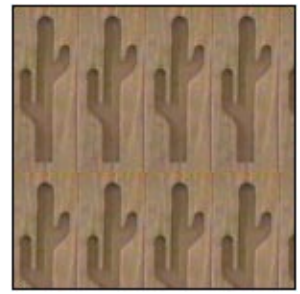
Brush pattern *fill styles* are patterns composed of a repeating 8x8 matrix of pixels. Brush patterns are drawn in the selected **FastCAD** color. This type of fill offers less constraint than others in pattern layout because you can "**paint**" it. Since brush patterns are drawn at the pixel level, they do not scale up or down with drawing. They will always appear the same size on your printouts.

Scaleable Hatching



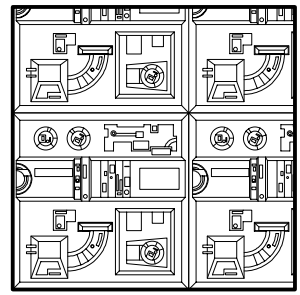
Scaleable hatching is a type of fill pattern composed of actual line entities. Up to four lines can be used to create the repeating pattern. Scaleable hatch patterns should be used when outputting to pen plotters since they consist of plottable vector entities.

Bitmap Files



Bitmap (BMP) files can be used as a fill pattern. A single image can comprise the entire fill, or smaller bitmap images can be tiled to produce a repeating effect. Bitmap fills can be useful in creating a realistic texture or background in a drawing.

Symbol Fills



Symbol fills provide a means of arraying **FastCAD** drawing entities as a fill pattern. This type of fill provides great flexibility in design since anything you draw in **FastCAD** can be turned into a symbol fill. Symbol fills are vector entities that can be plotted on a pen plotter.

Options to select and edit these ***fill styles*** appear on a single dialog box called a property sheet. Across the top of the property sheet are four tabs, each tab corresponding to a page displaying options for each type of ***fill style***. Make page selections by simply clicking on the appropriate tab.

All of the ***Fill Style*** dialogs share some common controls:

List window All available ***fill styles*** are graphically sampled in a scrollable list. When you click on any pattern, its editable properties are displayed in the dialog's right pane and the **Fill Style Name** updates accordingly.

New Creates a ***fill style.*** Select the type from the ***Type of New Style*** *dialog*. Four options are available:

- **Brush Pattern**
- **Hatch**
- **Bitmap File**
- **Symbol**

Clone Creates a new ***fill style*** by copying the currently selected ***fill style***. The new style is added to the end of the list and is made current for editing or use.

Delete Deletes the selected ***fill style.*** Any ***fill style*** can be deleted except "**Hollow**" and "**Solid**".

Rename Renames the selected fill style. Any fill style can be renamed except "**Hollow**" and "**Solid**".

Use Gallery File Opens the ***Load Gallery*** *dialog*, where you can specify an existing **FG7** gallery file. **FastCAD** lets you import styles contained in gallery files.

Fill Style Name Pick an available fill style from the list to apply, edit, or delete. When you click on any name, the pattern's editable properties are displayed in the dialog's right pane and the ***List*** *window* updates accordingly. The selected style is made current.

Outlined Draws the outline to be drawn around the fill pattern. The outline will be drawn in the current line style and color. When off, the bounding entity is suppressed from view.

Update List Resets the ***Fill Styles*** *status* to show only **HOLLOW**, **SOLID**, and those styles used in the *current drawing*. New ***fill styles*** are added to this list, whether they are used or not.

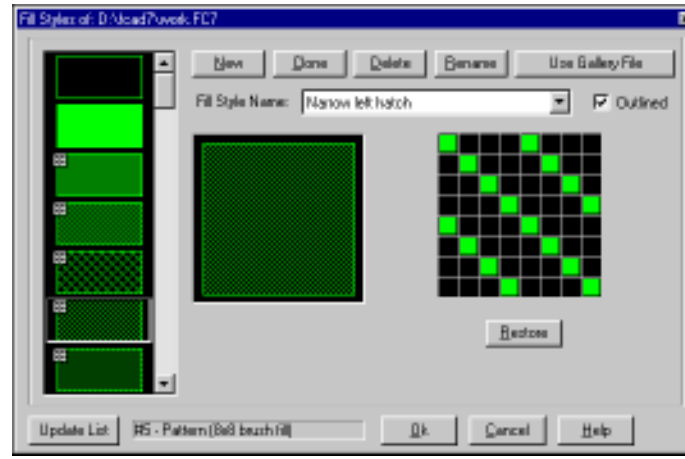
Style status Displays the selected style's ID# and type.

Text Equivalent: ***FSTYLE***

Brush Pattern Fill Style

BRUSH PATTERN fill patterns are fill patterns composed of a repeating 8x8 matrix of pixels. Brush pattern fills are drawn in the selected **FastCAD** color. This type of fill offers less constraint than others in pattern layout because you can "**paint**" it. Since brush patterns are drawn at the pixel level, they do not scale up or down with drawing. They will always appear the same size on your printouts.

The ***Fill Style*** *dialog* automatically updates to show the editing controls approprotiate to the selected **fill style** type.



You can ***create***, ***edit***, or ***delete*** brush pattern ***fill styles*.**

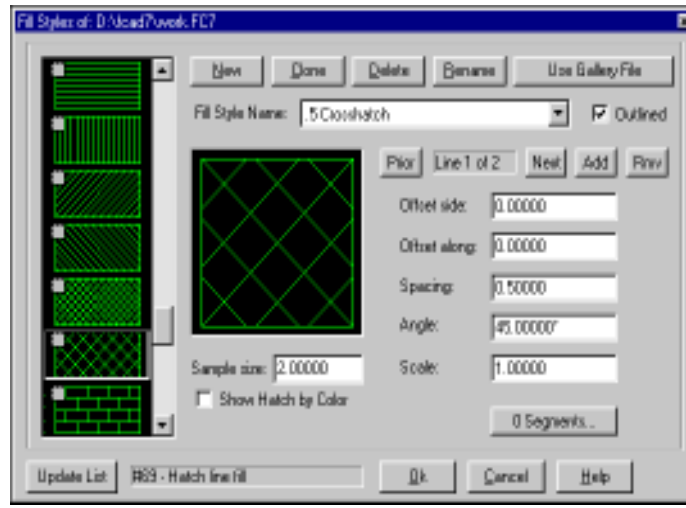
Pattern Sample Displays a real-size, repeating sample of the brush pattern as it would appear in actual use.

Pattern Edit An **8x8 matrix** of **pixels** that let you modify the selected brush pattern. Click on any pixel to toggle it **ON** or **OFF**. Watch the **Pattern Sample** to visually assess your modified pattern

Restore Restores the edited pattern to its state before the dialog was opened. All session pattern edits for the displayed ***fill style*** will be lost.

Hatch Fill Style

HATCH fill patterns are composed of true line entities. Up to four lines can be used to create the repeating pattern. Hatch patterns should be used when outputting to pen plotters since they consist of plottable vector entities.



Dialog Options

Sample Displays a scale sample of the ***fill style,*** reflecting pattern changes as you make them.

Sample size Sets the number of times the pattern is repeated in the sample. For instance, if you set the size to "**4**", the defined pattern is displayed in a **4x4 matrix**.

Show Hatch by Color Temporarily shows each line in a different color to assist in editing. These colors are never displayed in the actual drawing.

Hatch Line Edit

Hatch Edit describes the position, scale, and orientation of each line that comprises the hatch pattern. The display updates depending on which line is displayed (cycle using the **Prior** and **Next** buttons).

Prior Updates parameter display/edit for the prior hatch line.

Hatch Line status Displays the current hatch line. Use the Prior and Next buttons to move the list.

Next Updates parameter display/edit for the next *hatch line*.

Add Creates a new hatch line. The ***hatch line*** number is updated in the **Hatch Line** status. New ***hatch lines*** are created using default parameters. The new ***hatch line*** is immediately made current for editing.

Rmv Removes the current ***hatch line***.

Offset side Sets vertical displacement value for the current ***hatch line***, measured from the nominal origin of the hatch pattern (**lower left**).

Offset along Sets horizontal displacement value for the current **hatch line**, measured from the nominal origin of the hatch pattern (**lower left**).

Spacing Sets the repeat distance (in **FastCAD** units) for the current ***hatch line***. This spacing is the perpendicular distance between the repeated lines.

Angle Sets the bearing of each line in the line set. The angles correspond to the default **FastCAD** orientation, where the "**3 o'clock**" position equals 0° and positive rotation is counter-clockwise.

Scale Multiplies the offset, spacing, and segment values by the specified factor.

N Segments… Switches to the segment edit mode. N is the current number of segments.

Segment Edit

Use the Segment Edit mode to describe each of the lines that can comprise a hatch pattern. If your pattern uses only solid lines, then would not have to edit segments. However, if your hatch pattern requires the use of dashed lines, define them here.

Segment length Specifies the length of a line segment or gap, as shown in the Segment List below. You can enter a new segment by typing a length. Positive numbers are visible segments, and negative numbers create gaps of that length.

Change Updates the selected segment with the Segment length value.

Before Inserts the Segment length segment or gap before the selected segment.

After Inserts the Segment length segment or gap after the selected segment.

Remove Deletes the selected segment from the Segment List.

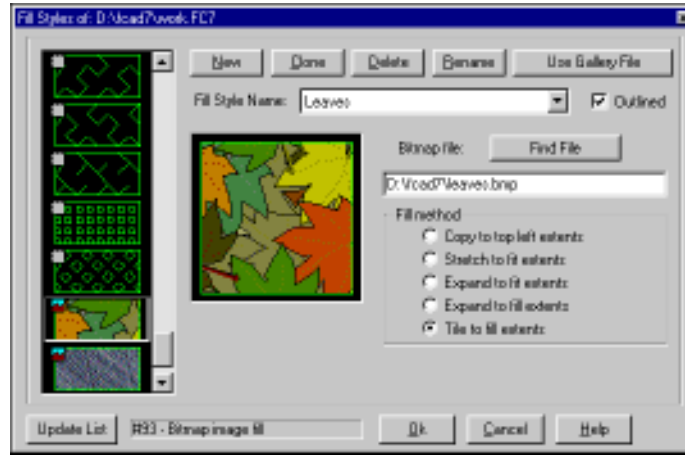
Segment List Displays all segments that define the current hatch line. To select any segment, pick it from the list.

Flip Toggles the +/- of each segment value. Positive values are visible segments, while negative values are gaps.

Close segment list Ends the segment edit mode and returns you to the regular fill style dialog.

Bitmap Files Fill Style

BITMAP (BMP) FILES can be used as a fill pattern. A single image can comprise the entire fill, or smaller bitmap images can be tiled to produce a repeating effect. Bitmap fills can be useful in creating a realistic texture or background in a drawing.



Segment Edit

Sample Displays a scale sample of the ***fill style,*** reflecting pattern changes as you make them.

Find File Opens the standard File dialog, filtered to display **BMP** files in the current folder. If you pick a valid bitmap image file, it is used for the selected ***bitmap fill style.***

Bitmap file Displays the current bitmap image used for the selected ***fill style***. To specify a different image file, type a new path/name or click Find File.

Choose one of the following options to specify ***bitmap fit*** and ***scaling***.

Fill Method

Copy to top left extents The specified image retains its original size, and it abuts itself to the upper left corner. If the image is larger than the drawn box, it will clip to the right or bottom as needed. If the image is smaller than the drawn box, the blank area will appear transparent.

Stretch to fit The image conforms to the aspect ratio of the rectangle you draw. The image is centered in the fill boundary. Note that this option may distort the image. For instance, a square image will appear stretched in one direction if you fill a rectangular box.

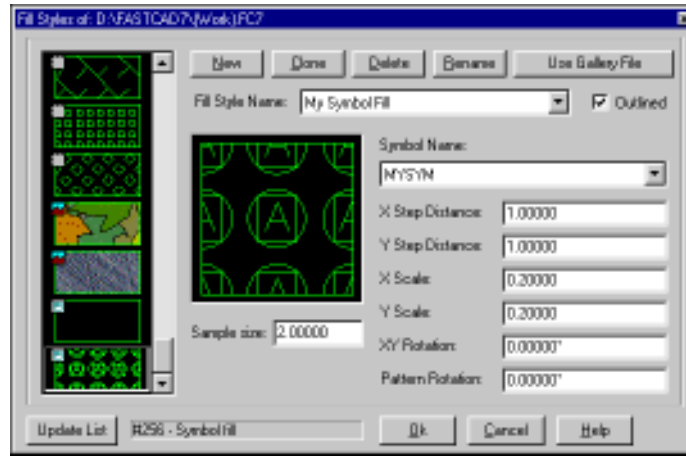
Expand to fit extents The image is scaled to fit entirely within the fill boundary, without clipping. The image is centered in the fill boundary and maintains its original aspect ratio. Any blank area in the box will appear transparent.

Expand to fill extents The image is scaled to completely fill the boundary, clipping as needed. The image is centered in the fill boundary and maintains its original aspect ratio.

Tiled to fill extents The image maintains its original size and aspect ratio, and repeats itself within the fill boundary as to fill it. The "**start**" image aligns to the top left extents. Edges will clip as needed.

Symbol Fill Style

SYMBOL FILLS provide a means of arraying **FastCAD** drawing entities as a fill pattern. This type of fill provides great flexibility in design since anything you draw in **FastCAD** can be turned into a symbol fill. Symbol fills are vector entities that can plotted on a pen plotter.



Sample Displays a scale sample of the ***fill style***, reflecting pattern changes as you make them.

Sample size Sets the number of times the pattern is repeated in the sample. For instance, if you set the size to "**4",** the defined pattern is displayed in a 4x4 matrix.

Symbol name Specifies the symbol to be used in the fill pattern. To see a list of all available symbols in the drawing, click the drop arrow. You can pick a symbol from this list.

X Step Distance Specifies the distance (in drawing units) between the repeating pattern objects in the **X** (**horizontal**) direction.

X Step Distance Specifies the distance (in drawing units) between the repeating pattern objects in the **Y** (**vertical**) direction.

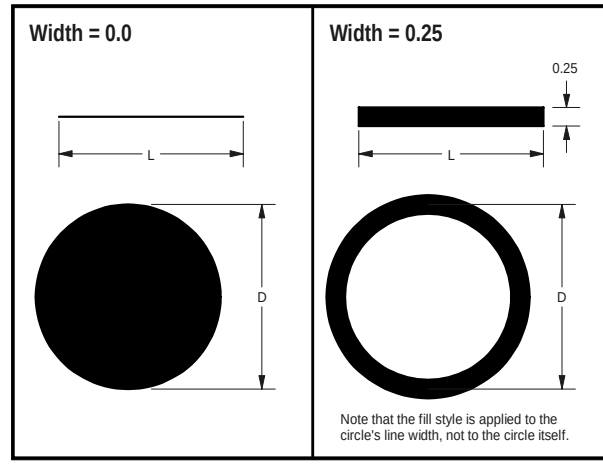
X Scale Horizontally scales the symbol in the fill pattern by the specified factor.

Y Scale Vertically scales the symbol in the fill pattern by the specified factor.

XY Rotation Rotates the symbol within the pattern by the specified angle.

Pattern Rotation Rotates the entire pattern by the specified angle.

Line Width



The **LINE WIDTH** command displays the *'**Line Width'** dialog box*. Use this command to set the ***line width*** for all new entities.

Understanding Line Widths

LINE WIDTH ***is a geometric entity property***, in the same manner that each entity possesses ***color*** and ***layer*** information. As such, the ***current line width*** is displayed on the ***Status*** *bar*, labeled with the tag '**LWid**:' as seen below.

Current Line Width status window

All new entities will be created with the ***current line width***. ***Line widths*** are always expressed in terms of the ***current drawing*** **UNITS**.

If the ***line width*** is **0.0** (**zero**), the entity will ***display*** and ***print*** in the usual ***one pixel width***.

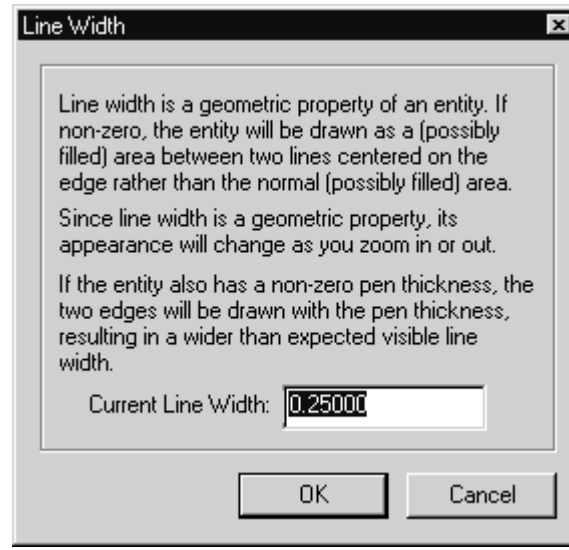
If the ***line width*** is greater than the ***nominal*** **0.0** ***width***, it will draw with the entity's f***ill style***. New entities with ***line width*** will ***fill*** with the ***current fill style*** as displayed on the ***Status*** *bar*. ***Line widths always scale up or down***, depending upon your ***zoom factor***, to the ***minimum one pixel display width***.

If you wish to draw ***filled polygons***, ***set a current fill style to something other than*** **HOLLOW** and use a **0.0 Line Width**. If you assign a ***line width*** to a ***polygon***, the ***polygon*** will be **HOLLOW** and the ***outline*** the ***polygon will be filled to the specified width***. For instance, if you draw a ***polygon*** with a Line Width of **0.25"**, **FastCAD** will

draw the ***polygon outline*** with a **0.25" Line Width** and ***fill only the outline***, ***not*** the ***interior of the polygon***.

Any entities combined in a **MULTIPOLY** will redraw with a ***line width*** of **0.0** for fill pattern display. Remember that the intention of the **MULTIPOLY** command is combining entities for fill or hatching. When a **MULTIPOLY** is **EXPLODED**, the component entities revert to their original ***line width*** and display appropriately.

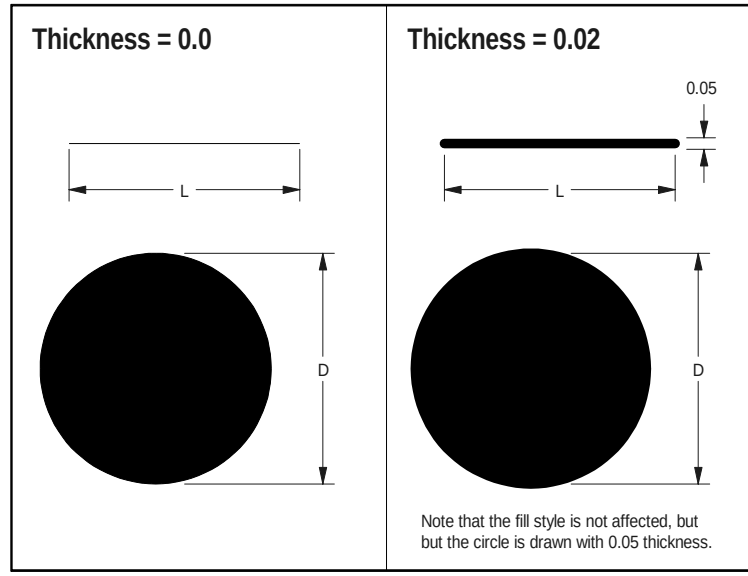
Line Width dialog box



NOTE: ***Line Widths*** **are always** ***geometrically scaled*** when ***zooming in*** or ***out*** and ***when printed they obey the scale set in the 'Print Drawing'*** *dialog box*.

Text equivalent to display the dialog box: ***LWIDTHD***
Macro equivalent to bypass the dialog box: ***LWIDTH***

Pen Thickness



PEN THICKNESS is the ***display*** and ***printing* of *thicker lines*** for entities. Specify or change the desired ***pen thickness*** by clicking directly on the indicator labeled with the tag '**PThk**:' as seen below.

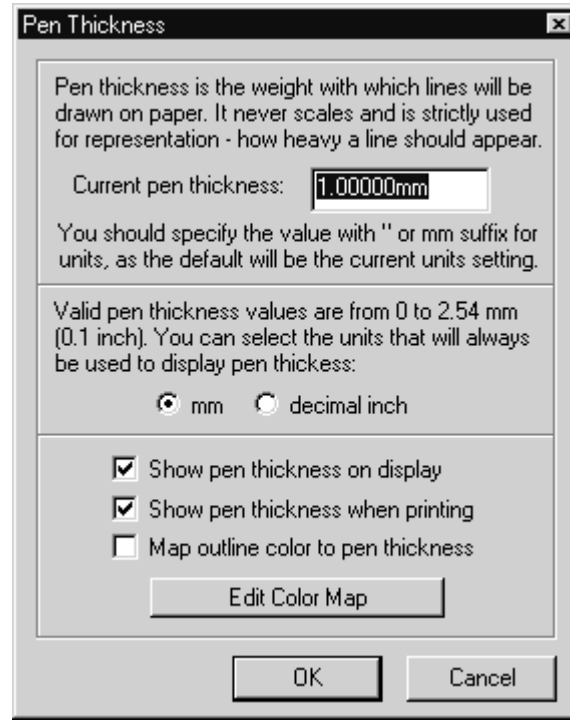
The term **PEN THICKNESS** is used because it is analgous to the older ***pen type plotters*** that used an ***ink pen of a specific thickness*** such as **.1mm, .3mm, 1mm** etc. These pens always draw that ***exact line thickness*** when plotting the drawing. Likewise our **PEN THICKNESS** property always ***prints*** and ***displays*** the s***ame thicknes*** no matter the ***scale*** or ***zoom factor*** you use when ***printing*** the drawing.

Current Prn Thickness status window

The ***Status*** *bar* displays the ***current pen thickness***. Any entities you draw will use this ***pen thickness***. You can access the ***'Pen Thickness'*** *dialog* by clicking directly on the ***'Pen Thickness'*** *indicator window*. **FastCAD** displays the ***'Pen Thickness'*** *dialog*, which lets you specify ***paper* scale *thickness*** and **MILLIMETER UNITS**.

Pen Thickness Dialog Box

Pen Thickness dialog box



Pen thickness is always in ***paper scale*** meaning that ***it is unaffected by the current drawing* UNITS *setting***. Its **UNITS** are always determined by that specified in the ***'Pen Thickness'*** *dialog*. Its **UNITS** are specified in **MILLIMETERS** (**mm**) or **INCHES**.

Valid values for ***pen thickness*** are **0.0 through 2.54 mm** (**0.0** through **0.100** inches).

Current Pen Thickness Specifies the ***pen thickness*** value in **MILLIMETERS**. This ***pen thickness*** will be applied to all new entities.

Show pen thickness on display If **unchecked** it ***suppresses screen display of pen thicknesses***. Use this option ***for faster screen redraws*** should it become an issue. Checking this option will **NOT** affect any entities' **actual** ***pen thickness settings*** or ***printing*** of them; only their display is affected.

Show pen thickness when printing If **unchecked**, it ***suppress transmission of pen thickness information to the printer without actually changing these settings.***

Map outline color to Pen Thickness All **256 drawing colors** are available for ***mapping to Pen Thickness***. Click the ***'Edit Color Map'*** *button* to configure the ***color mapping***.

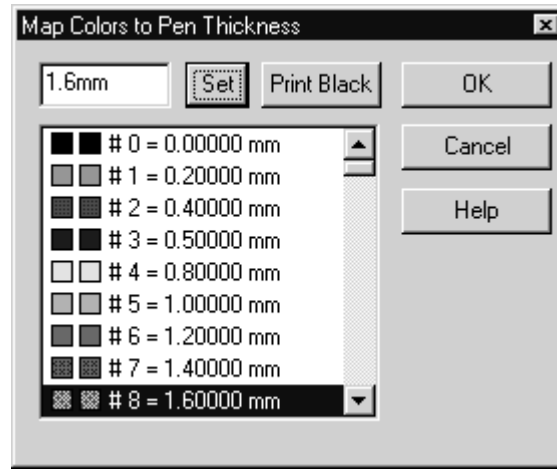
Edit Color Map button Displays the ***'Map Colors to Pen Thickness'*** *dialog box*. Settings configured in this dialog box only take effect if the ***'Map outline color to pen thickness'*** *box* is checked.

To change the ***pen thickness*** of an existing entity:

Use the **[Edit > Change > Pen Thickness]** command and select the entities to change and ***type a new pen thickness followed by the*** '**mm**' for **MILLIMETERS**. If you don't type the '**mm**' after the ***pen thickness value*** the entities **WILL NOT CHANGE**.

Text equivalent: PTHICK

Map Color to Pen Thickness dialog box



The ***'Map Color to Pen Thickness'*** *dialog* is displayed when you choose the ***'Edit Color Map'*** *button* on the ***'Pen Thickness'*** *dialog*.

The controls here allow you to assign a ***pen thickness*** to each of **FastCAD**'s ***256 drawing colors***. The ***pen thicknesses*** set in this dialog are **NOT** assigned to each entity. If you use the **[Info > List]** command and select an entity, its ***pen thickness*** would list as '**0.00000mm**' because the ***'Map outline color to pen thickness'*** *option* is just a ***translater*** and doesn't directly effect entities.

If you send a drawing to another **FastCAD** user and that **FastCAD**s ***color map*** was not enabled or was set with different ***pen thicknesses*** than your **FastCAD**, the ***pen thickness will obey that color map***. If you want you ***pen thicknesses*** to be ***portible*** between **FastCAD** users, you should ***manually assign pen thickness*** to each entity in the drawing as you draw it. You may also write a **Macro** to automate this process.

A ***pen thickness specifically assigned to an entity in the drawing take precedence over a pen thickness specified in the 'Map Color to Pen Thickness'*** *dialog*. If you set the default ***pen thickness*** to '**.4mm**' and draw a ***green line***, it will retain this ***pen thickness*** when the drawing is ***saved***. If you are using the ***'Map outline color to pen thickness'*** feature and **color #1 green** is set to '**.1mm**', the ***green line*** you drew previously will use the '**.4mm**' ***pen thickness*** in stead of '**.1mm**' set in the ***'Map Color to Pen Thickness'*** *dialog*.

This means that ***only entities with a*** '**0.00000mm**' ***pen thickness will obey the 'Map outline color to pen thickness'*** settings in **FastCAD.**

The settings in this ***'Map Color to Pen Thickness'*** *dialog* only take effect if the ***'Map outline color to line width'*** *box* is checked on the ***'Pen Thickness'*** *dialog*.

How Color Maps Are Stored, Saved and Restored

When a drawing is ***saved***, it includes a ***copy of the pen thickness color map*** settings then in effect is saved to the file '**FCW7.CMP**' in the **FastCAD7** folder. However, ***simply loading the file does not modify the current color map settings***. To use the ***current drawing's color map*** select the **[Specs > Color Map > Use Dwg Map]** command.

The **[Specs > Color Map > Use Dwg Map]** command **copies the drawing's color map to your *current settings***, and ***will be used until you change them***. It is best to save your ***standard color map settings* once**, and each time after you ***edit*** them. In that way, you can use **[Specs > Color Map > Use Dwg Map]** command to use settings appropriate to a particular drawing, select the **[Specs > Color Map > Load Saved]** command ***to restore your standard settings.***

Pen Thickness window Specifies the ***pen thickness*** to map for the ***selected color.*** The ***thickness*** you choose is in ***paper scale***; it will ***not scale up or down with the drawing print scale***. When you click on a color its ***current setting*** is displayrd in this ***value*** *window*. Change it by clicking in the window and typing a ***new value*** followed by '**mm**' for **MILLINETERS**. If you do **NOT** type the '**mm**', the default value will **NOT**change when you click the ***SET*** *button*..

Set Accepts the ***entered value*** as the ***pen thickness*** for the ***selected color.***

Print Black Forces printing of the selected color to ***black***. This display is **NOT** affected by this option. Only the ***print preview*** and ***actual plots*** will show selected lines i**n *black*.**

Color list Select a color from this scrolling list to assign a ***new Pen Thickness***. The actual drawing color is shown in the first small box, the plotting color is in the adjacent small box, followed by its palette number and the ***current Pen Thickkness***. The plotting color is only affected when the **Print Black** option is selected for that color. The colors **0-255** correspond to the **256** available drawing colors. Use the **[Specs > Select Color]** command to ***see*** or ***modify*** the ***palette***.

Load Color Map

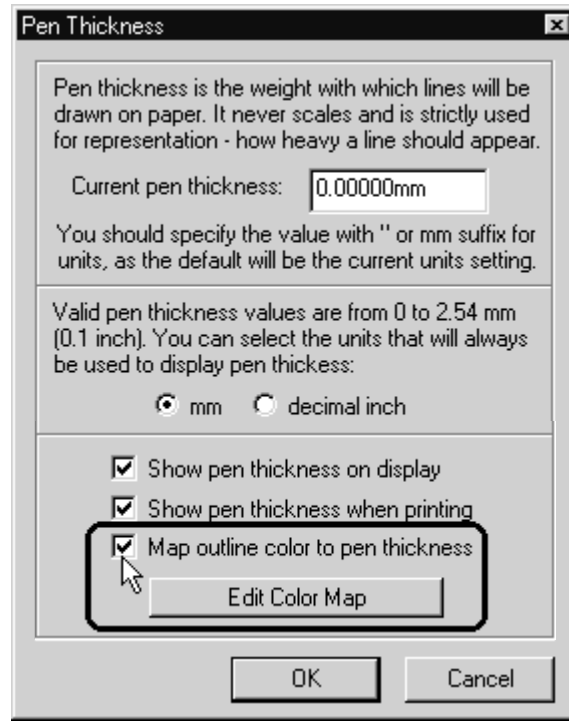
The **[Specs > Color Map > Load Saved]** command ***loads a color map saved*** with the **[Specs > Color Map > Save Current]** command. The file named **'FCW7.CMP'** ***is automatically created and saved to the*** **\FastCAD7** ***home folder with the*** **[Specs > Color Map > Save Current]** command. The **[Specs > Color Map > Load Saved]** command r***eads the values*** from the **'FCW7.CMP'** f***ile into the current color map settings***, which continue to be saved in the registry.

This command is helpful for restoring your regular settings after you've used the **[Specs > Color Map > Use Dgw Map]** command to temporarily update your color mapping.

To ***view*** or ***edit the current color pen thickness mapping***, click in the ***'Pen Thickness'*** *status window* or select the **[Specs > Pen Thickness]** command and click ***'Edit Color Map'*** *button* to display the ***'Map Colors to Pen Thickness'*** *dialog*.

Keep in mind that ***color map settings*** only take effect if the ***'Map outline color to pen thickness'*** *box* is checked on the ***'Pen Thickness' dialog*** as seen below.

Pen Thickness dialog box



Text equivalent: ***LOADCMAP***

Save Color Map

The **[Specs > Color Map > Save Current]** command writes the current color map settings to the file **'FCW7.CMP'** in the **\FastCAD** home directory. ***Use this command to preserve your preferred or default settings should you change them***. ***Restore your color map pen thickness set*** by using the **[Specs > Color Map > Load Saved]** command.

To ***view*** or ***edit*** **the** ***current color map settings*** select the **[Specs > Pen Thickness]** command and click ***'Edit Color Map'*** *button* to display the ***'Map outline color to pen thickness'*** *dialog*.

Keep in mind that ***color map settings*** only take effect if the ***'Map outline color to pen thickness'*** option is checked on the ***Pen Thickness dialog*** seen above.

Text equivalent: ***SAVECMAP***

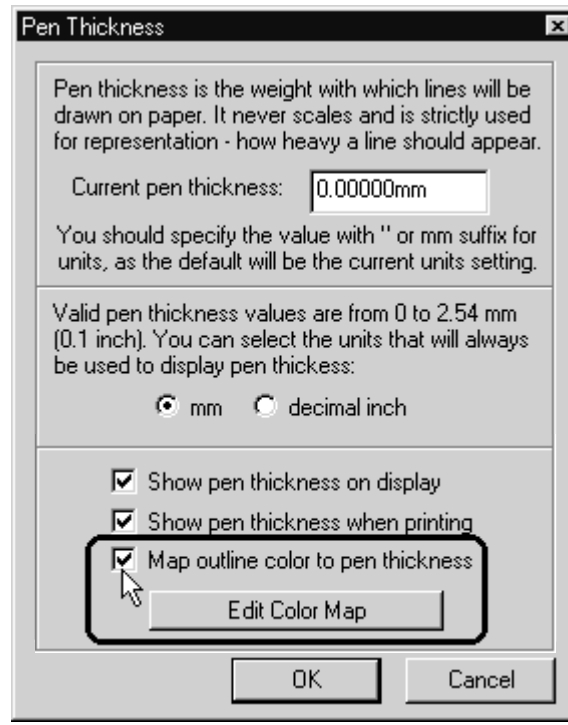
Useing a Drawing's Color Map

The **[Specs > Color Map >Use Dwg Map]** command ***loads and uses the active drawing's stored color map values for automatically mapping various pen thicknesses*** values to entities drawn with ***specific colors***. Unless the **[Specs > Color Map >Use Dwg Map]** command is used, **FastCAD** will continue to use the ***color map settings stored in the registry***.

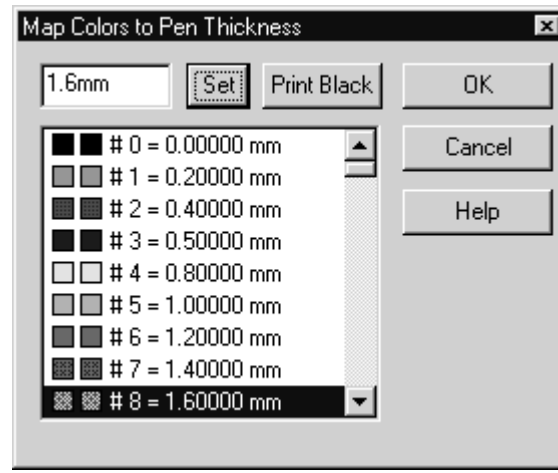
Note that using the **[Specs > Color Map > Use Dwg Map]** command ***will overwrite the current registry values, and all subsequent drawings will use the new settings***. To prevent accidental loss of your preferred or default settings, use the [**Specs > Color Map > Save Current]** command before using **[Specs > Color Map > Use Dwg Map]** command and use the **[Specs > Color Map > Load Saved]** command afterward to ***restore the colpr map values.***

To ***view*** or ***edit*** the ***current color map values***, click in the ***'Pen Thickness'*** *status window* or select the **[Specs > Pen Thickness]** command and click the ***'Edit Color Map'*** *button* to display the ***'Map Colors to Pen Thickness'*** *dialog*.

Pen Thickness dialog box



Map Color to Pen Thickness dialog box



Keep in mind that ***color map settings only take effect if the 'Map outline color to pen thickness' box*** is checked on the ***'Pen Thickness' dialog*** as seen on the previous page.

Text equivalent: ***DWGCMAP***

Dimension Style

Dimension Style icon

The **DIMENSION STYLES** command displays the ***Dimension Style*** *dialog box*, which allows you to set formatting options for dimensions you are about to draw. In other words, changes made with this command only affect new dimensions, not existing dimensions.

All dimension entities possess a *dimension style* entity property.

Dimension properties can be applied as a set using ***named dimension styles***, but if desired, each property can be modified independently from a defined style.

Named ***dimension styles*** are not entity properties. Rather, they only ***define a set of dimension specifications*** for ***new dimensions.*** That is why, for instance, you cannot select entities by ***dimension style***.

To set the text color different from the rest of the dimension, use the **CHANGEC2** command **[Edit > Change > Fill (2nd) Color]**, and apply it to the selected dimension(s). Only the text will redraw with the secondary color. The leaders, extension lines, and arrowheads will remain the same as the primary color.

Formatting options you can control include:

- **Arrowheads:** Type, Length, Height.
- **Leaders**: Formatting.
- **Options**: Tolerances, center marks, centerlines.
- **Text:** Font, placement, height, decimal places, prefix, suffix, alignment with arrows.

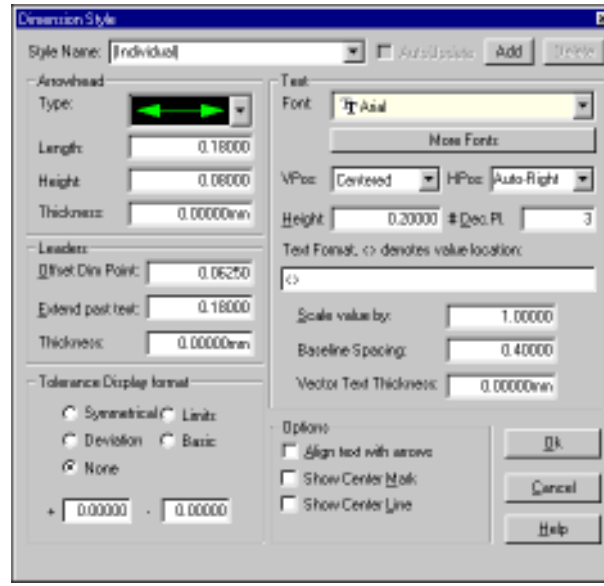
To change the dimension style for a single existing dimension:

Use the **EDIT** command **[Edit > Edit]**.

To change a collection of dimensions already drawn:

Use the **CHANGE-DIMENSION STYLE** command **[Edit > Change > Dimension Style]**.

Dimension Style dialog box



Dimension Style Names

Style Name Select a ***named dimension style*** from a drop-down list of available styles. **FastCAD** displays the format settings in the dialog.

AutoUpdate When checked, **FastCAD** automatically applies the current format to all dimensions with the same ***named dimension style.*** If you want dimensions of the same named style to have different formats, leave this option unchecked.

Add Creates a ***new dimension style***. Initially, the new style will acquire the current listed properties. As styles are added, they are auto-named #n, where n is the next ordinal number. To rename any style, click into the **Name** field and type in a new name.

Delete **Deletes** the selected ***dimension style.***

Arrowhead Properties

Type Select the **arrowhead style** from a drop-down list of available styles.

Length **Length** of **arrowhead** from endpoint to base (drawing units).

Height **Base width** of **arrowhead** (drawing units).

Leaders

Thickness **Pen thickness** of **arrowheads**.

Offset Dim Point Specifies the distance from the **edge** of a **dimensioned** entity to the s**tart of the extension line**.

Extend past text Specifies the distance of the extension line beyond the dimension line.

Thickness Pen thickness of leader lines.

Dimension Text Fonts and their Properties

Font Select a font for dimension text and numbers from the drop-list of available fonts.

More Fonts Select to add fonts to the drop-down list. For more information on specific font types, see the **TEXT PROPERTIES** command.

VPos Select placement of dimension text relative to the dimension line, for horizontal dimensions. Available choices are **Above/Centered/Below**. Centered text will

break the **dimension line**. During placement, Any of these settings can be manually overridden by holding down the **SHIFT** key.

HPos Select placement of dimension text relative to the dimension line, for horizontal dimensions. Available choices:

- **Auto-Left:** Dynamic cursor displays Dim Text in middle; but if text will not fit, the Dim Text position defaults to left. Can be overridden during placement.
- **Auto-Right**: Dynamic cursor displays Dim Text in middle; but if text will not fit, the Dim Text position defaults to right. Can be overridden during placement.
- **Center/Left/Right:** Dim Text forced to middle/left/right position, respectively. Cannot be overridden during placement (unless manually repositioned by holding down SHIFT key).

Height Specify the height of dimension text in drawing units.

 # Dec. Pl. Specify the number of decimal places to be displayed for individual dimensions drawn in this style.

Text Format Use this field to specify fixed prefix or suffix text that is appended to the true dimension value. The "<>" is used as a placeholder for the true value. If the value placeholder is omitted, the value is not included in the dimension output.

Scale value by Typically used only to deal with the **AutoCAD DWG/DXF import/export** issues. This value is used as a **scale factor**. The true **dimension** is multiplied by the **scale factor**, and the scaled value is represented by the **dimension text**. **Scale value = 1 represents actual dimension**.

Baseline Spacing Specifies the **automatic spacing between BASELINE DIMENSIONS** (**drawing units**).

Vector Text Thickness **Pen thickness** of text. Only applicable if **.FNT** or **.SHX** fonts are used (**not TrueType®**).

Options

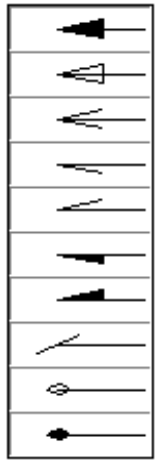
Align text with arrows Check to align text with the dimension line, regardless of orientation. If not checked, dimension text will always display horizontally.

Show Center Mark Check to display center marks, which are small crosses **FastCAD** creates at the center of dimensioned circles or arcs.

Show Center Line Check to display center lines, which consist of alternating short and long dashes. **FastCAD** automatically draws them as extension lines for dimensioned circles and arcs. Center marks are always created whenever center lines are drawn.

Text equivalent: ***DSTYLES***

Arrowhead Style

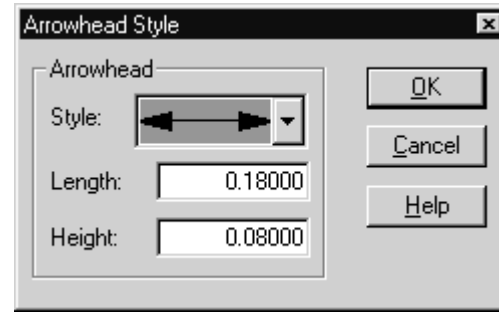


Arrowhead styles available in FastCAD

Changing the arrowhead style with the **ARROWHEAD STYLE** command changes the arrowheads on all existing entities in your drawing. If you want to add arrowheads to selected entities, use **CHANGE [Edit > Change]**.

This command does not affect dimension arrowheads in any way. To manage arrowheads on dimensions, use the **DSTYLES** command **[Specs > Dimension Style]**.

Arrowhead Style dialog box



To use, select an arrowhead type from the drop list, and edit the length and/or height of the arrowhead as desired.

Style Select the arrowhead style from the drop list. The arrowhead style you select will be applied to currently selected entity.

Length Sets the length of the arrowhead in drawing units. If you edit this value, the height will automatically update to maintain the arrow's aspect ratio.

Height Sets the height of the arrowhead in drawing units, measured as the total height of *both* arrowheads. When the **Length** value is changes, **Height** updates by the same factor to maintain the arrow's aspect ratio. You can override this suggested value by entering a different height.

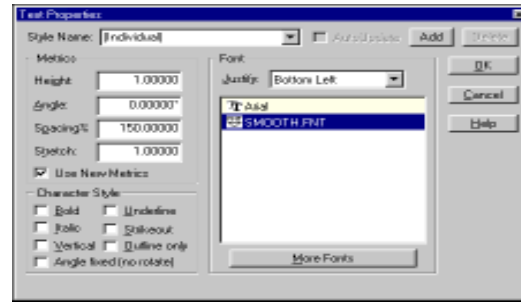
Text equivalent: ***ASTYLE***

Text Properties

TEXT PROPERTIES displays the ***Text Properties*** *dialog box* with choices for height, angle, line spacing, stretch, justification, and fonts for text you are about to create.

To change a single existing text entity (either the text or properties), use the **[Edit > Edit]** command. To change the text properties for any number of existing text entities, use **[Edit > Change > Text Properties]**. To change the text value for any number of existing text entities, use **[Edit > Change > Text]**.

Text Properties dialog box



Text Styles Creation

Style Name Select a named text style from a drop-down list of available styles. **EasyCAD** displays the format settings in the dialog.

AutoUpdate When checked, **FastCAD** automatically applies the current format to all text with the same named dimension style. If you want text of the same named style to have different formats, leave this option unchecked.

Add Creates a new text style. Initially, the new style will acquire the current listed properties. As styles are added, they are auto-named #n, where n is the next ordinal number. To rename any style, click into the Name field and type in a new name.

Delete Deletes the highlighted text style.

Text Properties

Height The height of a capital letter. The default text height is 0.2 units high. To change, type a new value in drawing units (for text 1/2 unit high, type .5 or 1/2).

Angle The bearing at which text is drawn. **Positive angles are measured counterclockwise**. For best results, use angles between -90° and 90°. The default angle is **0 degrees horizontal**. To change, type a positive or negative bearing.

Spacing % The vertical spacing between text baselines. The default spacing of each new text entity is specified at 150% of its height below the last text. (If the new text is 1 inch high, it's base line is 1.5 inches below the old text, making the lines 0.5 inch apart.) To change, type a different value in percent units.

Stretch The horizontal (or along the baseline) proportion of the text entity. The default stretch is 1 unit. A value of 2 would produce text that is twice as wide as normal. To change, type a different scale value.

Use New Metrics When on, this switch enables enhanced text handling (introduced in **FastCAD version 6.02**). Text heights for **TrueType®** fonts will be more closely matched over a wider variety of typefaces. For new drawings, this option defaults to **ON**, while the option defaults to **OFF** when existing drawings are loaded. You can choose individual text entities to use (or not use) new metrics by using the **CHANGET** command.

Character Style

Style switches Select the check boxes for the desired effects for the selected **TrueType®** font. Styles may be combined to produce differing effects. For new drawings, the switches default to all off. Results of each character style (Outline cannot be displayed through this help system):

Text Position

Angle fixed When checked, the bearing angle of the selected text is locked. This prevents the text characters from rotating, even if the text entity itself is rotated. This can be handy when text is in a callout or balloon.

Justify Justify sets text baseline justification. The default justification is Bottom Left. To change, select a different option.

Font Types you can use

Available Font list **FastCAD** supports three different types of fonts - **TrueType, FastCAD DOS .FNT and AutoCAD® .SHX vector fonts**. Which one you choose depends upon your needs.

To add more fonts to the available list, click the **More Fonts** button and add fonts in the Add New Font Properties dialog box.

Windows TrueType®

Windows comes pre-loaded with a number of **bitmap** based **TrueType®** fonts, and their presentation quality is quite good. Additional fonts can be readily purchased or obtained through many sources. Since **TrueType®** fonts are available for use in all your Windows programs, you can maintain a consistent typeface throughout your drawings and other Windows produced documents. The biggest drawback of **TrueType®** fonts is the inability to precisely control the size and aspect ratio of typefaces as seen on the screen (at different zoom factors), and in printed hardcopy. This inconsistency is an innate characteristic of **TrueType®** fonts and cannot be corrected by **FastCAD**. The default font is **Arial**.

EasyCAD DOS .FNT Vector Fonts

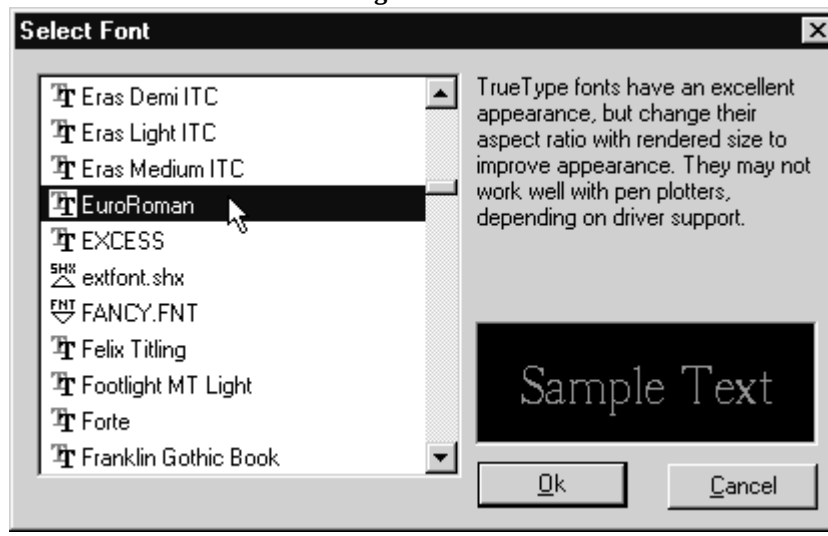
FNT Vector Fonts are font types used in the **DOS** version of **FastCAD** or **EasyCAD**. **Vector fonts** should be used when precise control over your text's size and aspect ratio is required. Text on printed circuit boards, or used as labels in boxes or balloons, are good applications for **vector fonts**. **Vector fonts** should also be used if you are printing to a **pen-type plotter**, since most pen plotters cannot print **TrueType®** fonts, especially if the text is rotated. The drawbacks to **vector fonts** are their relatively plain appearance and limited availability. **FastCAD** ships with eight basic types of **.FNT** ***vector fonts***.

AutoCAD® .SHX Vector Fonts

SHX Vector fonts are proprietary font types compatible with **AutoCAD®**.

You should use the **.SHX** font type if you are interchanging **FastCAD** files with **AutoCAD®** systems. The specified **.SHX** file must be in your **\FastCAD7 home folder** in order for these fonts to be used (they are not organic to an **AutoCAD® .DWG or .DXF** file).

Select Font dialog box



This dialog appears when you choose the ***More Fonts*** *button* in the ***Text Properties*** *dialog box* or the ***Dimension Styles*** *dialog box*.

Use this dialog to add fonts to the **available font list** in the ***Text Properties*** *dialog box*. Choose one of the fonts you wish to add. A list of all **TrueType®** fonts installed in **Windows, FastCAD .FNT** and **AutoCAD® .SHX vector fonts** that are resident in your **\FastCAD7 folder** are available in this window**.** When you select a font, the preview box shows you a sample of the typeface. Choose **OK** to add the ***selected font*** to the list, or **Cancel** to abort font addition.

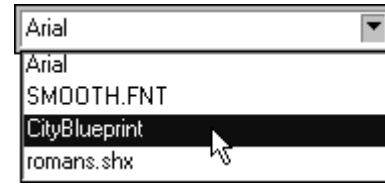
NOTE: Any font of a supported type can be used in **FastCAD**, but only after it has been "**added**" through this dialog. Any combination of font types, **.TTF**, **.FNT** and **.SHX** fonts can be used.

If you wish to use the same set of fonts in new drawings, use the **TEMPLATE** feature. Start a new drawing, add the desired **FONTS**, and use the **SAVE AS** command to save the file as a **.FT7 TEMPLATE** file. Then use the **TEMPLATE** command to make it the current **TEMPLATE** to be used when you select **[FIle > New Drawing]** to start a new drawing file.

Text equivalent: ***TSPEC***

Font

Click the drop arrow at the right end of the **FONT** ***Status window*** and select the ***current font*** from those available in the drop-down menu.



You can also type **FONT** at the command line and type a **FONT name** including its extension such as **SMOOTH.FNT**, type a font **ID#**, or right-click and select a **FONT** from the ***Select Font dialog*** that opens. All new text entities are drawn using the ***current font.***

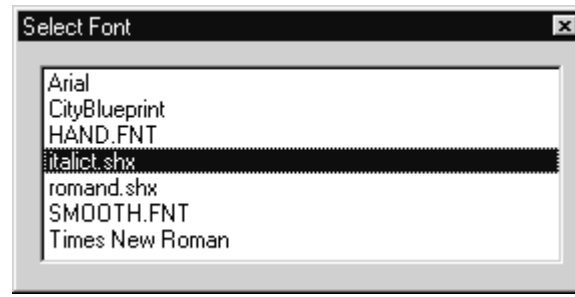
The specified **FONT** must already have been added to the ***available font list***, as displayed in the ***Text Properties*** *dialog*.

You can also left-click inside of the **FONT** ***Status window*** to open the ***'Font Management' diaog box*** or use the **[Specs > Text Properties]** command to open it to **ADD NEW FONTS** and manage ***available fonts***.

To make a **FONT active**, type **FONT** at the command line:

. The prompt reads "**Font name or # [dialog]:**". To specify the active font:

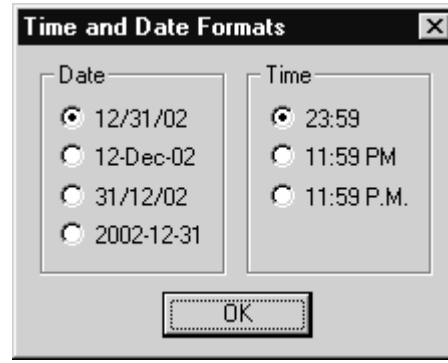
- Type a **font name** and **press ENTER**.
- Type a **font ID #** and **press ENTER**.
- Right-click to display the ***Select Font*** *dialog*. Click the desired **font name**.



FastCAD makes the specified ***font active***. The **Font** ***Status window*** is updated to reflect the change as seen below.

Text equivalent: ***TSPEC***

Specs > Time and Date



Clicking the **[Specs > Time and Date]** command opens the **TIME AND DATE FORMATS** dialog box.

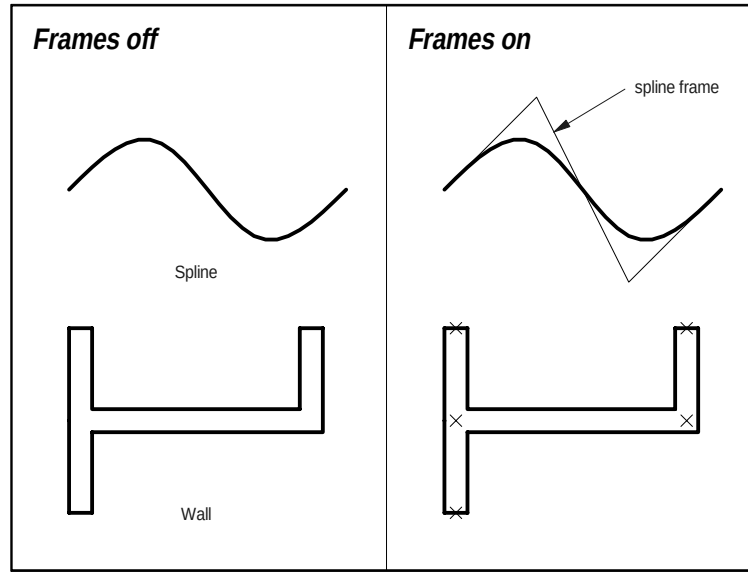
The settings in this dialog box determine the format for the **CURRENT DATE** amd **CUTTENT TIME** *text* entities that you place in the drawing by using the **[Draw > Text > Current Date]** and **[Draw > Text > Current Date]** commands. These ***text*** entities constantly update as you work on the drawing you place them in.

You may insert more than one **CURRENT DATE** and **CURRENT TIME** entities in the drawing and each may have a different format.

NOTE: If you want to freeze one of these **CURRENT DATE** and **CURRENT TIME** entities select the **[Edit > Expolde]** command and select it. It noe becomes a normal **TEXT** entity.

Text equivalent: ***TDFMT***

Toggle Frames



TOGGLE FRAMES displays or hides ***wall network***, ***smooth polygon***, and ***spline*** frames.

In the illustration above, the ***spline frames*** are the represented as ***angular lines***. On the ***wall network*** a ***dot*** will appear at each ***node***. ***Frames*** will appear ***light gray*** on the actual drawing screen.

When frames are on, a check mark appears on the menu showing this state.

By default, **CTRL+F** toggles frames on or off. You can also type the macro equivalents at the command prompt to toggle frames display (see the macro box below).

Text equivalent: ***FRAMETOG*** *or* ***CTRL+F***

Toggle Crossshairs

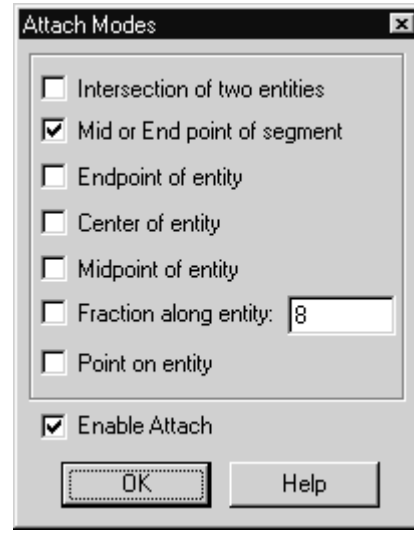
TOGGLE CROSSHAIRS enables or disables the full-screen crosshair cursor used for second and subsequent point (or entity) selections. The current state of the crosshairs cursor is saved in the Windows registry.

When crosshairs are on, a check mark appears on the menu showing this state.

Text equivalent: ***XHTOG*** *or* ***CTRL+T***

Attach Mode

Attach Mode dialog box



Attach Button ON

ATTACH MODE displays the ***Attach Modes*** *dialog* that allows you to set the active modifier(s). The ***active modifiers*** work with the **ATTACH** function and is only active when the **Attach button** is **ON**.

When multiple modifiers are selected, **FastCAD** process modifiers in the order encountered in the dialog. If both **EPT** and **MID** are enabled, the closest one of the two is used.

Attach Button OFF

Click the ***Attach*** *button* to activate this feature.Also, to display the dialog at any time, you can right-click the ***Attach*** *button.*

FastCAD allows you to select any combination of seven modifiers:

Intersection of two entities Uses the **INT** modifier when an intersection is picked. Note that the attach form on the intersection modifier works in single-pick mode. That is, the pick box must enclose an intersection. This form on **INT** does not prompt for two entities as with the regular **INT** modifier. If no valid intersection is picked, **FastCAD** tests the pick against the next checked modifier in this dialog.

Mid or End point of segment Uses the **MIDS** and **EPTS** modifiers when an entity is picked, using whichever is closest to the pick point. Note that these are the segment midpoint and endpoints. For instance, if you pick a side of a box, the point will lock to the midpoint or endpoint of that line segment, whichever is closer, and not the midpoint of the entire box.

Endpoint of entity Uses the standard **EPT** modifier, locking to the midpoint of the picked entity.

Center of entity Uses the standard **CEN** modifier, locking to the center point of the picked entity.

Midpoint of entity Uses the standard **MID** modifier, locking to the nearest midpoint of the picked entity.

Fraction along entity Uses the specified integer value to lock to the nearest fraction along the picked entity. For instance, if you enter the value "8" and pick a line, **FastCAD** locks to 1/8th division point nearest the point you picked. This could correspond to the endpoint or midpoint of the picked entity. Note that this modifier works along the entire entity length, not the picked segment of a polygon, path, etc. Note that most of the prior listed options, by their nature, supercede Fraction along entity. To ensure successful interpretation of this option, uncheck all preceding options except Intersection of two entities.

Point on entity Uses the standard **ON** modifier, locking to the point on the entity nearest to the pick point. Note that all of the prior listed options supercede Point on entity (except for an invalid Intersection… pick). To ensure successful interpretation of this option, uncheck all preceding options.

Text equivalent: ***ATCHMODE***

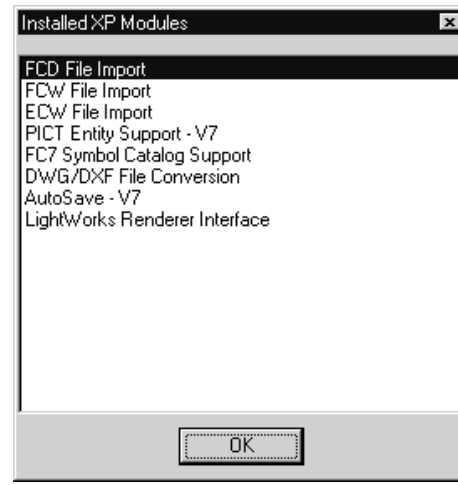
Installed XPs

eXtended Procedures are assembly-language modules that let qualified third party developers add new commands and entities to **FastCAD**. Each module can contain many commands. Unlike macros, external procedure commands run at full **FastCAD** speed and otherwise act just like normal **FastCAD** commands (they Undo, they repeat properly, modifiers work with them, and so on).

External modules can include custom associative dimensions; parametric symbols such as walls with intelligent windows and doors; real-time process control; date/time stamping; simulation modeling; pop-up calculators—almost anything programmers can imagine and tell their computers to do.

INSTALLED XPs displays the ***Installed XP Modules*** *dialog box*. All loaded **XP**'s are listed. Choose the ***About*** *button* to get additional information on the selected module. This information is provided by the author of the individual **XP** commands. A list of commands found in the **XP** text screen along with a short description of each function should appear.

Installed XP Modules window



Choose the OK button to close the dialog box and return to the drawing screen.

To Install an XP:

XP installation is automatic when a valid **XP** module (with ***.DLL** filename extension) is in the **FastCAD** folder.

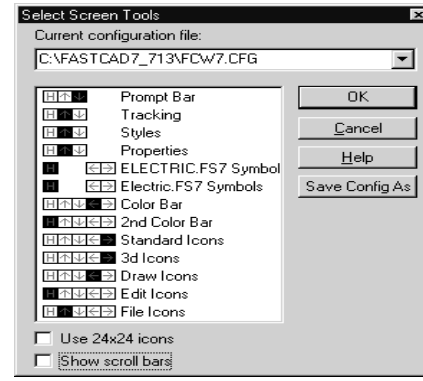
To Remove an XP:

To unload an **XP** module, remove the **.dll* file from the FastCAD folder.

Text equivalent: ***XPCFG***

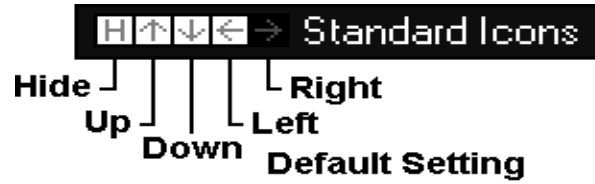
Screen Tools

TOOLS allows you to customize **FastCAD**'s appearance. Any of the available screen settings can be toggled on or off, or repositioned.



The exact number and names of each control bar can vary depending upon your configuration. The illustration below shows the default configuration:

Display/Position controls Click directly onto each check box to control visibility and positioning. The active or "**on**" setting is highlighted in red/black, while the available "**off**" options are highlighted in white/gray.



Prompt bar Located along the bottom edge, it contains the command prompt and button controls.

Tracking Contains cursor tracking, font, text height, pen thickness, and line width status controls.

Styles Contains text style, dimension style, workplane, and sheet status controls.

Properties Contains working drawing, layer, line style, and fill style status controls.

Sym.FS7 Symbols Symbol catalog viewer, using the specified symbol library file.

Electric.FS7 Symbols Symbol catalog, using the specified symbol library file.

Color Bar Color Bar, as configured.

2nd Color Bar Color Bar, as configured.

Standard Icons Contains zoom and spec icons.

3d Icon Bar Contains 3d viewing/drawing icons.

Draw Icons Contains 2d drawing icons.

Edit Icons Contains 2d editing icons.

File Icons Contains file, bookmark, and search icons.

Use 24x24 Icons Click this box to display the larger 24x24 pixel Icons.

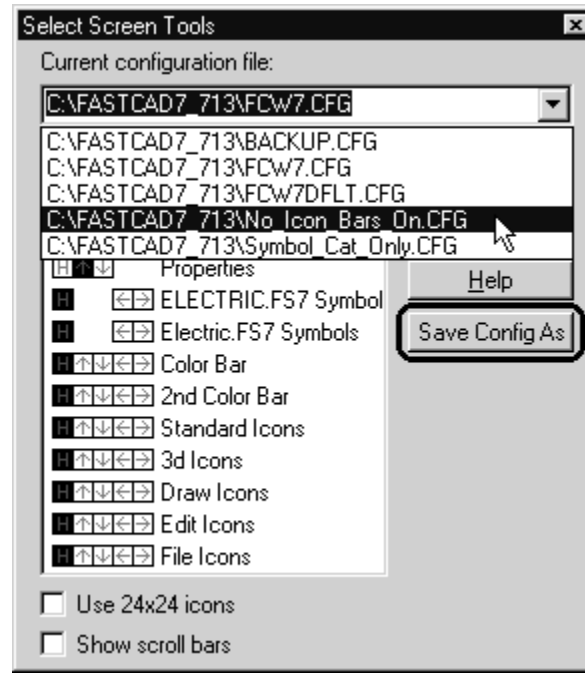
Show Scrool Bars Check this box to display the vertical and horizontal Scrool Bars in each drawing window

Text equivalent: ***TOOLS***

Loading a Custom .CFG Screen Configuration File

The **LDCFG Macro** command allows you to ***load a custom* .CFG *file*** saved from the **'Select Screen Tools'** dialog box as seen below.

Select Screen Tools dialog box



In the '**Select Screen Tools'** dialog box above the '**HIDE**' boxes have been selected to turn off '**All Icon Bars** and **Symbol Catalog Viewers**' to give the maximum screen area to display the drawing. The '**Save Config As**' button was clicked and the configuration file was saved as '**No_Icon_Bars_On**'. The file is automatically saved with the **.CFG** filename extension**.**
Any **.CFG** screen configuration file may be reloaded from the **'Current Configuration File'** windows pull down menu**.**
You may alternatly use the **LDCFG Macro** command and type the **'filename.cfg**' from the keyboard to load any **.CFG** file without having to open the '**Select Screen Tools**' dialog box. It may also be added to a ***custom menu selection*** or ***run from a* Macro.**

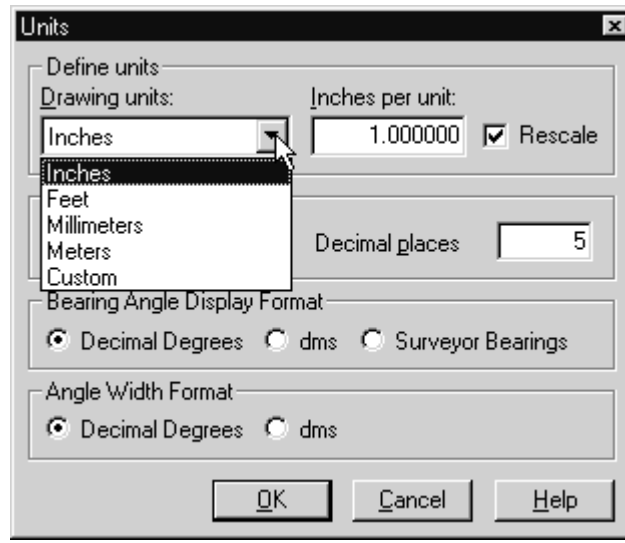
See the **FastCAD Help Topic 'Macros > Sample Macros > Load a Custom .CFG Screen Configuration File from a Menu or Macro' for more details.**

Macro syntax - LDCFG <filename.CFG>
Text equivalent: ***LDCFG***

Units

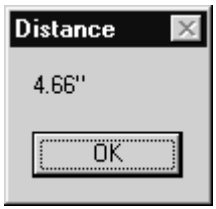
The **UNITS** command tells **FastCAD** how to relate its **internal unit system to the real world**. You can define **FastCAD**'s **UNITS** as anything you wish. When you start a drawing, the default **FastCAD UNIT equals one inch in the real world**. **Inches**, **feet**, **millimeters**, and **meters** are ***predefined shortcuts*** for changing to some commonly used **UNITS**. You can define other **UNITS**, such as **miles** or **microns**, and assign them to the **CUSTOM UNITS** definition. **UNITS** should be defined before you start to draw. Otherwise, you may have to ***scale individual entities later on***.

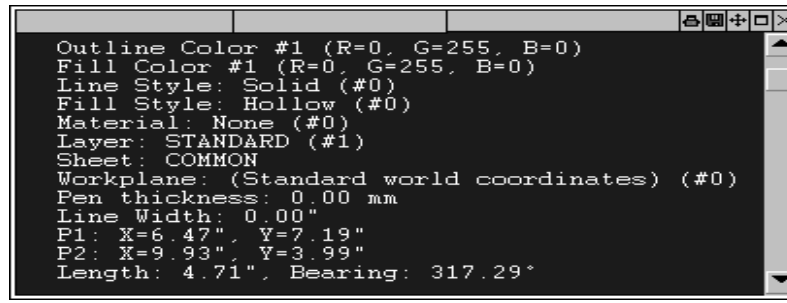
Inches per unit This is the conversion factor **FastCAD** needs to calculate **Dimensions** and ***print your drawing to scale***. Enter a new number of i**nches per unit**. For example, **to define each UNIT** as **one mile**, type "**63360**" (**5280 feet per mile times 12 inches per foo**t).



Decimal places This window allows you to change the **number of decimal places** that the **[Indo > Distance]** command displays in its on-screen dialog box and the **[Info > List]** command displays in the **LIST** window for **LENGTHS** of **LINES** and their **LINE WIDTH** and **BEARING ANGLES**.

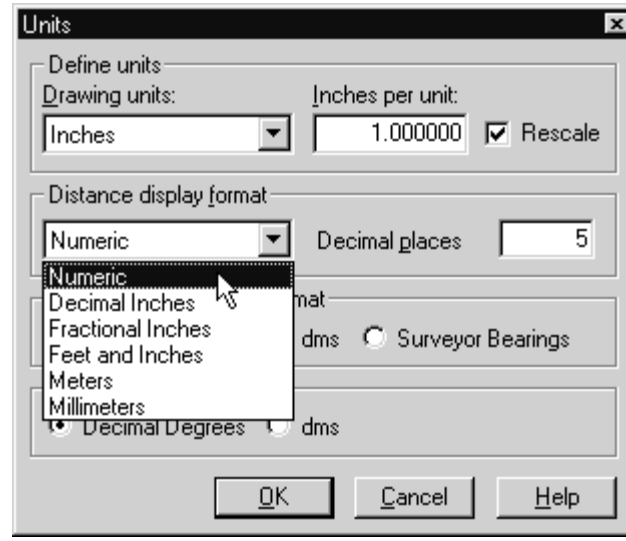
This setting has ***no control over*** the **number of decimal places** for **DIMENSIONS.**





Distance Display Format

UNITS dialog box



The setting in the '**Distance display format**' window determines how your **DIMENSIONS**, as well as the **DISTANCE** is calculated and displayed when using the **DIMENSION** commands or the **[Info > Distance]** command.

In the **DIMENSIONS** illustrated below the **UNITS** are set to **INCHES**. The **NUMERIC** '**Distance display format**' shows **509.504** which is **INCHES** since the **UNITS** are set to **INCHES**.

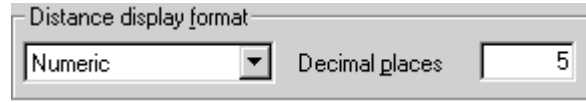
The other '**Distance display format**' settings show the **59.504 INCH DIMENSION** in each of the other equivalent formats. The only thing that was changed after the **DIEMSNION** was drawn was to select the **[Specs > Units]** command and select a different '**Distance display format**' option.

If you have a drawing and the '**Distance display format**' was set to '**Decimal Inches**' for **DIMENSIONS** and you need to give the drawing to someone who wants to see the **DIMENSIONS** in the equivalent of **MILLIMETERS** just select the **[Specs > Units]** command and select the '**MILLIMETERS**' option from the '**Distance display format**' window.

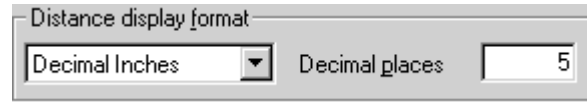
You '**DON'T NEED TO CHANGE THE UNITS VALUE TO MILLIMETERS**', just select **MILLIMETERS** from the '**Distance display format**' window in the **UNITS** dialog box.

The '**Distance display format**' may be set for the following:

Numeric format – **59.504** -- **Numeric Units** value displays with no **'**, **"**, **mm** or **m** designator. The **DIMENSIONS** numbers would display the same no matter what the **UNIT** value is set for.

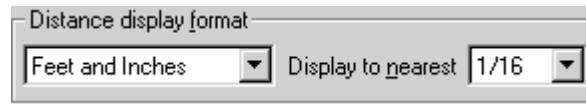


Decimal Inches format – **59.504"**

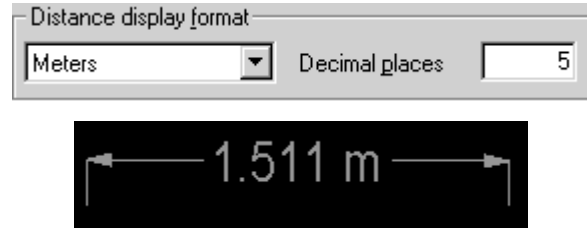


Fractional Inches format – **59 1/2"**

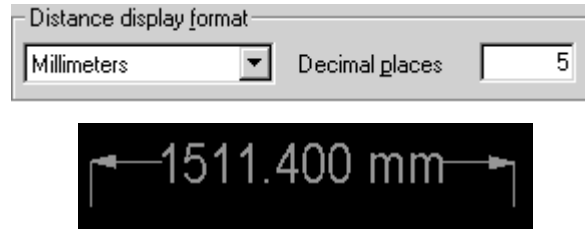
Feet and Inches format – **4'-11 1/2"**



Meters format – 1.511 m



Millimeters format – 1511.400 mm



NOTE: If you want **DIMENSIONS** to display as '**Feet & Inches**', you do **NOT** need to change your **UNITS** from **INCHES** to **FEET**. just select **'Feet and Inches'** from the **'Distance display format'** window in the **UNITS** dialog box**.**

Rescale Units

When ***checked*** (**default state**), ***changing the drawing*** **UNIT** setting will cause all entities ***scale so that they retain their same***, ***absolute size***.

For example, if **RESCALE** is ***checked***, a **line** with a **'1.0 inch length'** will **DIMENSION** as a **'25.4 mm'** if the **UNITS** are changed to **MILLIMETERS**. This is the number of **MILLIMETERS in an INCH.** In effect, ***the drawing will remain unchanged except for the drawing*** **UNITS** ***display in a*** **DIMENSION** ***or*** **DISTANCE** ***command***.

Remove ***the check from the*** **RESCALE** ***box then change the drawing*** **UNIT** ***setting and the entities will be*** **RESCALED** ***by the anount of the*** **UNIT** ***value***.

For example, when **RESCALE is UNCHECKED** and the **UNITS** are changed to **MILLIMETERS**, a **'1.0 inch line'** will **DIMENSION** as a **'1.0 mm line'**. All other ***scaleable properties*** will use the ***redefined*** **UNITS and will be resized by that value**.

Note: The **RESCALE** option is ***always checked*** (**on**) ***unless you explicitly uncheck the control and change the drawing*** **UNITS** ***to resize the entities.***

Using the RESCALE option to rescale an Entire Drawing.

If you get a drawing in **AutoCAD .DWG or .DXF** format and you measure the length of a **line** and it is supposted to be **'10 feet long'** but its **length measures '10 inches'**, you know that it is **'12 times to small'**.

Instead of using the **[Edit > Scale > Scale value]** commad to **resize** the ***whole drawing***, **remove** the check from the **RESCALE** box then select **'Feet'** in the '**Drawing Units**' window and click OK.

Now if you draw a **DIMENSION** on the **line** that measured **'10 inches'** before the **RESCALE,** it will now display as a **'10 foot long line'**.

Dimension Bearing Angle Distance Format

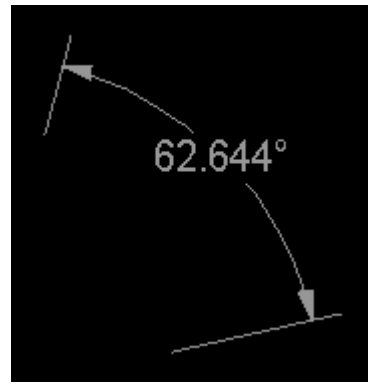
The **BEARING ANGLE DISTANCE FORMAT** setting determins display when you select the **[Info > Bearing]** command. You are prompted to pick two points and the **BEARING ANGLE** between them is displayed in a dialog box.

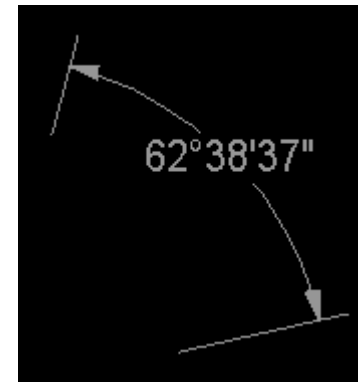
Decimal Degrees **Degrees Minutes Seconds** **Surveuors Bearings**

Dimension Angle Width Format

The **BEARING ANGLE WIDTH FORMAT** setting determins the display of **ANGULAR DIMENSIONS** when you select **[Dimensions > Full Angular]** and the **[Dimensions > Quick Angular] DIMENSIONS** commands.

Decimal Degrees **DMS Degrees, Minutes and Seconds**



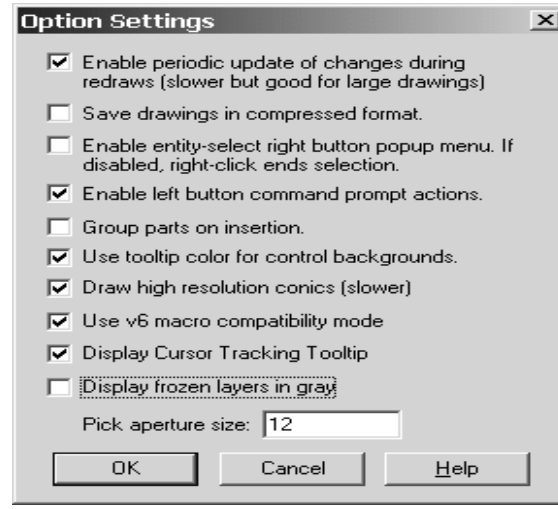


Options

The **OPTIONS** command lets you configure certain **FastCAD** screen and setup options. Invoking the command displays the ***Options dialog box***. Simply select to enable or disable available options.

Other display options such as *displaying* or *hiding* the ***Icon*** *bar*, ***Status*** *bar*, and ***Color*** *bar*, can be configured using the **SCREEN TOOLS** command, selectable from either **[Specs > Screen Tools]** or by choosing the ***Configure Screen Tools icon***.

Options Settings dialog box



Enable periodic updates of changes during redraws (slower but good for large drawings) This setting is recommended.

Save drawings in Compressed format Choose to enable automatic .**FC7** file compression. File compression is on the order of **50% of the original size** (actual rates vary depending upon the file content). Using this option can cause longer file saves. ***Opening compressed files is automatic and transparent***. Current versions of **FastCAD v7** can open all compressed .**FC7** files and open without any additional software or utilities. However, versions of **FastCAD** prior to **v6.03 cannot open compressed files.** This setting is saved in the registry. Use this option if you are transmitting drawing files over the Internet, or saving to limited capacity media.

Enable entity-select right -button popup menu Choose this option to enable the ***Entity Selection*** *popup menn* after a right-click when you are prompted to select entities..This presents a ***popup menu*** with the **'DO IT'** command at the top and additional entity selection methods such a by ***and***, ***not***, ***layer***, ***color***, ***entity type*** *etc*..If this box is ***unchecked*** and you have selected entities by ***each*** or ***window***, a right-click will immediatly process the command. If you had selected the **ERASE** command and picked some entities, a right-cliak would **immediatly ERASE them**. You don't need to click the **'DO IT'** command.

Enable left button command prompt actions **Enabled,** the **left button**, like the **right**, when clicked at the command prompt, will **repeat the prior command**.

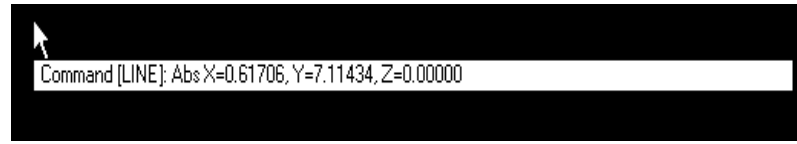
Disabled, an **XP, which is a custom add-on module,** can intercept the left click at command prompt and perform a **custom action**. Unless you have a **custom XP** add-on installed that uses this, you will see no difference when **Enabled** or **Disabled** .

Group parts on insertion Check this box if you want drawings that you insert with the **INSERT PART** command to automaticall be **GROUPED**. Remember that if the drawings you are inserting have **GROUPED** entities already, this will create an additional nested **GROUP**. If you need to use single entity edit commands on **GROUPED** entities you wil need to click the ***Locked*** *button*, edit the entities and then click the ***Unlocked*** *button* again to lock the **GROUPS**. If this box is unchecked, inserted drawings will **NOT** be **GROUPED**.

Use tooltip color for control background This allows the Windows tooltip background color to apply to the ***status*** *bars* and command line. This can be useful in making the text in these windows stound out. You could set your Windows ***tooltip text color to white*** and the ***tooltip background color*** *to* ***black***. This makes the text much easier to read.

Draw high resolution conics (slower) **Enabled** this makes the **CIRCLES, ARCSW** and **SPLINES** look much smoother on the screen. Screen refreshes will be slower on older video cards and computers especially if you have a large number of these entities in the drawing. The visual quality of these entities on the screen has no effect on the printed output. The quality of the printed drawing is totally dependent on the printer drivers resolution setting.

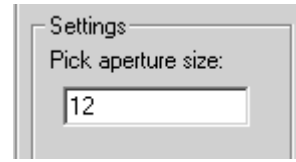
Display Cursor Tracking Tooltip This turns on the floating command line with the ***xyz tracking*** and the ***command prompts*** for the ***current command*** and follows the cursor arrow as you move it around the screen.



Because the command line prompts are displayed at the cursor location you can focus your attention there and not have to look at the command line at the bottom of the screen.

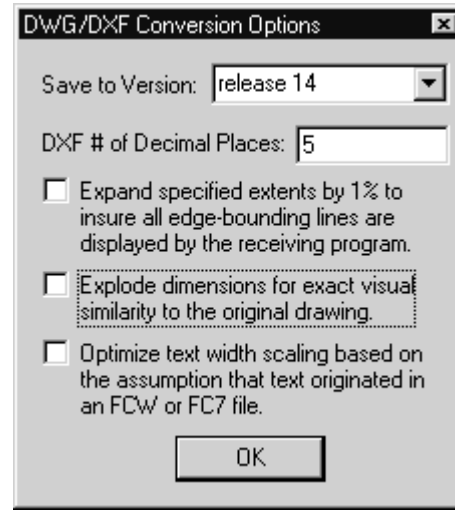
Display frozen Layers in Gray. When this **OPTION** is **ENABLED** all entities that are on ***currently frozen Layers*** will be displayed in **GRAY.** This makes it easy to knw which entities are on **FROZEN LAYERS** at a glance.

Pick Aperture The value entered here determines the pixel size of your entity selection pick box. The pick box is displayed when you select entities, usually during draw or edit commands. Changing this value is helpful if your screen resolution is finer or coarser than usual, or if extreme precision is required for your entity selections. The default aperture size is 4. The current setting is saved for all sessions until you change it again.



Text equivalent: ***OPTIONS***

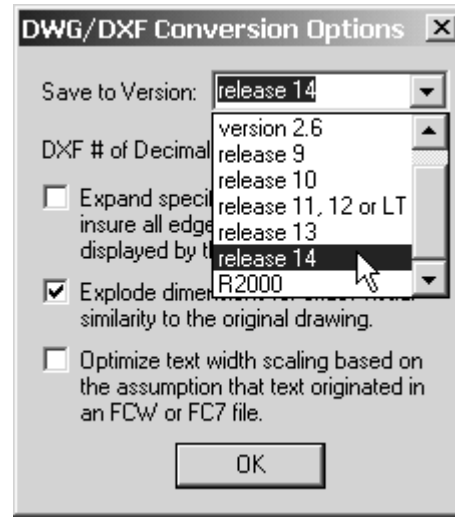
DWG/DXF Options



The **DWG/DXF** Options command allows you to save your **FastCAD** drawing to a specific version of **AutoCAD®**'s native **.DWG** or **.DXF** file formats. When you use the **[File > Save As]** command to save the drawing on your screen to a **.DXF** or **.DWG** format, it will save to the version you specify here. The option you select is saved in the registry, so it will persist for future **FastCAD** sessions.

Save to Version selection

To set the **AutoCAD® SAVE AS** version, click to expand the drop list, select the desired version, then choose the **OK** button.



By default, **FastCAD** saves to the **AutoCAD® release 11**, **12** or **LT** file format specifications. **AutoCAD rel. 14** offers the most versatility since newer releases of **AutoCAD®** can successfully import this drawing format. However, you can specifically direct the output to **release AutoCAD® 2000** file formats. **AutoCAD®** releases earlier than 11 are typically not in use, but **FastCAD** gives you the option of saving to release 2.5, 2.6, 9, and 10 specifications.

NOTE: Currently FastCAD v7 can NOT open or save to AutoCAD rel. 2004 or 2005. If you get drawings from an AutoCAD user of these versions have the save the drawing from AutoCAD as AutoCAD 2000. A later update of FastCAD v7 will support these formats.

Expand specified extents by 1% to insure all edge-bounding lines are received by the receiving program

Check this option if edge entities are not displaying properly (i.e., they are not visible) after importing into another program. This instance may occur when your image is brought in as **.DXF** clip-art to a **non-CAD** application such as word a processor.

Explode dimensions for exact visual similarity to the original drawing

Check this option to ***explode dimensions*** to component **text**, **lines**, **arcs**, **polys**, etc. Doing so will ensure that the **dimensions** will appear positionally correct after **import/export**. If not checked, **dimensions** are converted to **associative dimension** entities, but may not visually match. The following **associative dimension** entities are supported by this option:

Optomize text width scaling based on the assumprion that text originated in FC7 file

If you convert **AutoCAD® .DWG** or **.DXF** files you may consider purchasing or obtaining some **AutoCAD®** .**SHX vector** text fonts.These **.SHX vector** fonts should reside in the **\FastCAD7** home directory. When you import **.DWG** or **.DXF** files and you have the **.SHX** font file it uses, you will see them in the drawing.

If you do **NOT** have the **.SHX** font it references, **FastCAD** will substitute the **Arial TrueType®** bitmap font. When **TrueType®** fonts are substituted, the *font metrix* will change a bit each time you ***import*** or ***export*** to **.DWG** or **.DXF format**. If you use the **AutoCAD® .SHX vector** fonts, the text will make a **perfect round trip and always look the same**.

Text equivalent: ***ACADOPT***

Chapter 20

Info Commands

Getting information about your drawing

Commands under the *Info menu* calculate and display coordinates, angle, distance, and area of entities. Others count and total entities as well as provide information about them.

LIST displays entity properties and geometry for selected entities.

EXTENTS returns the drawing coordinate extents.

COUNT ALL totals all entities by type.

COUNT totals selected entities by type.

CUURDINATE displays the x,y,z coordinates of any specified location.

BEARING calculates the angle between two points.

DISTANCE calculates the distance between two points.

DISTANCE... calculates the distance between two points, returning extended info.

LENGTH ALONGTH calculates the distance along an entity.

AREA calculates the area of a specified polygon.

CALCULATE returns the soultion of an algebraic expression.

List

The **LIST** command displays the entity properties and geometry for each entity selected. For example, you can review the coordinates of an entity, the layer it resides on, its color and style characteristics, etc.

To get an entity **LIST**, select **[Info > List]**:

1. Select the entities you wish to obtain information on.
2. **FastCAD** displays a scrollable text window of information for each entity selected. Use the scroll bar to view more pages. To close the window and return to your drawing, press ESC or click the ***Close*** *icon*.

Note: Even though **LIST** displays all relevant properties of each selected entity, you cannot select or change anything in the list display. However, from the text window's title bar, you can click the ***Print*** *icon* to print the contents, or the ***Save*** *icon* to save the contents to a specified text file.

Entity properties typically returned by the **LIST** command:

- **Entity type and tag number.**
- **Outline color: # (RGB)**
- **Fill color: # (RGB)**
- **Line style:.name (#)**
- **Fill Style: name (#)**
- **Material: name (#)**
- **Lyer: name (#)**
- **Sheet: name**
- **Workplane: name (#)**
- **Pen thickness: mm**
- **Line width**
- **All geometric parameters that FastCAD uses to construct the entity.**
- **Node lists of paths, polygons, and splines.**
- **Groups, multipolys, and their sub-records.**
- **Text specifications.**
- **Area of polygons.**

For any ***symbol reference***, **LIST** displays the **name**, **insertion point**, **scaling**, and **rotation**. You cannot select ***symbol definitions*** using **LIST**. Instead, select the **LISTSYM** command **[Insert > List SymDef...]** to see list information about ***symbol definitions.***

Text equivalent: ***LIST***

Extents

The **EXTENTS** command lists the drawing extents in a dialog box. The drawing extents are the boundary coordinates (in drawing units) that would include every entity in your drawing, regardless of whether or not it were visible.

Sometimes, a drawing with very large extents, especially relative to the intended drawing area, can cause problems. For instance, if you were drawing a three-inch widget detail and you accidentally create/copy/move a rivet two miles away, your zooms and selection windows may behave erratically. Use this command to check the drawing extents.

For 2D drawings, only the X and Y values are applicable. The Z value coordinates define the vertical extents of 3D drawings.

Text equivalent: ***EXTENTS***

Count All

The **COUNT ALL** command totals all entities for an entire drawing, and displays quantities sorted by entity type.

When you invoke **COUNT ALL**, **FastCAD** displays a text window with as much information as will fit on one screen. Use the scroll bar to view more pages or close the window to return to your drawing.

Text equivalent: ***COUNTA***

Count

The **COUNT** command displays the quantity of selected entities, sorted by entity type (like **LIST** without the detail).

When you invoke **COUNT**, select entities using normal methods. **FastCAD** then displays a text window with as much information as will fit on one screen. Use the scroll bar to view more pages or close the window to return to your drawing.

Text equivalent: ***COUNT***

Coordinate

COORDINATE displays the ***x,y,z coordinates*** of any specified location in your drawing.

To get the *coordinates* of any point, select **[Info > Coordinate]**:

The prompt reads "**Point to identify:**". Pick a point in a drawing window.

FastCAD displays the x,y,z coordinates in a message box, using the current distance display format (feet-and-inches, numeric, etc.) as specified by the **UNITS** command **[Specs > Units]**. Click **OK** to clear the message.

Text equivalent: ***ID***

Bearing

BEARING calculates the angle (measured counterclockwise, from zero degrees at 3 o'clock) of an imaginary line between two points. The information is displayed in **3D** format, in both the **XY plane** (**projected 2D bearing**), and in the **ZX plane** (**elevation**).

To calculate the angle between two points, select **[Info > Bearing]**:

1. The prompt reads "**1st point:**" Pick a point in a drawing window, or type coordinates.
2. The prompt reads "**Next point [done]:**". Specify a second point.

FastCAD displays the angle from the first point to the second point in a message box. Click **OK** to clear the message.

Text equivalent: ***BEARING***

Distance

DISTANCE ***calculates*** the ***distance between two points***.

To ***calculate*** the ***distance between two points***, select **[Info > Distance]**:

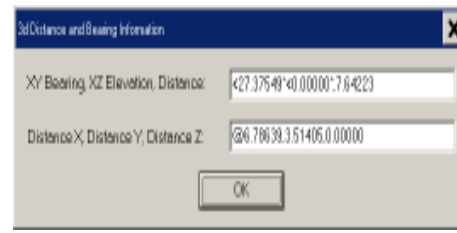
1. The prompt reads "**Distance from:**" Pick a point in a drawing window, or type numeric coordinates.
2. The prompt reads "**Distance to:**". Specify a second point. FastCAD displays the distance in a message box, using the current distance display format (feet-and-inches, numeric, etc.) as specified by the **UNITS** command **[Specs > Units]**. Click **OK** to clear the message.

NOTE: You can also use the **LIST** command **[Info > List]** to find the length of an existing line entity.

Text equivalent: ***DIST***

Distance...

DISTANCE... ***calculates*** the *distance* ***between two points***, returning the bearing, distance, and delta **x,y,z between** two specified points. This information is displayed in a dialog with edit controls so that **^C** can place the values on the *clipboard*.



Distance Information dialog box

To ***calculate*** the *distance* ***between two points***, select **[Info > Distance…]**:

1. The prompt reads "**1st point:**". Pick a point in a drawing window, or type numeric coordinates.
2. The prompt reads "**Next point [done]:**". Specify a second point. **FastCAD** displays the ***bearing***, ***distance***, and ***x,y,z distance*** in a message box, using the current ***distance display format*** (**feet-and-inches, numeric**, **millimeters** etc.) as specified by the **UNITS** command **[Specs > Units]**. Click **OK** to clear the message.

Information in this dialog can be highlighted and copied to the *clipboard* by pressing **CTRL+C**. It may then use the **[Clip > Paste]** command or **CTRL+V** to place it into the drawing as regular **TEXT** entities.

Text equivalent: ***DIST2***

Length Along

LENGTH ALONG calculates the distance along an entity between two points on that entity.

To measure the *length along* on an entity, select **[Info > Length along]**:

1. The prompt reads "**Select entity:**" Select an entity with the pick cursor.
2. The prompt reads "**From point on entity [all]:**" Specify a starting point. If you right-click or press ENTER, the entire length of the selected entity will be displayed.
3. If you selected a point on the entity, the prompt reads "**To point on entity:**" Select the ending measurement point on the entity.
4. **FastCAD** displays the distance in a message box, using the current distance display format (feet-and-inches, numeric, etc.) as specified by the **UNITS** command **[Specs > Units]**. Click **OK** to clear the message.

Text equivalent: ***LENGTH***

Area

AREA calculates the *area* bounded by an imaginary polygon, a real polygon, a smooth polygon, a circle, or an ellipse.

To measure the *area* on your drawing, select **[Info > Area]**:

1. The prompt reads "**1st point [enter to select]:**"
   - Right-click or press **ENTER** to select a *circle*, *ellipse*, or *poly* entity with the pick cursor. **FastCAD** displays the *area* in a *message box*. Click **OK** to clear the message.
   - Specify a start point for a polygon used to describe the bounded area you are measuring.
2. If you picked a point with the mouse, the prompt reads "**Next point:**". Specify a second point, then continue picking points around the area you want to measure.
3. Right-click to complete the imaginary polygon. **FastCAD** displays the area in a message box. Click **OK** to clear the message.

Text equivalent: ***AREA***

Calculate

The **CALCULATE** command accepts an algebraic expression, including parenthetical expressions, and returns the solution value in a message box.

Text equivalent: ***CALC***

Chapter 21

Macro Commands

Autmoate your tasks with simple macro or scripts

You can use **FastCAD**'s macro facility to increase your efficiency, especially if you use repetitive and complex command sequences to accomplish tasks. A macro is simply a series of **FastCAD** commands that are performed in sequence. But instead of typing the commands, they are run automatically from a script you write. A macro can be very simple or very complex.

Technically, here is how macros work: **FastCAD** keeps a lookup table or dictionary of macro names and their definitions. Whenever **FastCAD** gets a term (a word or value ended by a delimiter) from your typing, from a script, or from a menu, it checks the macro table to see if that term has been defined as a macro name. If it has, **FastCAD** replaces the macro name with the stored text (the macro code you have written).

You can store text in a macro name, either by defining a name with the **Macro** and **Endm** (**End Macro**) commands, or by using the name as a destination variable for a command like **GV** (**Get Value**). The stored text can be:

- A single value (for example, **2** stored in the name **RED**);
- A renamed command (for example, **DSTYLE** stored in the name **Dims**);
- A command with one or more parameters (for example, the name **Square** could contain **Rpoly 4** to let you start a 4-sided regular polygon by typing "Square");
- A series of commands making up a macro program.

You can use text equivalents of **FastCAD**'s menu commands in a macro program. You can also use the special macro programming commands explained in the ***Macro Command Reference*** (reference **FastCAD**'s on-line **Help facility**). These special commands let you write programs that:

- Ask the user to enter points, values, text, and other data (prompting for user input);
- Store the data in variables;
- Use the stored data in drawing and editing operations;
- Make calculations based on stored data (calculating with variables);
- Compare values in variables and make decisions based on the results (controlling program flow);
- Jump to a label within the macro or repeat a series of commands (controlling

program flow);

- Call other macros as subroutines, for up to 16 levels of nesting.

You do not have to write programs to use the special macro commands. For example, if you want to draw several lines radiating from a common point, you could use **GP** (**GET POINT**) to store the point in a *function key*, then press the key to enter the first point of each line. Using **Macro** commands from the keyboard can help you learn how they work. If you intend to write complex macros (using ***branching***, ***looping***, and so on), this discussion assumes that you are already familiar with basic programming concepts.

Quick Primer on Macro Creation

About Delimiters

A **DELIMITER** tells **FastCAD** where a command or parameter **ENDS**. A ***delimiter*** can be a **SPACE** ( ); a **SEMICOLON**(**;**); the **equal sign** (**=**); the **ENTER** key (**carriage return**); the **TAB** key; or the **right mouse button**.

When you **Define Macros**, you should keep these rules in mind:

1. Some commands ***require an extra delimiter*** before your **Macro** can continue to the next command. These include:
   - Commands that ***automatically repeat*** or continue until you press the right mouse button to end them. Examples include many **DRAW** commands.
   - Commands like **DISTANCE** (in the *Info menu*) that display an alert box.
2. **SPACEBAR** or **TAB** are always a ***delimiter*** except in the parameters (responses) that follow these commands:
   - **^D prompt** (Some text that asks for some input).
   - In values to be entered in editable dialog-box items.
3. **SEMICOLONS** are always a ***delimiter*** inside a **Macro definition**. Because a **SEMICOLON** does act as a ***delimiter*** in a **Script** or **Macro**, it is impossible to include ***literal semicolons*** in **Macro** parameters.
4. **ENTER** (**carriage return**) and the ***right mouse button*** only work from the keyboard. **FastCAD** "**counts**" the ***carriage returns that end each line in a*** **Macro**, **Menu**, or **Script** file.

About Function keys

A **function key** (for example, **F2**) is a special kind of macro name. The **function key F1** is special: it provides context-sensitive help anywhere in the program and cannot be used for anything else. Also, the **F10** key is used by Windows to access the menu bar, as is the **ALT** key. **FastCAD** recognizes the rest of the function keys, the **10 standard function keys** (**F2-F9**, **F11**, **F12**) and the same keys combined with the **CTRL** key, and/or the **SHIFT** key. You can refer to a ***function key*** by its name (**F2**, **CF1**, **SF6**, **SCF9**, etc.), or just press the key or key combination.

1. To ***define*** a ***function key*** in a **Macro** file, enter Macro, then the name of the ***function key*** (**F4**, **SF1**, etc.), then the text you want to store in it.
2. Enter "**Endm**" on a line by itself to end the definition. Function key variables are also useful when you are working within **FastCAD** for storing a point or a value to use over and over during a work session. Define the function key with the **Macro** and **Endm** commands, or use a **Get** command like **GP** (**Get Point**) to store data in a function key.

NOTE: In **FCW7.MNU**, several function keys have been predefined to select modifiers. For example, **F4** enters the **CEN** (**Center**) modifier. Several **CTRL**-alpha keys (**^A**, **^S**) are also defined there.

Programming macros

Unlike true programming languages, a **Macro** language does not know the difference between a **Macro** used as a ***program***, a **Macro** used as a ***variable*** (a point, value, or other data), and literal text (a **FastCAD** command, a file name, or text to be drawn). It is up to you to avoid name conflicts.

Terms that would be **Macro** expanded can avoid expansion by preceding them with an **exclamation mark** (**!**). This lets you define a new **Macro** called **Color** that can call **COLOR** (**FastCAD**'s color command) inside itself, yet use the original command.

Within a macro table, make sure that each ***variable***, ***label*** and **Macro name** you use is unique. Also, never use a **FastCAD** command (text equivalent) as a macro name (unless you want to redefine what that command does). Never use numbers as macro or variable names. Try to avoid using file names. If you write macros for other users, consider starting all **macro**, **label**, and **variable name**s with a **dollar sign** (**$**) or some other unlikely character to avoid name conflicts.

Variables are never further expanded. A value defined with **macro...endmacm** will be expanded as long as necessary.

To ***run*** a **Macro**, ***type its name***. You can also run a **Macro** by including it in a ***customized*** **Menu**. But remember, a **Macro** will run only if its ***macro definition*** is loaded (as a **.MAC** file) along with the **Menu** (**.MNU**).

A **Macro** can call another **Macro** by ***name***, effectively running it as a subroutine. **FastCAD** allows up to 16 levels of nested **Macros**. When **FastCAD** finishes performing the text in the called **Macro**, or executes an **EXITM** command, control returns to the next command in the calling **Macro**. Under some conditions (such as the user canceling by pressing the right mouse button at a prompt), you may want to end all **Macros** and return to the "**Command**:" prompt without returning to the calling **Macro**. To do this, use **EXITAM** (**Exit All Macros**).

Load Macros

LOAD MACROS clears any existing macro definitions, then loads a new macro table from a macro file. (This is faster than defining macros by using **EDIT MACRO**.)

To use, type a file name or select a file with the mouse. You need to select an existing macro file that ends in a *.mac* file name extension.

By default, when you start a work session, **FastCAD** looks for the file **FC7.MAC**, and automatically loads the file if it finds it in the current directory. **FastCAD** has provided this file to demonstrate some of the basic macro programming principles.

You can use the **LOADMACM** command to bypass the dialog box. This can be helpful if you are using a macro to load a new macro file. This action resets the variable table—**FastCAD** ignores everything in the current macro after it encounters **LOADMACM** <filename>.

Text equivalent: ***LOADMAC***

Edit Macros

When you select **EDIT MACROS**, **FastCAD** displays a text-editing window with which you can ***edit*** or ***modify*** the ***current macro table***. While typing, you can:

- Use the keyboard **ARROW** keys, **HOME**, and **END** to move the cursor within the editing window;
- Press **DEL** (**Delete**) to delete the character under the cursor;
- Press **BACKSPACE** to delete characters to the left of the cursor;
- Type special characters using the **ALT** key.
- Use Window's **Cut** (**^X**), **Copy** (**^C**), and **Paste** (**^V**) functions to transfer text in and out of a plain-text word processor, like **Notepad**.

The current macro table is the area in memory that **FastCAD** uses to store the active macro definitions. As such, if you make changes to the current macro table, they will be forgotten at the end of the session or when another macro file is loaded in its place. Use **SAVE MACROS** to make those changes permanent in a macro file.

Text equivalent: ***EDITMAC***

Save Macros

The **[Macros > Save Macros]** command saves all existing **Macro Definitions** to a **Macro** file for later loading with **[Macros > Load Macro]** command. Type a file name or select a file with the mouse. Use caution when ***saving*** over an existing **Macro** file. The best strategy is to ***create***, ***edit***, and ***test*** a set of **Macros** using small, individual **Macro** files. Do not save the **Macros** in one big **Macro** file until they are thoroughly debugged, and keep a backup copy of the **Macro** files in case you want to change the **Macros** later.

Text equivalent: ***SAVEMAC***

Clear Macros

The **[Macros > Clear Macros]** completely ***clears*** the ***Macro table***, eliminating all **Macro**, ***label***, ***function key,*** and ***variable definitions***. If your **Macros** do not work and you cannot figure out why, ***conflicting Macro*** and ***variable names*** are usually to blame. Use **CLEAR MACROS**, then reload the **Macros** ***one at a time*** from their individual files.

Text equivalent: ***CLEARM***

List Variables

The **[Macros > List Vatiables]** command displays the ***current variables*** and their associated values. Listed variables may have been processed directly from a **Macro** or from the keyboard. **Variables** persist until the current **FastCAD** session is closed, the **CLEAR MACROS** command is used, or **EDIT MACROS** is used and the **Macro** has been modified. The variable list is presented in a ***text window***.

Any time a **Macro** instruction is processed, the ***variable*** and its value is appropriately appended or updated in the list. In this manner, the ***table*** can be a valuable **Macro** debugging tool.

Text equivalent: ***LISTVARS***

Save Variables

The **[Macros > Save Variables]** command creates a script that may be run with **SCRIPT** or **SCRIPTM** to restore the values of the current ***variables***.

This command, and the ability to ***read saved variables***, is helpful in a number of ways. For instance, you can use **SAVEVARS** to ***save*** calculated or retrieved values that you can pass on to another ***Macro***, or even to another application. You use one **Macro** with different starting conditions, based upon user input, conditional tests, or other contexts. You can create a ***variable list*** with a text editor and ***save*** or ***edit*** it for use with your **Macros**.

Typically, you will run **SAVEVARS** from a **Macro**. The typical syntax is:

SAVEVARS <**filename**>

The *.scr* extension is automatically appended. The file is saved to the application path unless you specify otherwise.

To utilize the saved variables in a macro, use the **SCRIPTM** command:

SCRIPTM <**filename**>

Each line of the ***saved variable file*** is processed and loaded into memory. **Macros** can now utilize these **Macros** as if they were initialized internally.

To manually ***create a variable list***, use a ***text editor*** such as **NED4** by clicking **[File > Edit File]**. For each ***variable*** you want to ***define***, add a line using the following pattern:

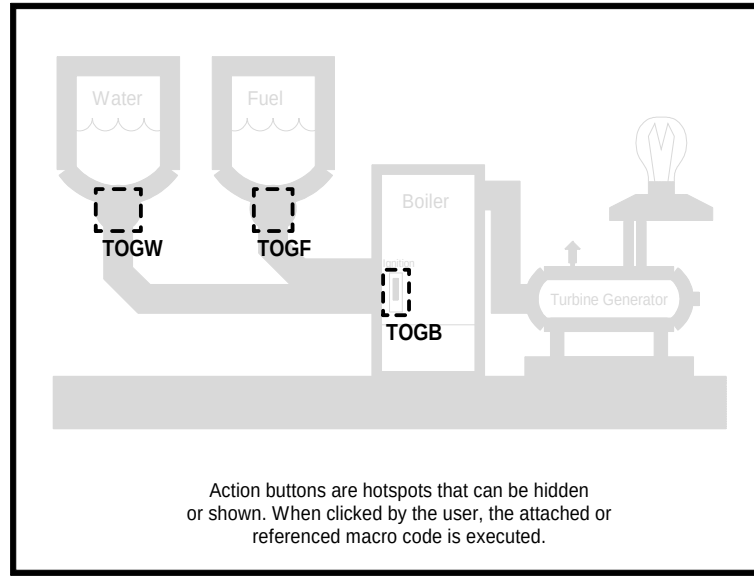
GL;<**variable name**>**;**<**variable value**>

You do not need to specify the ***variable type*** (such as ***integer***, ***word***, ***line***, etc.). **FastCAD** will determine the type based on the value assigned. Be sure to save the file as a ***regular text file*** with an **.SCR** extension.

Text equivalent: ***SAVEVARS***

Using Actions

ACTIONS allow you to make your drawings interactive. Like an Internet web page, you can define "**hotspots**" on your drawings that perform designated actions when you click on them with a mouse. These ***hot spots*** are called **ACTION BUTTONS**.



Action buttons are hotspots that can be hidden or shown. When clicked by the user, the attached or referenced macro code is executed.

The cursor changes to a pointer whenever it crosses an action hotspot.

An **ACTION BUTTON** is a ***hidden rectangular area*** in your drawing to which you attach a **FastCAD Macro**. When you ***activate*** the button by clicking it, **FastCAD** executes the designated **Macro**. **ACTIONS** are ideal for interactive tutorials, live presentations, and for creating active graphic models - drawings that do something instead of just sitting there.

To see how actions can be used, load the supplied drawing **Actdemo**.**FC7**. This drawing shows fluids moving through a system of pipes and valves. For each step in the actions sequence, layers are turned on and off.

A drawing with actions loads just like a regular drawing, except that its hotspots are always in an interactive state. (This behavior is different than in previous versions, where an action drawing had to be activated.)

Whenever you run a mouse over an action button, the cursor changes to a pointing cursor to let you know. To run the assigned function, just ***click on the hotspot*** (either a left-click or right-click works).

For example, move the mouse over the water valve. Note how the cursor arrow changes to the pointer hand. This indicates that you are over a hotspot. Click to open the valve. The previously empty pipe is now filled with water! Try the other hotspots and see if you can get light from the lamp. (Hint: the boiler needs water, fuel, and someone to hit the ignition switch.)

Define Action command

Use **ACTION** to create or add action buttons to a drawing.

To define an ***action*** *butto*n, select **[Macros > Define Action]** from the menu.

1. **FastCAD** opens an ***Edit Text*** *dialog box*. Type in a macro name or script text and click **OK**.

   The macro name or script you enter here will activate when the action button is clicked. If using a macro, the macro must be defined in a loaded .mac file. To define macros, see the section titled "**Adding New Action Macros**".

2. The prompt reads "**Hotspot window:**". Pick two corner points in a drawing window to define the size and shape of the action button.

   FastCAD draws an ***action*** as a *rectangular frame around its active area*, with the attached macro name displayed below it. The label is there so you can select the action for editing.

Once the action is tested, you will probably wish to make the button and text invisible. First you need to label the button for the user. There are two approaches:

- Add a box (polygon entity) and text to describe what the action does. This approach models a Windows-type button.
- Draw actual objects over the action button. This way, the user can activate the invisible button by selecting the visible object. For example, the user can select a door to open or close it, select a flower to make it bloom, or view a detail of a mechanical assembly by selecting the part to be magnified.

To make **action buttons visible**, use **ACTVIS**.

To ***hide action buttons***, use **ACTHIDE**.

Show Actions command

Use **[Macros > Show Actions]** to ***make action buttons visible***. **ACTION BUTTONS** are ***always active,*** even when they are ***hidden***. However, if you wish to ***edit*** or ***delete*** an action button, you need to first ***make it visible***.

[Macros > Show Actions] works with ***all action buttons in your drawing***. There is no way to select specific **ACTION BUTTONS** to be visible or hidden. You can simulate this behavior by isolating certain buttons on different layers, and then hide or show those layers as desired. Note that **ACTION BUTTONS** on ***hidden layers*** still respond to user clicks.

ACTION BUTTONS on ***hidden sheets*** are ***never visible*** or ***active***.

To make all action buttons visible, select **[Macros > Show Actions]** from the menu.

FastCAD redraws the drawing with all **ACTION BUTTONS** ***and their Macros visible***.

To ***hide all action buttons*** from view, use the **ACTHIDE** command **[Macros > Hide Actions]**.

TIP: You should draw entities on various LAYERS over the ACTION BUTTON areas to draw the users attention for specific HOT SPOTS. This could be a rectangle with TEXT inside that says 'Play Sound 1'.

Text equivalent: ***ACTVIS***

Hide Actions command

Use **[Macros > Hide Actions]** command ***makes action buttons and their Macros invisible*** to the user. ***Action*** *buttons* are always active, even when they are hidden. In most cases, you would hide the buttons from the user, using some other visual aid integral to the drawing to act as the hotspot indicator.

If you wish to edit or delete an action button, you need to first make it visible. To make action buttons visible, use the **ACTVIS** command **[Macros > Show Actions]**.

ACTHIDE works with all action buttons in your drawing. There is no way to select specific actions buttons to be visible or hidden. You can simulate this behavior by isolating certain buttons on different layers, and then hide or show those layers as desired. Note that action buttons on hidden layers still respond to user clicks.

ACTION BUTTONS on ***hidden sheets are never visible*** or ***active***.

To ***hide all*** ACTION BUTTONS select **[Macros > Hide Actions]** from the menu.

FastCAD redraws the drawing with all **ACTION BUTTONS** buttons ***hidden*** (but still active).

Text equivalent: ***ACTHIDE***

Chapter 22

Modifiers

Tools to help you draw with precision

Speed up with function keys

Many of the function keys have been preassigned to some of the modifiers using FastCAD's standard menu.

For instance, if you needed to use the EPT (endpoint) modifier, you could hit the F5 key instead of choosing [Mod > Endpoint] from the menu, or typing EPT at the command prompt.

To see how each Funtion key is assigned, just pull down the *Mod menu*.

The ***Mod*** *menu* includes several ***modifiers*** that lock to exact points on existing entities. The use of ***modifiers*** are essential to precision drawing. Without them, you would have difficulty selecting the exact **CENTER** of a ***circle***, or the **MIDPOINT** of a ***line***, for instance.

Modifiers often work in conjunction with another drawing or editing command. When such a command prompts you to select a point or entity, use ***modifiers*** to precisely specify and narrow the selection.

FastCAD's array of ***modifiers*** include the following:

- Endpoint
- Center
- Midpoint
- Nearest point on
- Intersection of
- Parallel to
- Perpendicular to
- Angle to
- Percent along
- Distance along
- Bearing to
- Tangent to
- Reference point
- Same X
- Same Y
- Like

Each specific modifier is described in more detail later in this section. But let's start with a general overview of how to use a modifier:

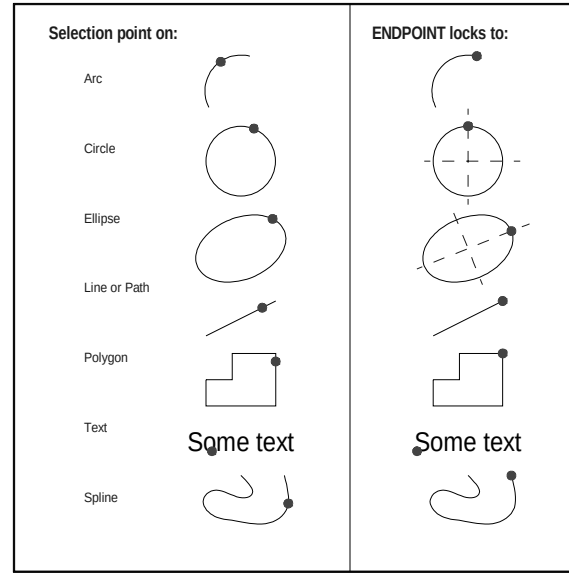
1. Activate a command (such as **LINE**) that prompts for a point.
2. Select a modifier to lock the first point to an existing entity. **FastCAD** asks what entity the modifier refers to.
3. Pick a point on the entity within the cursor's pick box. If **FastCAD** does not find an appropriate entity after using the left mouse button once, you must re-select the modifier again, then continue with the command.

Attach button

To specify a default, always ready-to-use modifier, use the **Attach mode**. Right-click the *Attach button* to specify a default modifier. Then make sure the *Attach button* is on (depressed). Now, whenever you are prompted for a point, that modifier is always active. It can be overridden by another modifier you select from the menu or type in.

Modifiers always override *Button bar* settings such as **Snap**, **Ortho**, or **Attach**.

Endpoint modifier

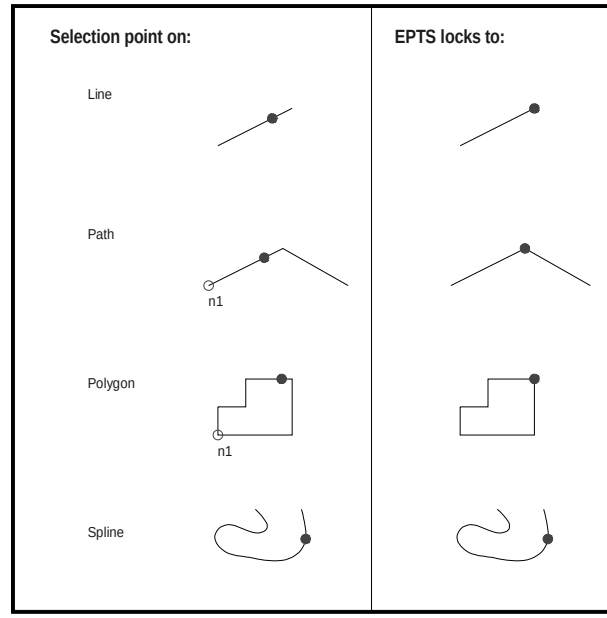


The **ENDPOINT** ***modifier*** locks to the **ENDPOINT** of certain entities. As with all modifiers, you must first select a drawing or editing command such as **LINE**. The prompt reads "**Endpoint os entity:**". Select a point on the desired entity. **FastCAD** will lock to the *nearest* endpoint. Please refer to the chart below for the effect of **ENDPOINT** on various types of entities.

Entity	***EPT locks on to***
Line *or* Line Arrow	Nearest end of line
Arc *or* Arc Arrow	Nearest end of arc
Dimension	End of leader closest to dimensioned entity
Polygon	First point on first node
Path *or* Path Arrow	End of total length neares pick point
Circle	Nearest (0°, 45°, 90°, 135°, 180°, 225°, 270°, 315°) point
Ellipse	Nearest end of either axis
Point	On the point
Smooth Polygon	First point on firstt node
Spline *or* Spline Arrow	First or last node, nearest to pick
Text	Nearest end of text, along baseline

Text equivalent: ***EPT***

Endpoint of seg modifier



Endpoint of Segment modifier icon

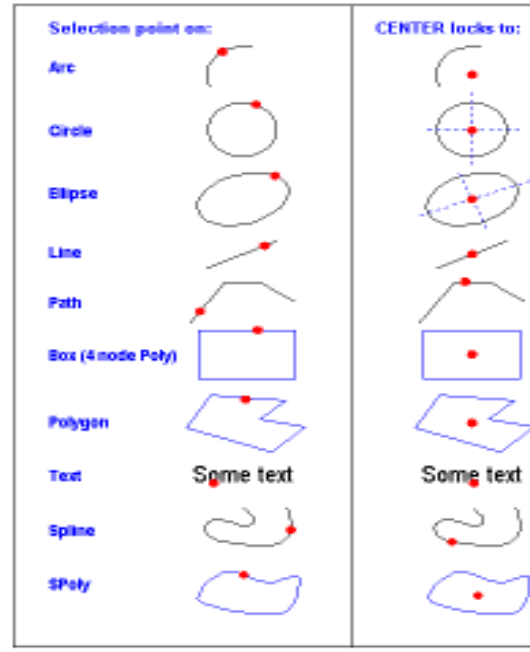
The **ENDPOINT OF SEG** *modifier* locks to ***segment endpoints***. It differs from the regular **ENDPOINT** *modifier* when you select ***paths***, ***polygons***, ***splines***, and ***SPolys***. When you use the regular **ENDPOINT** *modifier* on such entities, **FastCAD** always locks to the ***first or last points in the node list***. **ENDPOINT OF SEG** ***locks to the node nearest your pick point***. For **splines** and ***SPolys***, the *segment endpoint* is in part determined by the *spline resolutio*n set by the **SPLRESD** command.

As with all *modifiers*, you must first select a drawing or editing command such as **LINE**. The prompt reads "**Endpoint of segment:**" and a pick box cursor is displayed. Pick a point on the entity segment you want the endpoint of. Please refer to the chart below for the effect of **ENDPOINT OF SEG** on various types of entities.

Entity	***EPTS locks on to***
Line	Nearest endpoint point of segment
Arc *or* Arrow	Nearest end of arc
Dimension,	End of leader on dimensioned entity
Path *or* Arrow	Nearest end of segment nearest to pick
Polygon	Nearest end of segment nearest to pick
Circle	*Not applicable*
Ellipse	*Not applicable*
Point	On the point
Smooth Polygon	Node nearest to pick (see **SPLRESD**)
Text	*Not applicable*

Text equivalent: ***EPTS*** (***or press function key*** **F5)**

Center modifier



Center modifier icon

The **CENTER** *modifier* locks to the ***center*** of existing entities.

As with all modifiers, you must first select a drawing or editing command such as **LINE.**.

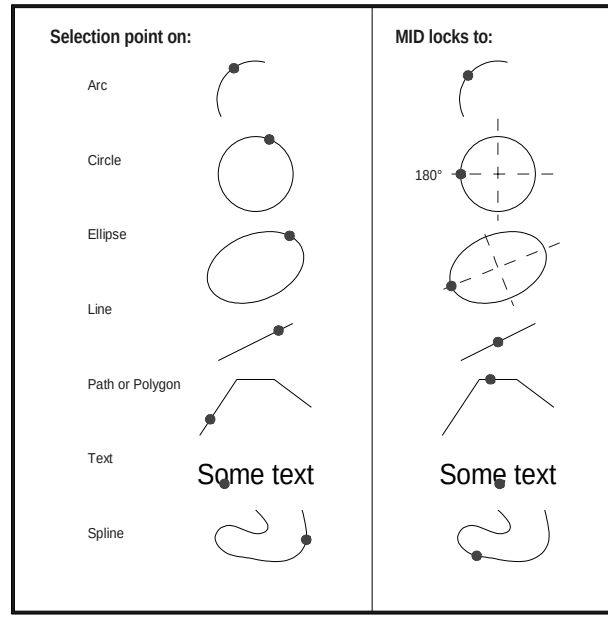
The prompt reads "**Center of entity**:". Select a point on the desired entity.

FastCAD locks to the entity's **CENTER**. Please refer to the chart below for the effect of the **CENTER** modifier on various types of entities.

Entity	***CEN locks on to***
Arc, Circle, or Ellipse	Center
Arrow, Line, Path, or Polygon	Middle of entity's geometry
Dimension	*Not applicable*
Point	On the point
Smooth Polygon or Spline	Middle of entity's geometry
Text	Middle of text, along baseline

Text equivalent: ***CEN*** **(*or press function key* F4)**

Midpoint modifier



The MIDPOINT *modifier* locks to the ***midpoint*** of certain entities.

As with all *modifiers*, you must first select a drawing or editing command such as **LINE.**.

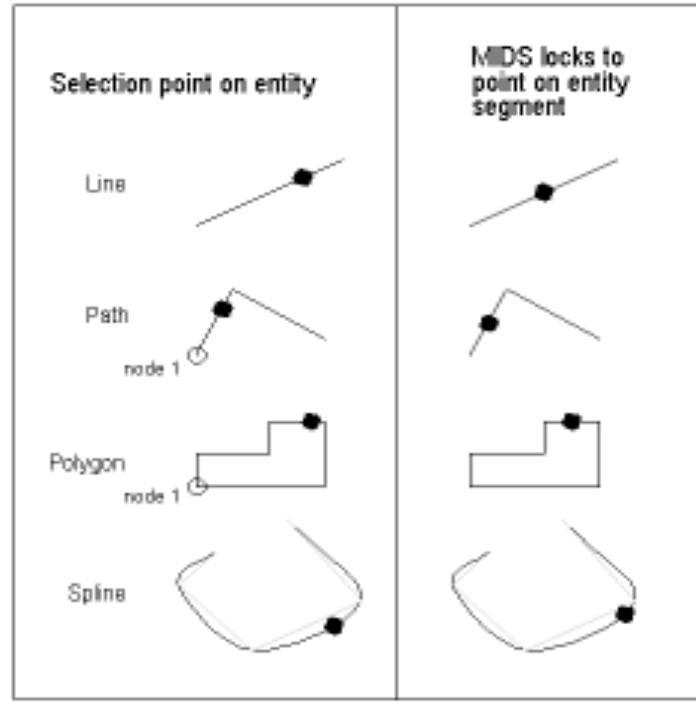
When you select **MID**, the prompt reads "**Middle of entity:**". Select a point on the desired entity.

FastCAD locks to the entity's **MIDPOINT**t. Please refer to the chart below for the effect of **MID** on various types of entities.

Entity	***MID locks on to***
Arc	Halfway along arc
Arrow or Line	Middle of picked line
Path or Polygon	Middle of total length (between first/last nodes)
Dimension	*Not applicable*
Circle	180° on circle
Ellipse	End of major axis (opposite from start)
Point	On the point
SPoly or Spline	Middle of total length (between first/last nodes)
Text	Middle of text, along baseline

Text equivalent: ***MID***

Midpoint of Seg modifier



Midpoint of Segment modifier icon

The MIDPOINT OF SEG *modifier* locks to the ***middle*** of certain entities ***segments*** such as ***polygon sides*** and ***path segments***..

As with all modifiers, you must first select a drawing or editing command such as **LINE**.

When you select **MIDPOINT OF SEG** *modifier* **[Mod > Midpoint of seg]** the prompt reads "**Middle of entity:**". Select a point on the desired entity.

FastCAD locks to the entity's ***segment*** **MIDPOINT**. Please refer to the chart above for the effect of **MIDPOINT OF SEG** on various types of entities.

Text equivalent: ***MIDS*** (***or press function key*** **F3)**

Mid of 2 EPTS modifier

The **MID OF 2 EPTS** *modifier* allows you to ***automayically*** use the **ENDPOINT OF SEG** *modifier* on two *selected entities* such as ***lines, splines*** or the ***segments of polygons*** or ***paths.***

As with all *modifiers*, you must first select a drawing or editing command such as **LINE.**.

1. When you select **the MID OF 2 EPTS *modifier* [Mod > Mid of 2 EPTS]**, the prompt reads "**Endpoint of segment:**".

2. Select a point on the ***1st entity*** and the prompt again reads "**Endpoint of segment:**"' Select a point on the ***2nd entity***..

FastCAD *starts the first point of the new* LINE *midway between these two segments* ENDPOINTS.

Text equivalent: ***MID2;EPTS;^D;EPTS;^D;***

Mid of 2 CEN modifier

The **MID OF 2 CEN** *modifier* allows you ***automayically*** use the **CENTER** *modifier* on two *selected entities* such as ***lines, circles, arcs, splines*** or the ***center of polygons*** and ***SPoly geometry.***

As with all *modifiers*, you must first select a drawing or editing command such as **LINE.**.

1. When you select **MID OF 2 CEN *modifier* [Mod > Mid of 2 CEN]**, the prompt reads "**Center of entity:**".

2. Select the***1st entity*** and the prompt again reads "**Center of entityt:**" Select the ***2nd entity***..

FastCAD *starts the first point of the new* LINE *midway between these two entities* CENTER *points*.

Text equivalent: ***MID2;CEN;^D;CEN;^D;***

Mid of 2 MIDS modifier

The **MID OF 2 MIDS** *modifier* allows you ***automayically*** use the **MIDPOINT OF SEG** *modifier* on two *selected entities* such as ***lines, splines*** or the ***segments of polygons*** or ***paths.***

As with all *modifiers*, you must first select a drawing or editing command such as **LINE.**.

1. When you select **MID OF 2 MIDS *modifier* [Mod > Mid of 2 MIDS]**, the prompt reads "**Middle of segment:**".

2. Select a point on the ***1st entity*** and the prompt again reads "**Middle of segment:**"' Select a point on the ***2nd entity***..

FastCAD *starts the first point of the new* LINE *midway between these two segments* MIDPOINTS.

Text equivalent: ***MID2;MIDS;^D;MIDS;^D;***

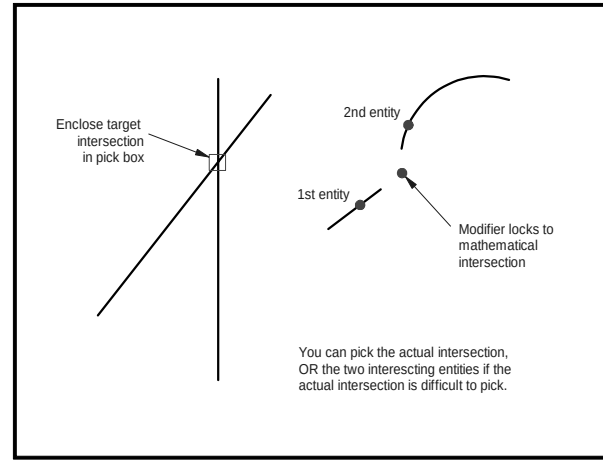
Nearest Point On modifier

The **NEAREST POINT ON** *modifier* locks to a point mathematically **ON** any entity selected within the cursor's pick box. As with all modifiers, you must first select a drawing or editing command such as LINE. The prompt reads "**Point on entity:**". Pick a point on the entity you want to lock on to. If you do not select an entity or valid point, **FastCAD** returns to the original prompt.

The **NEAREST POINT ON** modifier will supersede any **Snap** or **Attach** settings.

Text equivalent: ***ON*** **(*or press function key* F9)**

Intersection of modifier



Intersection modifier icon

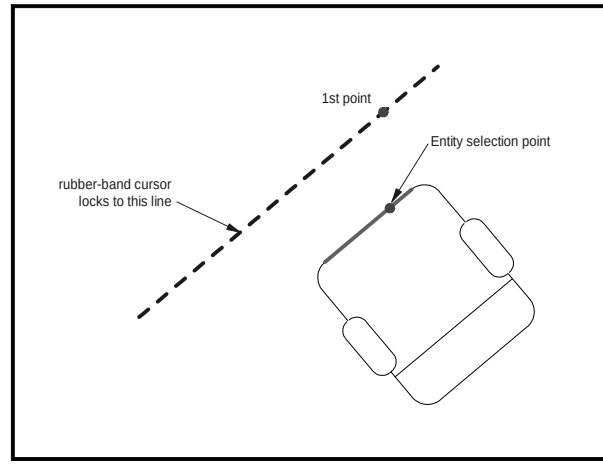
The **INTERSECTION OF** *modifier* locks to ***the point where two entities intersect***. As with all modifiers, you must first select a drawing or editing command such as **LINE**.

When you select **INTERSECTION OF** you are prompted for the "**Intersection point or 1st entity:**". If the pick box contains an intersection, it will automatically be selected. If you select an entity, but do not enclose an ***intersection*** in the pick box, you will see the "**2nd entity to intersect:**" prompt. Select a point on another entity and **FastCAD** will automatically find their (**nearest**) **INTERSECTION** ***point***..

Two selected entities don't actually have to cross. **FastCAD** will calculate the mathematical intersection of the entities.

Text equivalent: ***INT*** **(*or press function key* F6)**

Parallel To modifier



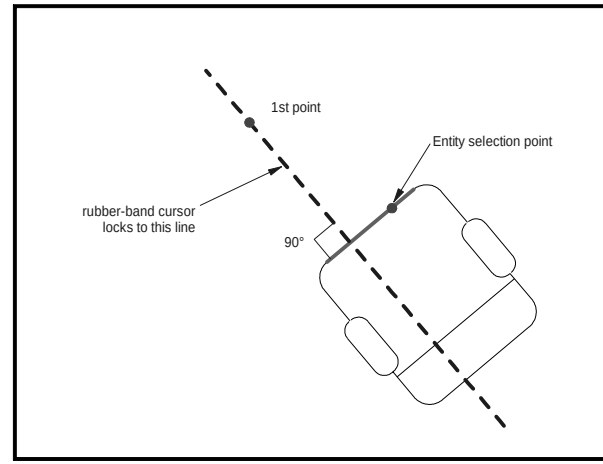
The **PAARALLEL TO** *modifier* locks the rubber-band cursor **PARALLEL** to an existing ***reference entity***. You can use **PAARALLEL TO** whenever **FastCAD** prompts you for a point. The ***reference entity*** can be a ***line***, ***text***, ***polygon side***, ***spline***, and so on. If you select a ***circular reference entity***, such as an ***arc***, ***circle***, or ***ellipse***, the direction is locked **PARALLEL** to the pick point's ***tangent***.

Usually you will place at least one point before selecting **PARALLEL TO**. The prompt reads "**Parallel to entity at:**".

1. Select a point on the reference entity. **FastCAD** returns to the command you were using, but now the rubber-band cursor locks to the same angle as the reference entity.
2. The prompt changes to "**Point along bearing line [distance]:**". Stretch the cursor in either direction until it looks the way you desire, then pick that point. To draw a line with an entered length (rather than one visually specified), type in a distance in drawing units. If the length is positive, the line is drawn towards the reference entity. If the length is negative, it draws away from the entity.

Text equivalent: ***PRL*** **(*or press function key* F11)**

Perpendicular To modifier



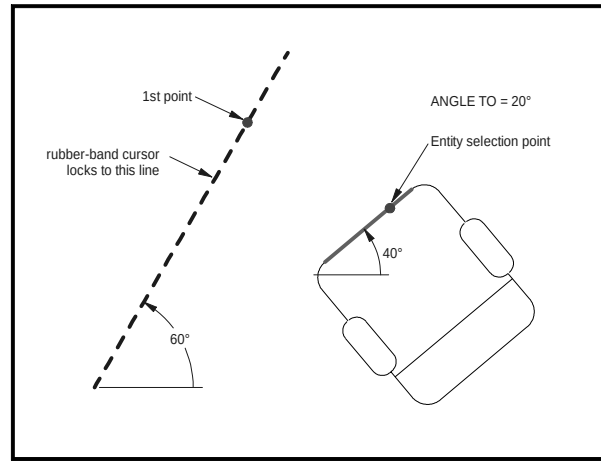
The **PERPENDICULAR TO** *modifier* locks the rubber-band cursor **PERPENDICULAR** to (**at right angles to**) an existing ***reference entity*** . You can use **PERPENDICULAR TO** whenever **FastCAD** prompts you for a point. The r***eference entity*** can be a ***line***, ***text***, ***polygon side***, ***spline***, and so on. If you select a ***circular reference entity***, such as an ***arc***, ***circle***, or ***ellipse***, the direction is locked **PERPENDICULAR** to the pick point's ***tangent***.

Usually you place at least one point before selecting **PERPENDICULAR TO** (or pressing **F12**). The prompt reads "**Perpendicular to entity at:**" and the cursor changes to a pick box.

1. Pick a point on the reference entity. **FastCAD** returns to the command you were using, but now the rubber-band cursor locks at the angle of the reference entity plus (or minus) **90°**.
2. The prompt changes to "**Point along bearing line [distance]:**". Stretch the cursor in either direction until it looks the way you desire, then pick that point. To draw a line with an entered length (rather than one visually specified), type in a distance in drawing units. If the length is positive, the line is drawn towards the reference entity. If the length is negative, it draws away from the entity.

Text equivalent: ***PRP*** **(*or press function key* F12)**

Angle To modifier



The **ANGLE TO** *modifier* locks the rubber-band cursor at an angle you specify to an existing reference entity. You can use **ANGLE TO** whenever **FastCAD** prompts you for a point. The ***reference entity*** can be a ***line***, ***text***, ***polygon side***, ***spline***, and so on. If you select a ***circular reference entity***, such as an ***arc***, ***circle***, or ***ellipse***, the direction is locked relative to the pick point's ***tangent***.

1. Usually you will place at least one point before selecting **ANG**. The prompt reads "**Degrees bearing:**". Type a positive or negative angle, which can include amounts like **11.05** or **22+1/2**.
2. Now the prompt reads "**At angle to entity at:**", and the cursor changes to a pick box. Pick a point on the reference entity. **FastCAD** adds the angle you entered to the angle of the reference entity, locking the rubber-band cursor to the resulting angle.
3. The prompt changes to "**Point along bearing line [distance]:**". Stretch the cursor in either direction until it looks the way you desire, then pick that point. To draw a line with an entered length (rather than one visually specified), type in a distance in drawing units. If the length is positive, the line is drawn towards the reference entity. If the length is negative, it draws away from the entity.

Text equivalent: ***ANG***

Percent Along modifier

The PERCENT ALONG *modifier* selects a point at a specified **PERCENTAGE** of the ***reference entity's length***. For most referenced entities, **PERCENT ALONG** treats the ***nearest endpoint from the selection point as its beginning***. For example, **75%** of a line is three-fourths of the way along its length, starting at the end nearest the selection point.

1. As with all modifiers, you must first select a drawing or editing command such as **LINE**. The prompt reads "**% along:**".
2. Enter a number. For example, to select a point **1/4** of the way along a line, type "**25**" and press ENTER. Now the prompt reads "**along entity:**".
3. Select a point on the entity you want a percentage of (remember that the selection point on the reference entity controls which end **FastCAD** starts from when calculating the percentage distance). You can use percentage values less than **0%** or more than **100%** on some entities. For example, you could use **125%** to select a point **1/4** of the way beyond the end of a line.

Entity	***% locks on to***
Arc,	% length, from nearest picked end.
Arrow, Dimension, or Line	% line length, from nearest picked end.
Path or Polygon	% overall length, from nearest picked end.
Circle	% circumference length from 0°, from direction closest to pick.
Ellipse, Elliptical Arc	*Not applicable.*
Point	On the point.
SPoly or Spline	% length along curve, from nearest picked end. (values <0% or >100% act like 0%)
Text	*Not applicable.*

Text equivalent: ***%***

Percent Along seg modifier

The **PERCENT ALONG SEGMENT** *modifier* selects a point at a s***pecified percentage of the reference entity's segment length***. For most referenced entities, **%S** treats the ***nearest endpoint from the selection point as its beginning***. For example, **75%** of a line is three-fourths of the way along its length, starting at the end nearest the selection point.

This command differs from the regular **PERCENT ALONG** *modifier* when certain entities are picked. **%** considers an entire **path**, **poly**, or **spline** ***length*** when determining ***distance along***, whereas **%S** considers only the selected ***path***, ***poly***, or ***spline segment***.

1. As with all modifiers, you must first select a drawing or editing command such as **LINE**. When prompted for a point, select **[Mod > Percent Along seg]**.
2. The prompt reads "**% along:**". Enter a number between **0** and **100**. For example, to select a point **1/4** of the way along a line, type **25** and press **ENTER**.
3. Now the prompt reads "**along segment:**", and the cursor changes to a pick box. Pick a point on the entity you want a percentage of (remember that the selection point on the reference entity controls which end **FastCAD** starts from when calculating the percentage distance).

FastCAD snaps the point as specified.

Note: On some entities, you can use percentage values less than **0%** or more than **100%**. For example, you could use **125%** to select a point **1/4** of the way beyond the end of a line.

Entity	%S locks on to
Arc	% length, from nearest picked end.
Arrow, Dimension, or Line	% line length, from nearest picked end.
Path or Polygon	% segment length, from nearest picked node.
Circle	% circumference length from 0°, from direction closest to pick.
Ellipse, Elliptical Arc	*Not applicable.*
Point	On the point.
SPoly or Spline	% segment length, from nearest picked node. (values <0% or >100% act like 0%)
Text	*Not applicable.*

Text equivalent: ***%S* (*or press function key* F7)**

Distance Along modifier

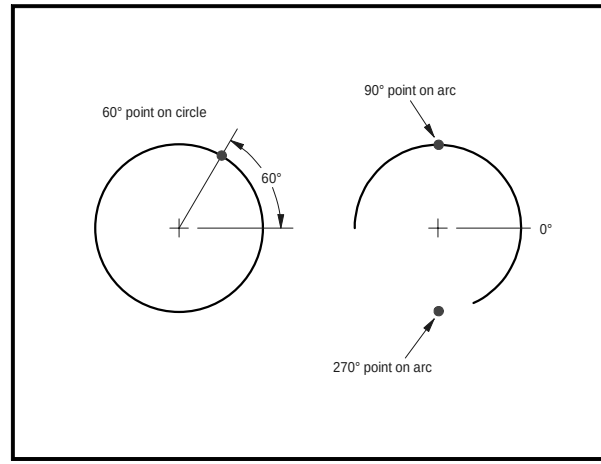
The **DISTANCE ALONG** *modifier* selects a point at a specified **DISTAAANCE** of the ***referenced entity's length***. For most referenced entities, **DISTANCE ALONG** treats the nearest endpoint from the selection point as its beginning.

1. As with all modifiers, you must first select a drawing or editing command such as **LINE**. To invoke the **DISTANCE ALONG** *modifier*, select **[Mod > Distance Along]**, or press **F8**.
2. The prompt reads "**Distance:**". Enter a ***length*** or ***distance*** value. For example, to select a point 1.5 inches of the way along a 10-inch line, type 1.5 and press enter. Now
3. The prompt reads "**Along entity:**" and the cursor changes to a pick box. Pick a reference entity. Where you select the point determines which end **FastCAD** starts from when calculating the distance. You can use distance values less than 0 on some entities. For example, you could use -2 to specify a point 2 units beyond the end of a line.

Entity	**DIST** locks to
Arc	Distance along circumference, from nearest end
Arrow, Dimension, or Line	Distance from nearest end point
Path or Polygon	Distance (overall) from nearest start/end node
Circle	Distance along circumference, counter-clockwise from 0° end
Ellipse, Elliptical Arc	*Not applicable*
Point	On the point
SPoly or Spline	Distance along curve, from nearest start/end node (values <0 act like 0)
Text	*Not applicable*

Text equivalent: ***DIST* (*or press function key* F8)**

Degrees on modifier



The **DEGREES ON** *modifier* selects a point at a specified ***number of*** **DEGREES***s* ***around an arc***, ***circle***, ***ellipse***, or ***elliptical arc***. **Angles are measured counterclockwise** from **zero degrees at 3 o'clock.**

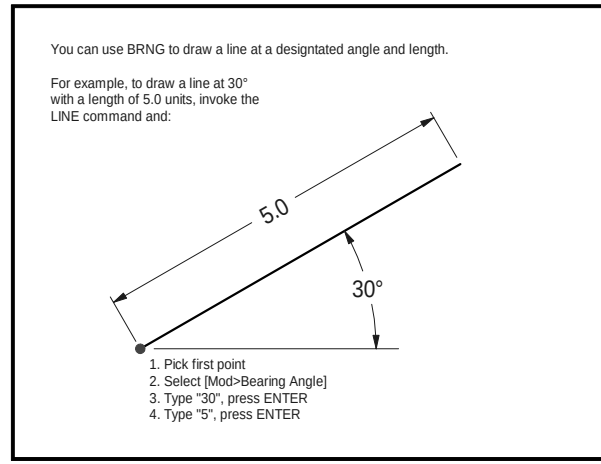
First select a drawing or editing command such as **LINE**. When prompted for a point, invoke the **DEGREES ON** *modifier*.

1. The prompt reads "**Degrees bearing:**". Enter an angle value. For example, to select the 1 o'clock point on a circle, type "60" and press **ENTER**.
2. The prompt reads "**Degrees relative to entity:**" and the cursor changes to a pick box. Pick a point on the reference entity.

Note: On an ***ellipse*** or ***elliptical arc***, **DEGREES ON** selects points at a true bearing angle from the ***center*** of the ***ellipse***. For example, points 10° apart around an ellipse are *not* evenly spaced on its circumference.

Text equivalent: ***DEG***

Bearing Angle modifier



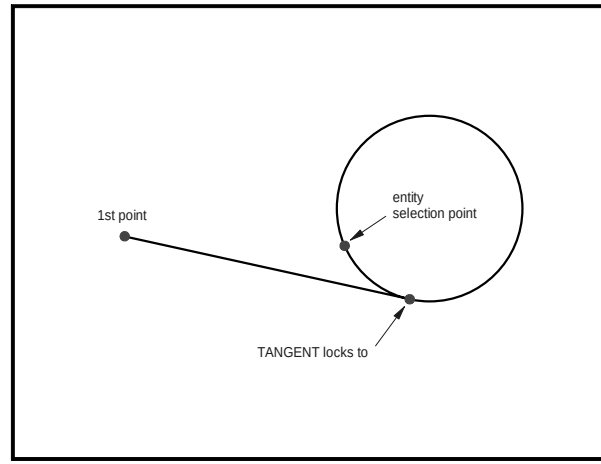
The **BEARING ANGLE** *modifier* locks the rubber-band cursor to a **BEARING ANGLE** that you specify. You can use **BEARING ANGLE** whenever **FastCAD** prompts you for a point. Unlike **PARALLEL TO**, **PERPENDICULAR TO**, and **ANGLE TO**, this *modifier* doesn't ask for a **reference entity-the angle is measured counter-clockwise from 0° at 3 o'clock** (**horizontal**). The **BEARING ANGLE** *modifier* works a little like ***polar coordinates***, except that here you may specify distance visually instead of typing both an angle and distance.

Usually you will place at least one point before selecting **BEARING ANGLE**.

1. The prompt reads "**Degrees bearing:**". Type a positive or negative angle, which can include amounts like **11.05** or **22+1/2**. **FastCAD** returns to the command you were using, but now the rubber-band cursor locks to the angle you specified.
2. The prompt changes to "**Point along bearing line [distance]:**". Stretch the cursor in either direction until it looks the way you want, then select the next point for your command. You can choose to enter a specified length. **FastCAD** them snaps to the specified distance in the specified bearing.

Text equivalent: ***BRNG***

Tangent to modifier



The **TANGENT TO** *modifier* selects a point on a ***circle***, ***arc***, ***ellipse***, ***spline***, or ***smooth polygon*** that forms a **TANGENT** from the last point selected. You can use **TANGENT TO** whenever **FastCAD** prompts you for a point.

Usually you will place at least one point before selecting **TANGENT TO**. First select a drawing or editing command. When prompted for a point, invoke the **TANGENT TO** modifier.

. The prompt reads "**Tangent to entity:**" and the cursor changes to a pick box. Select an entity near the desired point of tangency. **FastCAD** draws the line or line segment tangent to that entity.

Text equivalent: ***TAN***

Change @ Ref Point modifier

FastCAD stores the coordinates of the last point used in drawing or editing an entity - for example, the end of a line. **FastCAD** uses this stored point as a reference point for the **SAME X** (**vertical-lock**) and **SAME Y** (**horizontal-lock**) modifiers and to calculate the three forms of relative coordinates. You can use the **CHANGE @ REF POINT** modifier to change the *reference point* to *any other point*.

First, select a drawing or editing command, then select **CHANGE @ REF POINT**.

The prompt reads "**@ Reference point:**". Specify a new point by picking a point on the screen, or by typing numeric coordinates. **CHANGE @ REF POINT** is especially effective used in combination with other modifiers to set the *reference point* to the *endpoint*, *center*, etc., of an existing entity.

Text equivalent: ***REF***

Same X modifier

The **SAME X** *modifier* is a special *modifier* that ***aligns points*** **VERTICALLY** with the ***reference point*** by keeping the **x coordinate** ***the same as the last point***. (It's like a one-time **ORTHO** lock.) First select a drawing or editing command, specify the first point in the command, and then invoke the **SAME X** *modifier*.

. **FastCAD**'s prompt doesn't change. When you specify the next point, **FastCAD** locks the point to the nearest point that lines up vertically with the last point used in drawing or editing.

For an example of **SAME X** used in drawing a line, start the **LINE** command, and then specify the first point.

1. When **FastCAD** prompts for "**Next point:**", select **SAME X** from the Mod menu.
2. The prompt still reads "**Next point:**". Using the mouse, pick a point on the screen. **FastCAD** draws the line from the first point to a point directly above or below the first point (depending on where you picked). In other words, **FastCAD** gives the new point the same x coordinate as the first point, while using the **y coordinate** you picked with the cursor.

You can combine **SAME X** with **CHANGE @ REF POINT** to align a new entity with an existing entity. For example, to align a ***new circle*** with the ***center*** of an *existing* ***circle***: Select C**IRCLE:> RADIUS & CENTER [Draw > Circles > Radius and Center]**. When **FastCAD** asks for a *center point*, select **CHANGE @ REF POINT** from the ***Mod*** *menu*. Then select **CEENTER** from the ***Mod*** *menu*, and pick an existing **circle**. **FastCAD** resets the reference point to the ***center*** of the ***circle***. The prompt reads "**Center point:**" again. Select **SAME X** from the ***Mod*** *menu*, then select a point below the ***old circle***. **FastCAD** draws the ***new*** **circle** with its ***center*** perfectly aligned with the ***center*** of the ***old circle***.

Text equivalent: ***SAMEX***

Same Y modifier

The **SAME Y** *modifier* is a special *modifier* that ***aligns points*** **HORIZONTALLY** with a ***reference point*** by keeping the **y coordinate** ***the same as the last point***. (It's like a one-time **ORTHO** lock.) First select a drawing or editing command, specify the first point in the command, and then invoke the **SAME Y** modifier.

. **FastCAD**'s prompt doesn't change. When you specify the ***next point***, **FastCAD** locks the point to the ***nearest point that lines up horizontally with the last point used*** in ***drawing*** or ***editing***.

Text equivalent: ***SAMEY***

Same Z modifier

The **SAME Z** *modifier* is a special *modifier* that aligns points with the **z-axis** direction of a ***reference point*** by keeping the ***z coordinate the same as the last point***. First select a drawing or editing command such as **LINE**, specify the ***first point*** in the command, and then invoke the **SAME Z** *modifier*.

FastCAD's prompt doesn't change. When you specify the ***next point***, **FastCAD** locks the point to the ***nearest point aligned with the z-axis of the last point used*** in ***drawing*** or ***editing***

Text equivalent: ***SAMEZ***

Like modifier

Learn to Like Like and Keep Like

LIKE is an often underutilized modifier that can speed up your work considerably.

KEEP LIKE can quickly reset all your current drawing properties to that of an existing entity you select. Such properties include color, layer, line style, fill style, line width, pen thickness and text and dim styles if applicable. If you use many different properties in your drawing, KEEP LIKE can do in two steps what would otherwise take dozens!

The LIKE modifier isn't quite so powerful. But its use still precludes many dialog box selections you might otherwise make.

Because the LIKE modifier snares the property values from the entity, you don't need to know what those values are beforehand. That might save you a few [Info > List] calls.

The **LIKE** *modifier* allows you to ***select an entity and use all of its properties*** (***layer***, ***color***, ***line style***, ***fill style, line width***, ***pen thickness***, ***dimension style etc.***) ***and apply them to the next entity you will draw.*** **FastCAD's** ***default properties settings are*** **NOT** ***effected .*** This is a very quick way of ***using all the properties of an existing entity at once and applying them to only the next entity you draw***. This bypasses the need to reset each property from various dialog boxs such as ***Text Properties, Layer Management*** etc. for just drawing one entity and then having to reset them back for the next new entities you will draw. If you are using the **CHANGE** command to modify only one property of one or more entities such as ***color, pen thickness, layer*** etc.the **LIKE** *modifier* is a real tim saver.

For example, **LIKE** can help you ***change the color of selected entities to be the same color as an existing entity:***

1. Select the **CHANGE COLOR** command **[Edit > Change > Color]** to select the entities whose ***color*** you wish to modify.
2. After choosing **OK-Do it**, the prompt reads "**Color value [dialog]:**".
3. Select **[Mod > Like]** (or **press F2**).
4. The prompt reads "**Like entity:**".
5. Select an entity that has the ***desired color***. All selected entities will now have the desired color.

Another example is to move all entities that share a common layer:

1. Select the **MOVE** command and select by ***Layer***. The prompt reads "**Layer name or # [dialog]:**".
2. Select **[Mod > Like]** (or **press F2**).
3. The prompt reads "**Like entity**:".
4. Select an entity that you know is on the ***desired layer***. All entities that share that ***layer name*** will be selected for the **MOVE** command.

If you wish the ***properties of the selected entity to persist***, use the **KEEP LIKE** command *(see below)* instead.

Text equivalent: ***LIKE*** **(*or press function key* F2)**

Keep Like modifier

Keep Like Icon

The **[Mod > Keep Like]** command allows you to ***select an entity and use all of its properties*** (***layer***, ***color***, ***line style***, ***fill style, line width***, ***pen thickness***, ***dimension style etc.***) ***and apply them to all new entities you will draw. These properties reset all of the previous default property settings .*** This is a very quick way of resetting all properties at once without going to each dialog box such as ***Text Properties, Layer Management*** etc to do so.

KEEP LIKE makes current (for all new entities to be drawn) the following entity properties: ***to***

- **Color**
- **Fill (2nd) color**
- **Pen thickness**
- **Layer**
- **Line style**
- **Fill style**
- **Pen Thickness**
- **Line width**
- **Text properties**
- **Dimension properties**

For instance, if you wish to draw all new entities that will have the same ***properties as an existing entity*** (but different from your current settings),

1. Select the **KEEP LIKE** command **[Mod > Keep Line]**.

The prompt reads "**Keep properties of entity:**".

2. Select the existing entity that uses the properties you want the ***new entities to use***.

3. Select the **LINE** command and draw a **LINE.** The new line, as well as all new entities, will draw with the ***same properties as the entity you previously selected***.

NOTE: You may use the **KEEP LIKE** command in the ***middle*** of a **DRAW** command. Say you selected the **BOX** command and placed its first corner, you can go to the ***Mod*** *menu.*and select **KEEP LIKE,** pick an existing entity then place the opposite corner of the **BOX**. The new **BOX** will be the same ***layer***, ***color***, ***line style***, ***fill style, line width***, ***pen thickness***, ***dimension style*** as the selected entity and all of the ***default properties*** will have changed. You can see them update in the various ***property status windows*** on the ***Tool*** *bar.*

If you only want the ***next entity you are about to draw to use the properties*** of an existing entity, use the **LIKE** *modifier after you have selected a* **DRAW** ro **EDIT** command**. FastCAD's *default properties will* NOT *change*** when using the **LIKE** *modifier.*

Text equivalent: ***KEEP***

Chapter 23

Tablet Commands

Using your digitizing tablet with FastCAD v7

Tablets are also referred to as Digitizers, Digitizing Tablets, or Graphics Tablets.

Although we designed **FastCAD v7**'s user interface for a mouse, you can use a graphics tablet instead. Tablets let you trace existing drawings or maps by taping them to the surface of the board and digitizing points using the tablet puck. You can also set aside defined portions of the screen to activate certain commands or macros you often use.

In order to use your digitizing tablet, you must first install a **WinTab** driver, **version 1.0** or later. The **WinTab** driver should be included with your tablet. If not, or if your tablet is an older model, try contacting the **tablet manufacturer**. If the manufacturer has an Internet web site, the **WinTab** driver should be available there for immediate download.

When **FastCAD** detects a properly installed **WinTab** driver, the ***Tablet*** *menu* appears (before the ***Help*** *menu* entries). This menu provides you with four commands specific to tablets:

- **Tablet On**
- **Tablet Off**
- **Configure**
- **Calibrate**

Before you ***setup*** and ***calibrate*** your ***digitizer***, you can use **FastCAD** to draw an overlay that you can tape to the tablet showing the ***mouse***, ***menu***, and ***real*** (**tracing**) areas.

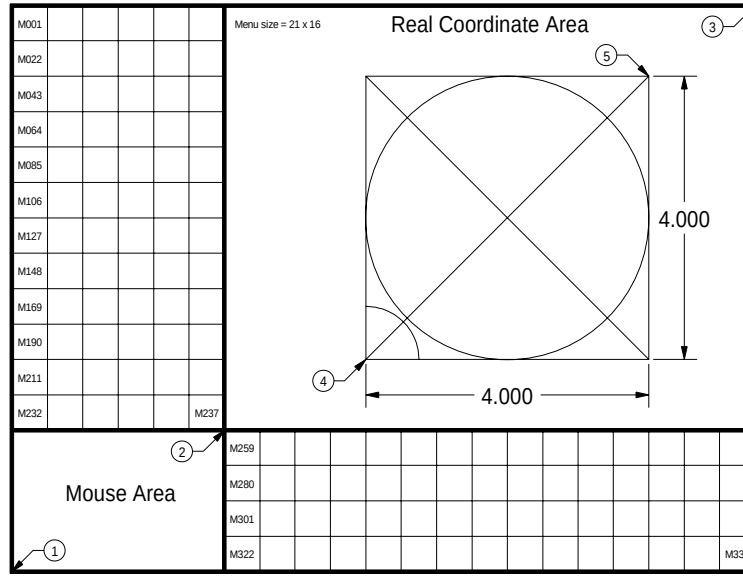
For a step-by-step introduction to tablet setup and calibration, see ***Using a Digitizing Tablet with*** **FastCAD.**

Using a Digitizing Tablet with FastCAD

First, connect the tablet to your computer and install its **WINTAB** driver. These drivers are now available from most tablet manufacturers.

If one did not come with your tablet, check the manufacturer's web site for one.

Once the **WINTAB** driver is installed and correctly reads your tablet, load FastCAD. Just to the left of the ***Help menu***, you will now see a ***Tablet menu***, containing 4 items: **Tablet On**, **Tablet Off**, **Configure**, and **Calibrate**.



Print out the sample drawing ***tablet.fc7*** at 1x scale and use it for your initial testing. Fix it to the surface of your tablet so that it will not move.

Configuring the Tablet

In **FastCAD**, select the **TABCFG** command **[Tablet > Configure]**. This allocates tablet space to three different areas:

- The ***mouse screen area***, in which your tablet will emulate a mouse and move the cursor across the computer screen.
- The ***menu area***, which allows you to easily select macros from a visual tablet layout.
- The ***real digitizing area***, in which you identify coordinates.

You are asked to pick first the lower left, then the upper right location of each screen area.

1. For the ***Mouse Screen Area***, pick points (1) then (2).
2. For the ***Real Area***, pick points (2) then (3).
3. For the ***Menu Area***, pick points (1) then (3).
4. Next you are asked to enter the ***number of rows and columns*** for the menu area.

The ***Real area*** and the mouse ***Screen area*** overlay the ***Menu area***. To allow multiple menu areas, define the menu area as including everything, and enter the number of evenly spaced columns and rows to cover the entire area. The sample ***tablet.fc7*** uses 21 columns by 16 rows.

This completes the ***Configure*** command. The settings are saved from one work session to the next, along with the most recent calibration (see below). The tablet may now be used for screen and menu mode, as well as for cursor button macros, by selecting the **TABLET ON** command. We will first calibrate our tablet's real area.

You only need to reconfigure your tablet when the location on the surface of the tablet of any the three areas changes.

Calibrating the Tablet

The **[Tablet > Calibrate]** command asks you to digitize two known locations in the real area. The sample overlay drawing ***tablet.fcw*** shows a square (from the sample drawing *square.fcw*). Load the drawing square.fcw now.

Select the **TABCAL** command from the tablet menu **[Tablet > Calibrate].**

1. Digitize point (4), then enter its location from the keyboard: 3,1.
2. Digitize point (5), then enter its location from the keyboard: 7,5.

This completes the ***tablet calibration*** for this drawing.

Using the Tablet

Now enter the **TABON** command from the ***Tablet menu*** **[Tablet > Tablet On]**. When you select any command that asks for **2D coordinates**, the tablet real area becomes active— you can move the tablet cursor, and the screen cursor shows the current real position.

Enter the **LINE** command, then move the tablet cursor over the circle center—the screen cursor should also be over the circle center. Press the left (or 0) button on your tablet cursor to select the point. Move the tablet cursor over some other feature on the drawing and complete the line. Press the next button (RIGHT or 1) or the keyboard **ENTER** key to complete the line command.

You need to recalibrate the tablet anytime you wish to change its ***Real*** coordinate location, or when you have changed your drawing UNITS setting. Whenever you place new work on your tablet from which to digitize, you will need to recalibrate.

Pick the two known points so they are not close on the surface of the tablet, and not orthogonal (horizontal or vertical) to each other. Points far apart and nearly a 45° bearing to each other will give the best results.

A left (0) button click in the menu area will generate a macro call of the form **M001** through **M999**. The sample *tablet.fcw* shows the number of several locations. The upper-left item is M001, and they increase going across (then the next row down, and the one after, in order). Note that numbers are assigned without taking into account the real area or screen area overlays.

Extra buttons on the tablet cursor generate macros of the form **T02** through **T15**. Note that these buttons do **NOT** work while the tablet cursor is in the mouse screen area—only the **LEFT** and **RIGHT** (0 and 1) buttons work there.

When you are not using the tablet, be sure to turn it off with the **TABOFF** command **[Tablet > Tablet Off]**, and if possible, remove the cursor from the tablet surface. Many tablets generate samples at a very high data rate, which can slow down throughput on your system. Also, when the cursor is on the tablet, **WINTAB** has it and the system mouse pointer interacting, which can lead to strange results when using programs such as text editors that are unaware of the tablet.

Tablet On

The **TABLET ON** command activates an installed digitizing tablet for immediate use in **FastCAD**.

Before using the **TABLET ON** command, the digitizer should be properly configured and calibrated.

To de-activate a tablet, use the **TABLET OFF** command.

When you are not using the tablet, be sure to turn it off with the **TABOFF** command **[Tablet > Tablet Off]**, and if possible, remove the cursor from the tablet surface. Many tablets generate samples at a very high data rate, which can slow down throughput on your system. Also, when the cursor is on the tablet, **WINTAB** has it and the system mouse pointer interacting, which can lead to strange results when using programs such as text editors that are unaware of the tablet.

Text equivalent: ***TABON***

Tablet Off

The **TABLET OFF** command de-activates an installed digitizing tablet. Use of this command does not un-install the tablet; it simply toggles off **FastCAD**'s recognition of the device, reverting screen pointer control exclusively back to the mouse.

To reactivate a tablet, use the **TABLET ON** command.

Text equivalent: ***TABOFF***

Tablet Configure

The **TABCFG** command allows you to allocate the tablet space to three different areas:

- A ***mouse screen area***, in which you use the tablet puck or stylus like a mouse to select menu items, icons, and screen locations for drawing.
- A ***real digitizing area***, for tracing existing drawings or maps.
- A ***menu area***, a rectangular array of cells that you can program like function keys.

To configure your tablet, select **[Tablet > Configure]**:

You are asked to pick first the lower left, then the upper right location of each screen area.

1. Define the ***Mouse Screen Area***, picking two defining points.

   FastCAD scales the pointer area to the aspect ratio of the screen, so select an area similar in shape to your monitor. Drawing and pointing will be easier if you use a small area—try 4 by 3 inches or less.

2. Define the ***Real Area***, picking two defining points.
3. Define the ***Menu Area***, picking two defining points.
4. Enter the number of rows and columns for the menu area.

The real area and the mouse screen area overlay the menu area. To allow multiple menu areas, define the menu area as including everything, and enter the number of evenly spaced columns and rows to cover the entire area.

Each cell on the tablet is automatically defined as a macro named **M001** through **M999,** starting at the upper-left, and increasing left to right, then top to bottom. **Macros** functions are defined in the *.mac* file.

Configuration settings are saved from one work session to the next, along with the most recent calibration.

You only need to reconfigure your tablet when the location on the surface of the tablet of any the three areas changes.

Text equivalent: ***TABCFG***

Tablet Calibrate

The **TABCAL** command asks you to digitize two known locations in the ***real area.*** The two selected points tell **FastCAD** how to scale the coordinate system in the real digitizing area.

To **calibrate** your tablet, select **[Tablet > Calibrate]**:

1. Select a first reference point on the tablet's ***real digitizing area***, then enter its (**x,y**) coordinates via the keyboard.
2. Select a second reference point on the tablet's real digitizing area, then enter its (x,y) coordinates via the keyboard.

Pick two known points that are not close on the surface of the tablet, and not orthogonal (horizontal or vertical) to each other. Points far apart and nearly a 45° bearing to each other will give the best results.

You need to re-calibrate the tablet each time you trace something that is scaled differently from your current configuration. You do not need to reconfigure unless you change the size of the real digitizing area.

Text equivalent: ***TABCAL***

Chapter 24

Help Commands

Instant on-line help is a mouse click away

Help menu

FastCAD provides comprehensive on-line help through the ***Help*** *menu*. The on-line help is handy when you need instruction or clarification during a command. This feature is called ***context sensitivity***. For instance, if you are ***dimensioning an angle*** and need assistance, you can hit the **F1 key** ***while in the command*** to automatically display the **DIMA** help topic. Note that the term "**on-line**", in this sense, refers to help topic availabity while working with the program, *not* transmission via the Internet.

New help files are provided along with each relevant program update. The on-line help file also contains advanced information not found in this printed volume, such as macro programming commands, and conventions governing customization of menus, icon bars, etc.

Help

Use the **HELP** command to display the contents/index window for **FastCAD**'s **on-line** help. You can open the topic directly, or search for it in the on-line index. Access **HELP** by pressing **F1** while **FastCAD**'s prompt is in the "**Command**:" state, or by selecting **[Help > Contents]** from the ***Help*** *menu*.

Unlike most keyboard function keys, **F1** ***cannot be redefined***. Windows always reserves **F1** for ***on-line help system***s. (***F10 is the other keyboard function key that cannot be redefined; Windows reserves F10 for menu focus.***)

Quick Help is essentially identical to **Help**.

Text equivalent: ***HELP (or press function key F1)***
Text equivalent: ***QHELP***

Current Command Help

Use **HELPCON** in mid-command to activate context sensitive help for that command. You can also invoke this command by **pressing the F1 key** during a command.

For instance, run the **LINE** command **[Draw > Line]** command. When the prompt reads "**1st point:**", **press** the **F1 key** or choose **[Help > Current Command]** from the ***Help*** *menu*. The on-line help topic for the **LINE** command is displayed.

Text equivalent: ***HELPCON***

HelpHelp

This is the Windows “How to use Help” topic. If you have any trouble navigating or using the on-line help program, try this command.

Text equivalent: ***HELPHELP***

About FastCAD

The **ABOUT** command displays the **FastCAD** “**About...**” message box. Important information such as the exact ***version number***, ***release date***, and ***serial number*** can be found here.

Text equivalent: ***ABOUT*** *or* ***VER***

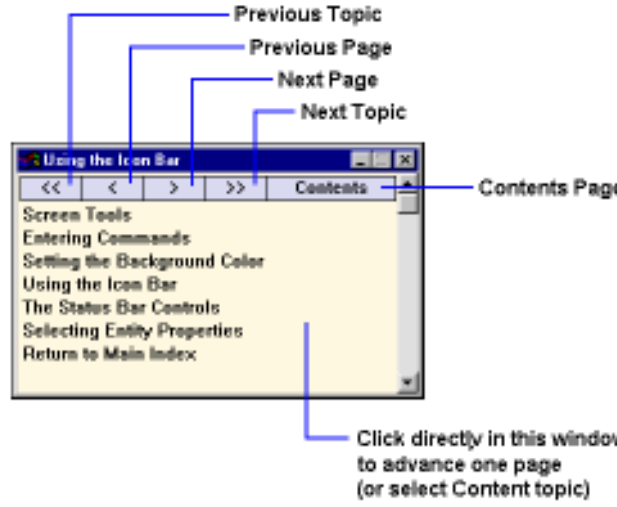
Webinfo

WEBINFO provides information on the **Evolution Computing website**, as well as on option to link directly if you are Internet capable. Access this command by using **[Help > About our website...]**.

Text equivalent: ***WEBINFO***

Guided Tour

FastCAD Guided Tours are interactive, message box-driven tutorial systems. If you’re new to **FastCAD**, these guides offer a useful introduction to basic concepts.



Text equivalent: ***GUIDE***

Appendix

FastCAD V7 Command List

Alphabetical list of FastCAD's command set

%	Percent Along modifier
%S	Percent Along Segment modifier (**F7**)
ABOUT	Display Version & Registration Information
ABOVE	Move entity to display after another
ACTHIDE	Hide Action buttons from user
ACTION	Define an Action button
ACTVIS	Show Action buttons
ADDSHT	Add Sheet
AGRID	Align Grid
AHEAD	Add Arrowhead to existing entity
ALOOP	Animate Path Loop
ALOOPM	Animate Path Loop - name from command line
ANG	Angle to modifier
AONCE	Animate path once
AONCEM	Animate Path Once - name from command line
APOV	Animation Point of View - define animation path
ARC3	Start, Middle and End Arc, draw
ARC3D	Create 3d Arc
ARCA	Arc w/ Arrowhead to end, draw
ARCAS	Arc w/ Arrowhead from start, draw
ARCB	Start, End and Bulge Arc, draw
ARCDA	Arc w/ Double Arrowheads, draw
ARCR	Center and Radius Arc, draw
ARCS	Center, Start, End Arc, draw
ARCW	Wedge (multipoly), draw
AREA	Area display, calc
ASAVE	AutoSave options
ASTYLE	Arrowhead Style
ATCHMODE	Select Attach mode
ATCHOFF	Turn Attach mode **Off**
ATCHON	Turn Attach mode **On**
ATCHT	Toggle Attach mode **On/Off**
ATCHTOG	Toggle Attach mode **On/Off**
ATTRIB	Define Symbol Attribute
BACK	Draw entity behind others
BEARING	Bearing display, calc

BELOW	Move entity to display before another
BEZ	Bezier curve, draw
BKGND	Background color
BOOKMARK	Edit bookmarks
BOX	Box, draw
BOX3	3d Box
BREAK	Break entity
BRNG	Bearing modifier
CALC	Calculate
CARCD	Chained Arcs, draw
CARY	Circular Array, draw
CEN	Center modifier (**F4**)
CHAM	Chamfer two entities, draw
CHAMFER	Chamfer two entities, draw
CHANGE	Change properties, display dialog
CHANGEAS	Change Arrowhead Style
CHANGEC	Color, Change
CHANGEC2	Secondary color, Change
CHANGED	Dimension Style, Change
CHANGEFS	Fill Style, Change
CHANGEL	Layer, Change
CHANGEM	Material, Change
CHANGELS	Line Style, Change
CHANGELW	Line Width, Change
CHANGEPT	Pen thickness, Change
CHANGESC	Changes all symbol definition colors to current
CHANGESCBB	Changes all symbol definition colors to color by block
CHANGESLYR	Change all symbol definitions entities to a new Layer
CHANGET	Text Properties, Change
CHANGETX	Text, Change (same as **CHANGEX**)
CHANGEX	Text, Change (same as **CHANGETX**)
CIR2	2 Point Circle, draw
CIR3	3 Point Circle, draw
CIR3D	3d Circle
CIRD	Diameter and Center Circle, draw
CIRP	Center and Point On Circle, draw
CIRR	Radius and Center Circle, draw
CIRTAN2	Circle Tangent to 2 Circles, draw
CIRTAN3	Circle Tangent to 3 Circles, draw
CLEARM	Clear macros from memory
CLIPCOPY	Copy to Clipboard (**^C**)
CLIPCUT	Cut to Clipboard (**^X**)
CLIPOPT	Options for Clipboard
CLONESYM	Clone Symbol Definition
CLOSE	Close Drawing Window
COLOR	Set Current Color
COLORDLG	Set Current Color with dialog
CONNECT	Connect entity
CONNECTC	Connect Copy entity

CONVF	Convert Multiple Files
CONNECTCPY	Connect Copy entity
COPY	Copy entity
COPYSHT	Copy entities to another sheet
COUNT	Count selected entities
COUNTA	Count All entities
CPART	Create Part
CPARY	Circular Part Array
CPARYM	Circular Part Array - macro version
CSNAPOFF	Turn Cursor Snap **Off**
CSNAPON	Turn Cursor Snap **On**
CSNAPTOG	Toggle Cursor Snap **On/Off**
CSURF	Create Coons Surface
CTRLP	Create Control Points for SmartSym
CYL	Create Cylinder
DBLN	Chain of Parallel Lines, draw
DEFSYM	Define a Symbol
DEG	Degree modifier
DELNODE	Delete Node
DELNV	Delete Named Vied
DELSHT	Delete Sheet
DELSYM	Delete Symbol Definition
DPTHOFF	Depth Mode **Off**
DPTHON	Depth Mode **On**
DPTHTOG	Depth Mode Toggle **On/Off**
DIMA	Angular Dimension
DIMAQ	Quick Angular Dimension
DIMB	Baseline Dimension
DIMD	Diameter Dimension
DIMH	Horizontal Dimension
DIML	Linear Dimension
DIMO	Ordinate Dimension
DIMP	Parallel Dimension
DIMR	Radius Dimension
DIMV	Vertical Dimension
DISCARD	Discard current drawing
DIST	Distance Along modifier (**F8**) (in mod context)
DIST	Distance display, calc (in command context)
DIST2	Expanded distance display, calc
DISTANCE	Distance display, calc
DOME	Create Dome
DONUT	Create Donut (multipoly)
DRAG	Drag (move while displaying dynamic cursor)
DRAGCPY	Drag copy
DSTRETCH	Dynamic Stretch
DSTYLES	Dimension Style dialog box
DWGCMAP	Use drawing's color map
DYNEDIT	Dynamic Edit entity
DYNPAN	Dynamic Pan

DYNROT	Dynamic Rotate
DYNROTC	Dynamic Rotate Copy
DYNROTCPY	Dynamic Rotate Copy
DYNSCL	Dynamic Scale
DNSCLC	Dynamic Scale Copy
DYNSCLCPY	Dynamic Scale Copy
ECOFF	Echo Off (macro stream)
ECON	Echo On (live input)
EDIT	Edit entity
EDITMAC	Edit Macro file
EDSYMDEF	Edit Symbol Definition
EGRID	Edit Grid
ELIPA	Elliptical Arc, draw
ELIPC	Ellipse, Circle at Angle to Window, draw
ELIPI	Ellipse, Center and Inclination, draw
ELIPP	Ellipse, Center and Axis Points, draw
END	Exit FastCAD
EPT	Endpoint modifier
EPTS	Endpoint of Segment (**F5**)
ERASE	Erase entity
EXIT	Exit FastCAD
EXITAM	Exit All Nested Macros
EXITM	Exit Current Macro
EXPLODE	Explode entity
EXPSYM	Explode symbol reference
EXTENTS	List drawing extents
EXTRACT	Extract Symbol Attributes
EXTRUDE	Extrude Profile in 3d
FIL	Fillet two entities
FILLET	Fillet two entities
FLIP	Flip between last UNDO and REDO
FONT	Select Current Font
FONTDLG	Select Current Font
FRAMEOFF	Turn Frame display Off
FRAMEON	Turn Frame display On
FRAMETOG	Toggle Frame display visibility (^F)
FREEZE	Freeze layer
FREEZEA	Freeze All layers
FRONT	Front entity
FSTYLE	Fill Style
FSTYLEDLG	Fill Style dialog box
GA	Get bearing angle
GD	Get distance
GE	Get Point on Entity
GL	Get Line (CR ends)
GN	Get Number (integer)
GO	Go to Macro Label

GOTO	Go to Macro Label
GP	Get Point (**2d/3d**)
GRIDANG	Rotate angular grid
GRIDOFF	Turn Grid display **Off**
GRIDON	Turn Grid display **On**
GRIDT	Toggle Grid display **On/Off**
GRIDTOG	Toggle Grid display **On/Off**
GRIDV	Specify square Grid Value
GROUP	Group entity
GUIDE	On-line Guide (tutorial)
GUIDEM	On-Line Guide from Macro
GV	Get real value
GW	Get Word (space/tab/CR ends)
HELP	Contents of Help
HELPCON	Current command Help
HELPHELP	How to Use Help
HELPN	Get Help by Context Number
HIDE	Hide layer
HIDEA	Hide All layers
HIDESHT	Hide Sheet
HORZ	SameY modifier
ID	Coordinate display, calc
IFDEF	IF (Variable defined) Go
IFERR	IF (Error) Go
IFN	IF (Negative Value) Go
IFP	IF (Positive Value) Go
IFZ	IF (Zero Value Go
INDEX	Make Fast Search Index
INSNODE	Insert Node
INSSYM	Draw Symbol
INSSYMD	Draw Symbol
INT	Intersection of modifier (**F6**)
INT2	Intersection (force 2 picks for macros)
KEEP	Keep Like
KEEPTHA	Keep text height & angle
LAYER	Select Layer
LAYERCPY	Copy entity to Layer
LAYERDLG	Select Layer dialog box
LDCFG	Load a Custom .CFG screen configuration file [Macro]
LDSYMDEF	Load part as symbol definition
LENGTH	Length along modifier
LIGHTS	Specify Lighting
LIKE	Like modifier (**F2**), Like command
LINE	2d Line
LINE2	2d Line
LINE3	3d Line

LINEA	Arrow Line
LINEDA	Double arrow line
LIST	List entity
LISTLYR	List layers
LISTSYM	List Symbol Definitions
LISTVARS	List Macro Variables
LNPRP	Perpendicular modifier
LNPRPL	Perpendicular at length modifier
LOAD	Open drawing file (**^O**)
LOADCMAP	Load saved .cwm color map file
LOADM	Load (name from cmd line)
LOADMAC	Load Macro file
LOADMACM	Load Macro File (name from cmd line)
LOCKOFF	Turn Group Locking **Off**
LOCKON	Turn Group Locking **On**
LOCKT	Toggle Group Locking **On/Off**
LOCKTOG	Toggle Group Locking **On/Off**
LSTYLE	Line Style
LSTYLEDLG	Line Style dialog box
LWIDTH	Line Width
LWIDTHD	Line Width dialog box
MATRL	Specify Default Material
MATRLSCL	Scale Material Texture
MATRLSCLA	Scale All Material Textures
MCOLOR	Change marking (entity selection) color
MDCTL	Multi-Drawing Control
MENU	Load Menu file
MENUD	Specify Default Menu
MENUDM	Specify Default Menu (from cmd line)
MID	Midpoint modifier
MID2	Midpoint between 2 points
MIDS	Midpoint of Segment (**F3**)
MIRCPY	Mirrored Copies
MIRROR	Mirror entity
MOVE	Move entity
MOVESHT	Move entities to another sheet
MPOLY	Create 2d Multipoly
MPOLY2	Create 2dMultipoly
MPOLY3	Create 3d Multipoly
MSURF	Create Mesh Surface
NED	Launch NED Text Editor
NEW	New drawing (**^N**)
NEWW	New Drawing Window
NOTES	File Notes
OFFSET	Offset a chain of entities
OFFSET1	Offset a single entity
OFFSETCPY	Offset a chain of entities, copy properties

ON	On modifier (**F9**)
OPEN	Open drawing file (**^O**)
OPENBC	Open current bookmarked drawing
OPENBN	Open next bookmarked drawing
OPENBP	Open prior bookmarked drawing
OPENDOC	Open File with Associated Program
OPENND	Open next drawing in file history
OPENOVL	Open Overlay File
OPENPD	Open prior drawing in file history
OPENREF	Open Reference File
OPENSEP	Open Separate File
OPTIONS	Select Program Options
ORIGIN	Change Origin
ORTHOFF	Turn Ortho **Off**
ORTHON	Turn Ortho **On**
ORTHT	Toggle Ortho **On/Off**
ORTHTOG	Toggle Ortho **On/Off**
PAN	Pan drawing view
PART	Insert Part
PARTM	Insert Parm (cmd line)
PARY	Part Array
PARYM	Part Array (cmd line)
PASTE	Paste from Clipboard (**^V**)
PATH	Create Path
PATH3	Create Path3
PATHA	Path w/ Arrowhead, draw
PATHDA	Path w/ Double Arrowheads, draw
PCHANGE	Change Properties Dialog
PEDIT	Properties Edit
PITCH	Rotate View Vertically
PICT	Insert bitmap image into drawing
POINT	Create Point
POLY	Create 2d Polygon
POLY3	Create 3d Polygon
PRINT	Print current drawing (**^P**)
PRINTSU	Print Setup
PRL	Parallel to modifier (**F11**)
PROTECT	Prevent files from being modified and/or printed
PRP	Perpendicular to modifier (**F12**)
PSPLINE	Create 2d Parabolic Spline
PSPLINE3	Create 3d Parabolic Spline
PSPOLY	Create 2d Parabolic SPoly
PSPOLY3	Create 3d Parabolic SPoly
PTHICK	Set pen thickness
PURGESYM	Purge unused symbol definitions
PURGEUNDO	Remove current undo information
QUIT	Exit **FastCAD**

RDOFF	Redraw **Off**
RDON	Redraw **On**
REDO	Redo last UNDO (**^Y**)
REDOA	Redo all UNDO's
REDRAW	Redraw the screen
REF	Reference Point modifier
RENDXF	Render Export to File
RENSYM	Rename symbol reference
REPEAT	Rectangular Array
REPSYM	Replace Symbol
RESTORE	Restore drawing file
RMODE	Render Mode # (1, 2, 3, 4, 5, 6, 7, 8, 9)
ROLL	Spin Current View
ROOM	Draw room (wall network)
ROTALN	Rotate and align
ROTATE	Rotate entity
ROTCPY	Rotated Copies
RPOLY	Create Regular Polygon
RSURF	Create Ruled Surface
RUN	Run Actions
RUNAPP	Run external application
SAMEX	Same X modifier
SAMEY	Same Y modifier
SAMEZ	Same Z modifier
SAVE	Save Current drawing (**^S**)
SAVEALL	Save All
SAVEAS	Save As... (**^A**)
SAVECMAP	Save current color map to .cwm file
SAVEMAC	Save Macro File
SAVEMACM	Save Macro File (cmd line)
SAVEVARS	Save macro variables
SCALE	Scale entity
SCALEXY	2d Differential Scale
SCALEXYZ	3d Differential Scale
SCLCPY	Scaled Copies
SCRIPT	Load Script file
SCRIPTM	Load Script file (cmd line)
SEARCH	Search drawing files for specified text
SELAW	Select Active Window
SELRENDER	Select Renderer
SELSHT	Select Sheet
SETLAYER	Set Current Layer
SGRID	Select Grid dialog box
SHEET	Manage sheets
SHOW	Show layer
SHOWA	Show All layers
SHOWSHT	Show Sheet
SNAPOFF	Turn Snap **Off**
SNAPON	Turn Snap **On**

SNAPT	Toggle Snap **On/Off**
SNAPTOG	Toggle Snap **On/Off**
SNAPV	Specify Snap Value
SOLID	Create 3d Solid
SOUND	Plays a standard .WAV sounf file [Macro]
SPLINE	Create 2d Cubic B-Spline
SPLINE3	Create 3d Cubic B-Spline
SPLINEA	Spline w/ Arrowhead, draw
SPLINEDA	Spline w/ Double Arrowheads, draw
SPLIT	Split entity
SPLRES	Change Spline Resolution
SPLRESD	Specify Default Spline Resolution
SPOLY	Create 2d Cubic B-SPoly
SPSEG	Create 3d Spherical Segment
SREV	Create 3d Surface of Revolution
SPOLY3	Create 3d Cubic B-SPoly
STRETCH	Stretch entity
SVNAME	Set view name
SYMBOL	Draw Symbol Reference
SYMBOLD	Draw Symbol Reference
TABCAL	Calibrate an installed digitizing tablet
TABCFG	Configure an installed digitizing tablet
TABLE	Make a TABLE of Rows and Columns of Text
TABOFF	De-activate an installed digitizing tablet
TABON	Activate an installed digitizing tablet
TAN	Tangent to modifier
TANGENT	Tangent Line to two entities, draw
TCHAM	Chamfer & Trim
TDATE	Inserts the Current Date as a piece of Text
TDFMT	Opens the Time and Date Format dialog box
TEMPLATE	Load new template
TEMPLATE	Load new template (cmd line)
TEXPORT	Export the data in a TABLE to .CSV format
TEXT	Text, draw
TFIL	Fillet & Trim
TFILE	File Text, draw
TFILENAME	Inserts the Current drawing file name as a piece of Text
THAW	Thaw layer
THAWA	Thaw All layers
TIMPORT	Import .CSV data from a file into a TABLE
TOOLS	Display Tools dialog box
TPRINTSU	Setup Text Printing Options
TPRO	Create 3d Tabulated Profile
TRIM	Trim entity
TRIMIE	Trim inside entities
TRIMINT	Trim to intersection
TRIML	Trim to specified length
TRIMOE	Trim outside entities
TRIMTO	Trim to specified entity

TSPEC	Text Properties dialog box
TTIME	Inserts the Current Time as a piece of Text
UNDO	Undo (**^Z**)
UNGROUP	Ungroup entities
UNITS	Units dialog box
VER	About **FastCAD**...
VSET	View Sets
VNAMES	View Names Dialog
WALL	Wall, draw
WCENTER	Set Walls Specified by Center Mode
WCWIDA	Change wall width
WCWIDE	Change wall end width
WCWIDS	Change wall span width
WEBINFO	Go to our web site
WLADD	Add wall
WLCUT	Cut wall
WLEFT	Set Walls Specified by Left Side Mode
WLOFST	Offset wall
WLRMV	Remove wall
WLWID	Set wall width
WPDEF	Define Workplane
WRIGHT	Set Walls Specified by Right Side Mode
XHTOG	Toggle crosshairs display
XHOFF	Turn Crosshairs Off
XHON	Turn Crosshairs On
XPINFO	Get Info on Installed XP Modules
XREF	Insert eXterneal REFerence
XREFDLG	External Reference Management Dialog
YAW	Rotate View Sideways
ZBACK	Zoom Backward (3d out from screen)
ZCEN	Zoom Center view
ZDOWN	Zoom (pan) Down
ZEXT	Zoom Extents view
ZFWD	Zoom Forward (3d into screen)
ZHGT	Zoom Height view
ZIN	Zoom In view
ZLAST	Zoom Last view
ZLEFT	Zoom (pan) view left
ZLOOK	Zoom 3d to Look From/At
ZNAME	Zoom Named View
ZOUT	Zoom Out view
ZRIGHT	Zoom (pan) Right
ZSPEC	Zoom to dialog-entered view specs
ZSTD	Chznge to 3 standard 3d views
ZTEXT	Zoom Text

ZTEXTD	Zoom Text (dialog)
ZWID	Zoom Width view
ZUP	Zoom (pan) Up
ZWIN	Zoom Window view
ZXLEFT	Zoom 3d to Left-Side Extents
ZXOBLL	Zoom 3d Left Oblique
ZXOBLR	Zoom 3d Right Oblique
ZXRIGHT	Zoom 3d Right Extents
ZXTOP	Zoom 3d Top Extents

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